Experimental and Numerical Investigation of the Intermittent Heating Performance Improvement of Novel Flat-heat-pipe Terminal

Yifan Wu¹,², Hongli Sun²,³, Mengfan Duan¹,², Borong Lin¹,²,*, Zixu Yang¹,², Hengxin Zhao¹,²
¹ Department of Building Science, Tsinghua University, Beijing 100084, China
² Key Laboratory of Eco Planning & Green Building, Ministry of Education, Tsinghua University
³ College of Architecture & Environment, Sichuan University, Sichuan 610065, China
*Corresponding email: linbr@tsinghua.edu.cn
Corresponding phone: (+86) 010 62785691

Abstract
Intermittent heating has the potential to achieve energy conservation in the temporal dimension, making it a subject of significant interest. However, existing heating terminals struggle to meet the demands of intermittent heating, presenting a challenge in balancing speed, comfort, and energy conservation. This study proposes a novel flat heat pipe terminal for intermittent heating and presents relevant experimental data and environmental simulation analysis. Results demonstrate that the flat heat pipe can provide a uniform surface temperature, with a maximum temperature difference of only 2.2°C, while providing excellent thermal comfort in radiation mode. To increase heating capacity, a convection mode is incorporated into the novel flat heat pipe terminal through forced convection, resulting in a fourfold increase. The most suitable intermittent heating strategy is found to involve convection preceding radiation, as it can effectively satisfy demands for speed, comfort, and energy conservation of intermittent heating.

Highlights
- A novel terminal was proposed for intermittent heating based on the flat heat pipe as a radiation surface.
- Basic performance experiments and environmental simulations were conducted.
- Convection mode and radiation mode were analyzed through simulations.
- A regulation strategy for intermittent heating was proposed.

Introduction
In China, the carbon emissions from building operations account for approximately 20% of the total emissions (Building Energy Saving Research Center of Tsinghua University, 2022). Meeting the increasing demand for indoor environmental quality while achieving carbon peak and carbon neutrality goals is a key challenge facing the building industry in the future. For non-central heating areas, intermittent heating based on occupants' behavior has high potential for energy savings from a temporal dimension.

Residential buildings in China's Hot Summer and Cold Winter Areas are typical application scenarios for intermittent heating. With high population density and economic level, the annual average growth rate of heating energy consumption per household in this region has exceeded 50% (Wang, 2022). From an objective perspective, the indoor-outdoor temperature difference in this region during winter is relatively small but fluctuates significantly, and the building envelope's air tightness and insulation performance are poor (Zhao, 2013). From the occupants' subjective perspective, the local residents have strong heat adaptation ability and significant habits of opening windows for ventilation (Duan, 2021). Therefore, if local residents can be guided to adopt flexible intermittent heating methods, there is great potential for energy savings.

The heating terminal is crucial to achieving intermittent heating. Only when the heating terminal has a fast thermal response and can ensure good thermal comfort, will users adjust the operation of the heating terminal based on the occupancy status of the room, thereby achieving energy-saving effects through intermittent operation. However, existing heating terminals have difficulty in balancing intermittent operation with thermal comfort. Forced convection terminals (such as fan coils and room air conditioners) have fast start-up and strong heating capacity, but commonly have issues with large vertical temperature differences and draughts (Yang, 2022). Natural convection terminals (such as radiators) generally have slow thermal response. Traditional radiant terminals (such as floor heating) are comfortable without draughts and have small vertical temperature differences (Zeiler, 2009), but they have a key problem of large thermal inertia and slow thermal response, making them unsuitable for intermittent heating modes (Tang, 2017).

Therefore, this paper proposes a novel heating terminal based on flat heat pipes, aiming to reduce the thermal inertia of radiant heating terminals through the phase-change heat transfer of flat heat pipes and investigate the feasibility of utilizing flat heat pipes for intermittent heating. Subsequently, experimental studies combined with computational fluid dynamics (CFD) simulations are conducted to compare the convection mode and radiation mode of heating terminals, and to determine the suitable control strategy for intermittent heating.

Methods
Working principle of flat heat pipes for heating
The heat pipe is a high-performance heat transfer device that utilizes phase change. It has been widely used in various fields of heat transfer in recent years. Among
them, the flat heat pipe is known for its flat and thin structure, which enables rapid and uniform heat transfer. The use of flat heat pipes for phase change heat transfer enables heating terminals to have lower thermal inertia and better uniformity, making them more suitable for meeting the rapid heating demands of intermittent heating. The flat heat pipe consists of multiple microchannels filled with a certain amount of working fluid, such as acetone. The microchannels are kept at a certain vacuum. When a local area of the flat heat pipe is subjected to a sufficiently large heat flux density, the liquid working fluid inside the microchannels evaporates and absorbs heat. It then condenses and releases heat when it encounters a relatively low-temperature area. This forms a closed cycle process, which is facilitated by the combined action of gravitational and capillary forces. Based on this principle, a heat source can be arranged on the lower side of the flat heat pipe, such as supplying hot water. The liquid working fluid inside the lower side of the flat heat pipe evaporates and absorbs heat from the hot water. This enables a quick realization of the phase change heat transfer process, as it condenses on the upper surface of the flat heat pipe and transfers heat to the room. The working principle of the flat heat pipe for heating is illustrated in Figure 1.

![Figure 1: Working principle of flat heat pipes for heating.](image)

**Experimental system**

This study conducted heating performance experiments on the terminal based on the flat heat pipe, with a focus on the heating capacity of the terminal and the temperature uniformity of the flat heat pipe as a phase change radiation surface. To this end, the measurement points were arranged, and the measurement parameters were obtained as follows:

1. Thermocouples (measurement accuracy ±0.3°C) were arranged horizontally and vertically on the surface of the terminal of the flat heat pipe to obtain the actual surface temperature distribution.
2. Armored thermal resistance (measurement accuracy ±0.1°C) were arranged at the inlet and outlet of the water supply pipe, and were connected to a Coriolis flowmeter.
(measurement accuracy ±0.005 kg/h) to ensure the accuracy of the measurement results.

Based on this, the heating capacity of the terminal and the surface heat transfer coefficient can be calculated by Equations (1) and (2):

\[
Q = cm(T_i - T_a) \quad (1) \\
Q = KA(T_s - T_a) \quad (2)
\]

Where \(Q\) is the heating capacity of the terminal, \(W\); \(c\) is the specific heat of water, \(4,200 \text{ J kg}^{-1} \text{ K}^{-1}\); \(m\) is the mass flow rate of water, in \(\text{kg s}^{-1}\); \(T_i\) and \(T_a\) are the inlet and outlet water temperatures of the terminal, °C; \(K\) is the surface heat transfer coefficient of the flat heat pipe, \(\text{W m}^{-2} \text{ K}^{-1}\); \(A\) is the surface area of the flat heat pipe, \(\text{m}^2\); \(T_s\) and \(T_a\) are the average surface temperature and ambient temperature of the flat heat pipe, °C.

Based on the study of the heating performance of the terminal, we further conducted experimental measurements and simulation analysis of the indoor environmental effect of the terminal. Referring to a small-sized room and an independent office, we constructed a real experimental room with dimensions of \(3 \times 3 \times 3\) m, and a south-facing window with dimensions of \(2 \times 2\) m was installed, equipped with external shading curtains to minimize the influence of irregular solar radiation on indoor temperature during the experiment.

Referring to the design standards for low heating load areas in China (the Hot Summer and Cold Winter Areas), the thermal parameters of the building envelope were as follows: exterior wall (K = 0.9 W m\(^{-2}\) K\(^{-1}\), C = 1.05 kJ kg\(^{-1}\) K\(^{-1}\)), roof (K = 0.5 W m\(^{-2}\) K\(^{-1}\), C = 0.75 kJ kg\(^{-1}\) K\(^{-1}\)), and window (K = 2.8 W m\(^{-2}\) K\(^{-1}\)). The air exchange rate in the experimental room was approximately 0.5 h\(^{-1}\). Air temperature measurement points were arranged in the experimental room to obtain the indoor temperature distribution as comprehensively as possible.

**Numerical setup**

Based on the experimental measurements, we further conducted a CFD simulation study. In addition to the surface temperature measurement points on the flat heat pipe, a thermocouple was installed at the air outlet of the terminal, and armored thermal resistances were arranged on the inner wall surfaces of the room.

Using the above-mentioned measurement points, the air outlet temperature and surface temperature of the terminal, and room wall surface temperature were measured to determine the first category boundary conditions for the CFD simulation of the indoor environment. These parameters were used as input to obtain the temperature and velocity fields of the indoor environment in radiation mode, convection mode, and different air velocities at air outlet through CFD simulation.

Considering the actual situation, the following settings were made during the simulation. Boussinesq assumption was chosen for the air. The k-epsilon model is suitable for fully developed turbulent conditions and the DO model is suitable for radiation. Mesh independence validation is conducted and the mesh is refined for the air outlet, air inlet, and radiant surfaces of the novel terminal to ensure better accuracy. The dimensions of the room, the dimensions of the heating terminal, and the location of the heating terminal are kept consistent with the laboratory room.

**Results**

**Heating performance of the terminal based on the flat heat pipe**

Figure 2 shows the actual heating performance of the terminal based on the flat heat pipe. The overall size of the terminal used in the experiment is 0.98 m in length, 0.86 m in width, and 0.01 m in thickness. The ultra-thin thickness of the flat heat pipe has the potential to reduce space occupancy. Water pipes are arranged behind the flat heat pipe, using silicon grease to attach the flat heat pipe to provide a basic hot source. Considering practical applications, the supply water temperature in the experiment was controlled at 50°C.

![Figure 2: Heating performance of the terminal based on the flat heat pipe.](https://doi.org/10.26868/25222708.2023.1225)

The results show that the calculated heating capacity of the terminal based on the flat heat pipe under experimental conditions is 147.1 W, with a surface heat transfer coefficient of 5.6 W m\(^{-2}\) K\(^{-1}\) and a calculated uncertainty of ±9.2%. The surface temperature of the flat heat pipe is uniform, with a maximum temperature difference of only 1.0°C in the vertical direction, except for the heating source segment, and a maximum temperature difference of only 2.2°C in the horizontal direction. As a phase-change radiant surface, the flat heat pipe can provide better radiant comfort.

The flat heat pipe serves as the radiation surface of the heating terminal. The heat transfer effectiveness of the flat heat pipe transferring heat from the heat source can be evaluated using thermal efficiency. We further calculated the thermal efficiency of the terminal under different supply water temperatures and flow rates, which reflects the ratio of the actual heating capacity to the maximum theoretical heating capacity. The thermal efficiency \(\eta\) is defined as follows:

\[
\eta = \frac{KA(T_s - T_a)}{KA(T_i - T_a)} \quad (3)
\]
Where $K$ is the surface heat transfer coefficient of the flat heat pipe, W m$^{-2}$ K$^{-1}$; $A$ is the surface area of the flat heat pipe, m$^2$; $T_s$ is the average surface temperature of the flat heat pipe, °C; $T_i$ is the supply water temperature, °C; $T_a$ is the ambient temperature, °C.

Based on the calculations, the thermal efficiencies of the flat heat pipe were determined to be 58.9%, 61.7%, 62.1%, 60.3%, and 60.5% at supply water temperatures of 40°C, 45°C, 50°C, 55°C, and 60°C, respectively. The thermal efficiencies of the flat heat pipe were 57.5%, 58.4%, 62.1%, and 64.1% at supply water flow rates of 60 L/h, 80 L/h, 100 L/h, and 120 L/h, respectively. The results indicate that as the supply water flow rate increases, the thermal efficiency increases due to enhanced internal heat transfer caused by the increased flow velocity.

However, an increase in the supply water temperature does not necessarily lead to a simple increase in thermal efficiency. This is because there is an optimal operating point for phase change inside the flat heat pipe, which is closely related to factors such as the working fluid charge and internal vacuum level of the heat pipe. The supply water temperature has an optimal solution. Overall, the overall thermal efficiency of the flat heat pipe for heat extraction ranges from 55% to 65%, mainly due to the presence of certain contact thermal resistance between the heat exchanger and the flat heat pipe. This can be further optimized by using brazing to reduce the contact thermal resistance.

**Problems with using only the flat heat pipe for intermittent heating**

The experimental results show that the heating capacity and surface heat transfer coefficient of the flat heat pipe are relatively small due to the low emissivity of its metal surface, which weakens its radiative heating capability.

Moreover, the current method of combining the flat heat pipe with the water system has the following issues: (1) The heating capacity is insufficient for intermittent heating. (2) The thermal inertia of water is large and the thermal response is slow. (3) The complex structure of the terminal is difficult to adjust by users in the practical applications. Additionally, the current combination of flat heat pipe and water pipe requires a fixed hot source to provide heating capacity, which is difficult to arrange and has high cost.

**Improvement design of the heating terminal based on the flat heat pipe for intermittent heating**

Based on the above-mentioned issues, we further optimized and improved the heating terminal based on the flat heat pipe.

Firstly, the surface of the flat heat pipe was treated by uniformly applying a layer of silicone grease with an emissivity of approximately 0.9 to enhance the radiative heat transfer capability of the terminal. Under the same controlled water supply temperature, increasing the surface emissivity improved the heating capacity of the flat heat pipe to 209.8 W and the surface heat transfer coefficient to 10.9 W m$^{-2}$ K$^{-1}$. It can be seen that increasing the surface emissivity has certain benefits for the terminal based on the flat heat pipe.

More importantly, based on the existing flat heat pipe terminal, we attempted to add fins and fans to the heat exchanger to increase the forced convection heat transfer mode of the flat heat pipe terminal, thereby significantly improving its heating capacity. Figure 3 shows the novel flat heat pipe terminal with optimized forced convection, which combines both convection and radiation heating modes.

![Figure 3: Different heating modes (convection and radiation mode) of the novel terminal based on the flat heat pipe.](https://doi.org/10.26868/25222708.2023.1225)
In addition to the issue of peak heating capacity, there is also a problem with the difficulty of arranging the water system for the flat heat pipe terminal in practical applications. Moreover, the water system itself has a large thermal inertia, which affects the intermittent heating performance of the terminal to some extent. Therefore, a direct-expansion design was carried out for the flat heat pipe terminal, which was combined with an air-source heat pump to replace the original water supply system. The working fluid in the heat exchanger is R134a, and the novel flat heat pipe terminal prototype is shown in Figure 4.

Figure 4: Novel flat heat pipe terminal combined with an air-source heat pump.

The full electrification of buildings is key to achieving low-carbon or even zero-carbon buildings. By utilizing air-source heat pumps, it is possible to further improve the efficiency of "electricity-to-heat" conversion on the basis of electrification. Meanwhile, compared to water as a medium, the compact internal structure of a direct expansion system is more conducive to practical installation, layout, and application adjustment.

Discussion

Indoor temperature distribution under convection mode and radiation mode

We conducted further practical performance testing on the novel flat heat pipe terminal. The experimental results showed that the terminal heating capacity increased fourfold, from 209.8 W to 836.8 W, by adding fins and a cross-flow fan.

Furthermore, we conducted a simulation study of the indoor environment created by the novel flat heat pipe terminal in the test room, and validated the accuracy of the simulation results using measured indoor temperature distribution.

Figure 5 shows the indoor environmental conditions created by the novel flat heat pipe terminal under different modes and airflow velocities (3.5 m/s, 2.5 m/s and 1.5 m/s) obtained through simulation using ANSYS-Fluent. It reveals that the convection mode creates a more uniform temperature distribution. For different wind speeds, the difference is mainly reflected due to the effective distance traveled by air. As the wind speed gradually decreases, the hot air will rise earlier, and resultantly the effective zone up to which the air reach shrinks. However, for the radiation mode, due to thermal buoyancy, even if the air outlet retains a weak air supply (0.5 m/s), there exists a relatively higher vertical temperature gradient in the room. Under this situation, heat is concentrated in the upper left region of the room, which prevents heat flow to the personnel activity area. The convection mode, however, has generally large wind speeds at the height of 0.6 m. A long time heating with convection also brings out discomfort.

Figure 5: Numerical simulation.
Moreover, we compared the simulated results with 20 environmental temperature measurement points obtained from experimental testing in Figure 5. The results showed that the majority of the measuring points were found within 10% error and the maximum error was only 2.0°C (10.8%), which proved the accuracy of simulation results to a satisfactory level.

**Full-process simulation of the radiant-convective terminal from outdoor to terminal to indoor environment**

Through experimental measurement, the surface temperature of the novel terminal, the outlet air temperature of the novel terminal, and the wall temperature are obtained. The indoor environment distribution of the radiation mode and convection mode, therefore, can be simulated through CFD. However, the integration of measured results of the novel terminal into the simulation as input boundary conditions can be a challenge for practical applications. Therefore, we aimed to expand the joint simulation of the radiant-convective terminal from outdoor to terminal to indoor environment, using only the outdoor temperature, building envelope parameters, and the expected heating capacity of the terminal (i.e., the room's heating load) as input.

Based on the simulation of indoor environment distribution through ANSYS-Fluent, we further supplemented the RC model for building envelope and Flomaster for heat pump to achieve a more extensive application of the "outdoor-terminal-indoor environment" joint simulation process, as shown in Figure 6.

**Figure 6: "outdoor-terminal-indoor environment" joint simulation process for the radiant-convective heating terminal.**

The RC model assumes a convective heat transfer coefficient $h_0$ of the wall to calculate the heat transfer resistance on the inner surface of the wall. By inputting different terminal heating capacities $Q$, the corresponding interior surface temperature $T_{ei}$ and indoor environmental temperature $T_{ia}$ can be calculated, as shown in Figure 7. Equations (3), (4), and (5) demonstrate the specific calculation process of the RC model. Based on this, the novel terminal was simulated using Flomaster, and the indoor environmental temperature $T_{ia}$ obtained from the RC model was used as the inlet temperature of the terminal. By inputting different outdoor temperatures $T_{ao}$ and terminal air supply rates $m$, the outlet air temperature and surface temperature of the terminal can be simulated.

**Figure 7: RC model.**

Equations (3), (4) and (5):

$$C_a \frac{dT_{ei}}{dt} = Q + \frac{T_{ei} - T_{ai}}{R_{ei}},$$  

$$C_a \frac{dT_{ei}}{dt} = \frac{T_{ei} - T_{ai}}{R_{ei} + R_{e-o}} + \frac{T_{ei} - T_{ao}}{R_{e-o}} + \frac{T_{ei} - T_{ao}}{R_{e-o} + R_{e-i}};$$  

$$C_a \frac{dT_{ei}}{dt} = \frac{T_{ei} - T_{ai}}{R_{ei} + R_{e-o}} + \frac{T_{ei} - T_{ao}}{R_{e-o}}.$$  

Based on this, by using the interior surface temperature of the building envelope, the outlet air temperature, and the surface temperature of the novel terminal as boundary conditions, the indoor environmental distribution can be obtained through CFD simulation. Importantly, the obtained indoor environmental distribution results are compared and verified against the initial assumed values. The simulated inlet air temperature $T_{it}$ of the terminal is judged to determine if it matches the initially assumed input indoor environmental temperature $T_{ai}$. Likewise, the simulated convective heat transfer coefficient $h_1$ is compared against the initially assumed input $h_0$. If the simulated error is within a certain range (such as ±10%), the simulation results are considered correct. If the simulated error exceeds the range, the obtained new results will replace the original assumed values, and the entire simulation process will be repeated. This iterative cycle continues until the error is acceptable, and accurate simulation results are obtained.
to the relatively larger vertical temperature difference. Generally, a new round of simulation is required to obtain more accurate results. This also demonstrates the applicability of the newly proposed joint simulation process for intermittent heating with radiant-convective terminals, which can more accurately obtain the indoor environmental distribution under different convection modes and radiation modes during intermittent heating.

Table 1: Simulation results of inlet air temperature in convection mode.

<table>
<thead>
<tr>
<th>Input (indoor environmental temperature $T_{ai}$) $^\circ$C</th>
<th>Simulated results (inlet air temperature $T_{it}$) $^\circ$C</th>
<th>Absolute error $^\circ$C</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.7</td>
<td>13.7</td>
<td>-1.0</td>
</tr>
<tr>
<td>18.8</td>
<td>19.0</td>
<td>0.2</td>
</tr>
<tr>
<td>20.6</td>
<td>21.2</td>
<td>0.6</td>
</tr>
<tr>
<td>21.9</td>
<td>22.6</td>
<td>0.7</td>
</tr>
<tr>
<td>22.8</td>
<td>23.4</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Table 2: Simulation results of inlet air temperature in radiation mode (first time).

<table>
<thead>
<tr>
<th>Input (indoor environmental temperature $T_{ai}$) $^\circ$C</th>
<th>Simulated results (inlet air temperature $T_{it}$) $^\circ$C</th>
<th>Absolute error $^\circ$C</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.7</td>
<td>9.8</td>
<td>-4.9</td>
</tr>
<tr>
<td>18.8</td>
<td>14.2</td>
<td>-4.6</td>
</tr>
<tr>
<td>20.6</td>
<td>16.4</td>
<td>-4.2</td>
</tr>
<tr>
<td>21.9</td>
<td>17.8</td>
<td>-4.1</td>
</tr>
<tr>
<td>22.8</td>
<td>18.8</td>
<td>-4.0</td>
</tr>
</tbody>
</table>

Table 3: Simulation results of inlet air temperature in radiation mode (second time).

<table>
<thead>
<tr>
<th>Input (simulated results in the first time) $^\circ$C</th>
<th>Simulated results (inlet air temperature $T_{it}$) $^\circ$C</th>
<th>Absolute error $^\circ$C</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.8</td>
<td>10.2</td>
<td>0.4</td>
</tr>
<tr>
<td>14.2</td>
<td>14.5</td>
<td>0.3</td>
</tr>
<tr>
<td>16.4</td>
<td>16.7</td>
<td>0.3</td>
</tr>
<tr>
<td>17.8</td>
<td>18.1</td>
<td>0.3</td>
</tr>
<tr>
<td>18.8</td>
<td>19.0</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Combined operation strategy of convection mode and radiation mode under intermittent heating

Based on the above research results, it can be seen that the downward airflow convection mode has strong heating capacity and the heat is concentrated in the lower area of the room where people are active, but long-term use may cause draught and discomfort. The radiation mode has weaker heating capacity but the flat heat pipe surface temperature is uniform, providing better thermal comfort. This inspires us to develop a heating regulation strategy that combines the advantages of convection and radiation modes for intermittent heating demands. The experiment of first convection mode and then radiation mode was conducted, as shown in Figure 8.

As shown in Figure 8, the convection mode is used in the start-up period to quickly heat the room. When the room temperature meets the user's needs, the fan is turned off and the radiation mode is switched to provide better comfort. The experimental results show that the room temperature rose to $18^\circ$C within 40 minutes, which can preliminarily achieve a rapid thermal response process to meet the requirements of intermittent heating. Intermittent heating based on occupants' behavior for energy savings becomes achievable from a temporal dimension. After that, the novel terminal is kept in radiation mode for the following 1 hour and 20 minutes, and the room temperature is stable at about $20^\circ$C. A higher radiation ratio is provided to enhance the thermal comfort of the terminal, which preliminarily achieves a balance between energy saving and thermal comfort under the intermittent heating mode.

From an energy-saving perspective, the novel terminal provides a suitable method for intermittent heating, which achieves energy savings from a temporal perspective compared to continuous heating. In addition, the terminal was combined with an air-source heat pump, and experimental results showed that the coefficient of performance (COP) for the convection mode and radiation mode were 2.6 and 2.5, respectively. Furthermore, during the start-up period, the convection mode with downward airflow can efficiently supply heat to the lower part of the room, achieving efficient heat supply in the indoor space.

From a thermal comfort perspective, the novel terminal provides an adjustable radiation mode, which reduces the feeling of prolonged blowing compared to the convection mode. On this basis, we calculated the thermal comfort index for the convection mode and radiation mode, and the results showed that the radiation mode had a higher PMV (+0.1) compared to the convection mode (-0.5). This indicates that during the stable heating period, switching to the radiation mode can better enhance the thermal comfort of intermittent heating.
Conclusion
This paper presents an experimental and simulation study on a novel flat heat pipe terminal under intermittent heating demand. The main conclusions are as follows:
(1) The flat heat pipe terminal has a small thermal inertia and a uniform surface temperature, with a maximum surface temperature difference of only 2.2°C. It can provide good thermal comfort in radiation mode.
(2) The heating capacity of the flat heat pipe in radiation mode alone is low, but increasing the surface emissivity and using forced convection mode can significantly improve the heating capacity of the flat heat pipe, better meeting the requirements of intermittent heating.
(3) A heating regulation strategy combining convection and radiation modes is proposed for intermittent heating demand. Under the convection mode, the room temperature can rise to 18°C within 40 minutes, achieving a preliminary and rapid thermal response process. Then, the radiation mode can be switched on to reduce the draught and enhance the thermal comfort.
The results show that the novel flat heat pipe terminal can meet the heating demand under intermittent heating mode and provide fast, energy-saving, and comfortable heating. The proposed novel terminal and the first convection and then radiation heating regulation strategy can provide technical support and reference for research on intermittent heating.

Acknowledgement
The study was supported by the National Natural Science Foundation of China (Grant No. 52130803) and the National Science Fund for Distinguished Young Scholars (Grant No. 51825802).

References