Numerical investigation of the heat transfer in realistic rooms with a two-panel radiator

Zhenming Peng¹, Svenja Carrigan¹, Oliver Kornadt¹
¹University of Kaiserslautern-Landau, Kaiserslautern, Germany

Abstract
The heat transfer in room with heating provided by a radiator can be strongly affected by the positioning and the operating temperature of the radiator as well as the thermal transmittance (U-value) of the room envelope. The effect of the U-value on the heat transfer has not been researched in detail. Therefore, this study presents the results of three-dimensional computational fluid dynamics (CFD) simulations of the conjugate heat transfer in rooms with realistic constructions. For each simulation, a different U-value of the envelope or a different operating temperature of the radiator is chosen. The heat transfer coefficients for the exterior walls differ up to 3.8 times from the existing values from the literature and European/German standards. New revised values are proposed. The results can be used in building simulations for rooms with radiators for a more accurate estimation of the energy demand or the critical surface temperature for protection against moisture.

Highlights
- CFD simulations of conjugate heat transfer in a room with Reynolds-Averaged Navier-Stokes approach and Surface-to-Surface radiation model.
- Consideration of realistic wall constructions with various U-values and thermal bridge effects.

Introduction
In 2021, up to 50 % of the energy demand for buildings worldwide was devoted to heating (Goodson et al., 2022), of which around 70 % is used for space heating. The energy demand needed for space heating primarily depends on the local climate and the envelope quality of the buildings as well as the energy performance of the heating systems. The heat transfer through the building envelope varies significantly between different heating systems (Peng, Carrigan, & Kornadt, 2022b). One of the most widely used space heating systems is the panel radiator. The numerical and experimental research on the panel radiator itself is not novel. There are already numerous investigations on enhancing the energy performance of the panel radiator from the design perspective to allow a lower inlet water temperature while achieving the same heating load in the room. Several studies have proven that by adding optimised convection fins, the thermal efficiency of the radiator can be increased (Beck et al., 2004; Myhren & Holmberg, 2011). With the combination of the panel radiator and an integrated ventilation unit, the energy efficiency of the radiator and the thermal comfort in the room can also be improved (Myhren & Holmberg, 2009).

The energy demand of a heated room depends not only on the efficiency and type of the heating systems but also on the structural properties of the room, such as the thermal transmittance (U-value), the thermal mass of the building elements and the geometry of the envelope. All these parameters have significant impact on the heat transfer mechanisms. With different structural properties, the heating demand can change significantly to achieve the same room temperature (Peng, Carrigan, & Kornadt, 2022a). For a room with a radiator, the mechanisms include: (1) the convective heat transfer through the airflow driven by buoyancy, (2) the radiative heat transfer between the indoor wall surfaces, (3) the radiative heat transfer between the radiator and indoor wall surfaces, and (4) the conductive heat loss through the envelope.

One of the earlier studies on heat transfer in a room equipped with a radiator was performed by Khalifa & Marshall, 1990, in a 2-zone climate chamber. Several heat transfer correlations for the convective heat transfer coefficient (CHTC) on the surfaces of building elements were developed with different radiator positions in the chamber. However, the investigated walls only consisted of a single isolation layer, and one-dimensional conduction was assumed. Furthermore, the radiation effect was minimised through aluminium foil covering both sides of the building elements. These assumptions can lead to an unrealistic estimation of the CHTC when applying the correlations to real-life buildings. Followed by the investigation of Wallentén, 2001, the local CHTC was investigated on a wood-framed stud wall with an U-value of 0.27 W/m²K in a climate chamber with a radiator. The author concluded the significant effect of the positioning of the radiator on the heat transfer in the room. The local CHTCs were found to be more than ten times the value from the model of CHTC for uniformly heated plates by Churchill & Chu, 1975. Although the study considered the radiative heat transfer in the calculation, the thermal conditions of realistic constructions, such as conventional building materials with a much higher heat storage capacity, different U-values of the envelope and exhibiting thermal bridges, were not considered.

Table 1 summarises the CHTC correlations from literature. Additionally, it presents the European standard CHTC value, along with the general combined values.
Table 1: Correlations for the convective (h_{conv}, CHTC) or total (h_t) heat transfer coefficient for vertical walls in a room with a radiator from two different publications as well as the general values for vertical walls according to DIN EN ISO 6946 and DIN 4108-2.

<table>
<thead>
<tr>
<th>Heat transfer coefficient [W/m²K]</th>
<th>Remarks</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h_{conv} = 2.20 \cdot \Delta T^{0.21}$</td>
<td>Adjacent to radiator with unknown distance</td>
<td>Khalifa &amp; Marshall, 1990</td>
</tr>
<tr>
<td>$h_{conv} = 1.85 \cdot \Delta T^{0.27}$</td>
<td>Other wall positions</td>
<td></td>
</tr>
<tr>
<td>$h_{conv} = 0.93 \cdot (\Delta T/H)^{0.25}, \Delta T \cdot H^3 &lt; 9.5$</td>
<td>Adjacent to radiator with 0.12 m</td>
<td>Wallentén, 2001</td>
</tr>
<tr>
<td>$h_{conv} = 0.7 \cdot (1.33 \cdot \Delta T^{1/3} - 0.474/H), \Delta T \cdot H^3 &gt; 9.5$</td>
<td>General value</td>
<td>DIN EN ISO 6946</td>
</tr>
<tr>
<td>$h_t = 7.69$</td>
<td>General combined value for energy calculation</td>
<td></td>
</tr>
<tr>
<td>$h_t = 4$</td>
<td>General combined value for preventing mould growth and condensation</td>
<td>DIN 4108-2</td>
</tr>
</tbody>
</table>

(total heat transfer coefficient) that include both CHTC and RHTC (radiative heat transfer coefficient) from DIN EN ISO 6946 (DIN, 2018), and the German standard values from DIN 4108-2 (DIN, 2013) for vertical walls. $\Delta T$ is the temperature difference between the surface temperature and the average air temperature in the room. $H$ is the wall height in meters.

As can be seen, CHTC correlations from Khalifa were developed for different wall positions as a function of the temperature difference. The CHTC correlation from Wallentén considers the room height as an extra parameter, however, the author suggested that it can be only applied in a room with room height of approximately 2.5 m. The standardised values for the heat transfer coefficient are used for two different purposes. For the energy demand calculation, values of 2.5 and 7.69 W/m²K are applied for CHTC and total heat transfer coefficient, respectively. For protection against moisture to prevent mould growth and condensation, a reduced total heat transfer coefficient of 4 W/m²K is used to determine the critical surface temperature.

The present study aims to numerically investigate the room-side heat transfer in realistic rooms of concrete structure equipped with a two-panel radiator. For each simulation, the U-value of the building envelope or the operating temperature of the radiator is varied. Furthermore, the thermal effects of the geometric thermal bridge between two exterior walls are considered. Three-dimensional computational fluid dynamics (CFD) simulations are conducted to examine the heat transfer characteristics of the rooms. The heat transfer coefficients obtained from the different room conditions are analysed and compared with those in the literature and standards.

Methods

Figures 1 and 2 show the detailed dimensions of the room and the panel radiator, respectively. The investigations focused on a room with internal measurements of 4.2 meters in length, 3 meters in width and 3 meters in height. This room features four walls, two of which are adjoining exterior walls. The exterior wall build-up is (from inside to outside): 15 mm plaster, 300 mm concrete, an insulation layer with variable thickness and 15 mm plaster. The thickness of the insulation layer with a thermal conductivity of 0.035 W/mK is chosen as 29 mm, 85 mm, 115 mm and 160 mm to achieve a total U-value of 0.84 W/m²K, 0.35 W/m²K, 0.28 W/m²K and 0.20 W/m²K, respectively. These U-values represent U-values commonly found in German buildings over the last 50 years. The internal wall build-up is 15 mm plaster, 175 mm concrete and 15 mm plaster. The internal floor and ceiling consist of 45 mm screed, 40 mm insulation layer and 150 mm concrete.

A two-panel radiator with dimensions 1.25 m × 0.85 m × 0.102 m is placed 50 mm to the exterior wall 1 and 110 mm to the floor. The panels are simplified with a homogeneous thickness of 15 mm. The length and the shape of the convection fins are selected according to DIN EN 442-2 (DIN, 2015). The thickness of the fin is assumed to be 1 mm.
Physical model

In order to obtain the temperature, pressure and velocity field of the domains, the continuity, momentum, and energy conservation equations need to be solved. Only large-scale turbulence was solved to achieve a reasonable simulation time. Therefore, the Reynolds-averaged Navier-Stokes approach was employed. This approach will statistically average the conservation equations, resulting in an additional term, the “Reynolds stress” and needs to be solved by additional transport equations.

Different turbulence models have been developed to solve the Reynolds stress for different flow fields. The k-ε (kinetic-energy – energy-dissipation rate) model is the more popular of the two-equation turbulence models. This model gives sufficient accuracy of the flow field far away from the fluid-solid interface with the application of wall functions. However, this results in a coarse mesh of the first layer to the interface, leading to a low resolution of the near-wall solutions. The k-ω (kinetic energy – specific-energy-dissipation rate) based shear-stress transport (SST) provides the possibility to achieve higher accuracy and resolution of the near-wall solution, more specifically the solution of the viscous sublayer (Bardina, Huang, & Coakley, 1997), which is needed for the heat transfer analysis. Furthermore, compared to the other two-equation turbulence models, the SST model also improves the prediction of separated flows which are typical for buoyancy-driven air flows in closed spaces (Bardina et al., 1997). Moreover, several studies on natural convection in an enclosure have successfully validated the solution with experimental data (Rundle et al., 2011; Zhang et al., 2007). Hence, the k-ω-SST model was used.

The radiation heat transfer in the room was accounted for by the Surface-to-Surface radiation model. It considers the radiation exchange between the surfaces in the room and assumes them to be grey with constant emissivity under 0.3, the modified “ideal gas law” is used to calculate the density, which is a function of temperature. Other air properties, such as viscosity and thermal conductivity, were modelled as piecewise linear functions of the temperature.

To analyse the heat transfer, the CHTC ($h_{\text{conv}}$) and RHTC ($h_{\text{rad}}$) as well as the total heat transfer coefficient ($h_t$) were calculated based on the following equations:

\[ q_{\text{conv}} = J_i = (E_i - \varepsilon_i G_i) \]  
\[ G_i = \sum_{j=1}^{N} F_{ij} J_j \]  
\[ J_j = E_j + (1 - \varepsilon_j) G_j \]  
\[ h_{\text{conv}} = \frac{q_{\text{conv}}}{T_{\text{ref}} - T_{\text{ref}}} \]  
\[ h_{\text{rad}} = \frac{q_{\text{rad}}}{T_{\text{ref}} - T_{\text{ref}}} \]  
\[ h_t = h_{\text{conv}} + h_{\text{rad}} \]

where \( q_{\text{conv}} \) and \( q_{\text{rad}} \) are the convective and radiative heat flux, respectively. \( q_{\text{conv}} \) is solved by equation 1, which applies Fourier’s law of conduction, as heat is transferred through conduction at the surface in the conductive sublayer. \( q_{\text{rad},ij} \) is the net radiative heat flux of cell face i. \( \lambda \) is the thermal conductivity and \( \varepsilon \) denotes the emissivity, which is equal to the absorptivity. Equation 2 states that \( q_{\text{rad},i} \) is the difference between its emitted heat flux \( E_i \) and the absorbed heat flux from incoming radiation \( G_i \). To solve \( G_i \), equation 3 is needed, which indicates that \( G_i \) is equal to the sum of the radiation reaching face i from all surfaces. \( F_{ij} \) accounts for the proportion of radiation energy transfers from j to i. \( J_j \) is the difference between emitted and absorbed radiation solved by Equation 4. \( T_{\text{sl}} \) and \( T_{\text{ref}} \) are the surface temperature and the reference temperature, respectively. The average room air temperature was used as the reference temperature in this study to maintain the usability of the derived heat transfer coefficient in most simulation programmes and their comparability to those existing in the literature and standards. The surface-averaged heat transfer coefficients were determined using integrated average temperatures and heat fluxes.

Numerical procedure

Ansys Fluent was used to solve the above-mentioned physical models (ANSYS Inc). The finite volume approach is employed, wherein the equations are integrated over control volumes of defined mesh elements. All practical numerical schemes employed for fluid flow based on the discretised form of the equations produce an amount of numerical diffusion. This results, e.g., from truncation of the Taylor series during the interpolation process to obtain the solution on the cell surfaces. In order to minimise the truncation error, second-order discretisation schemes are used. It’s notable that the pressure is discretized with the ‘body force weighted’ scheme due to buoyancy force dominance, and updated simultaneously with velocity in each iteration using the ‘Coupled Algorithm’.

The steady-state simulation was performed. As the studied cases exhibit natural convection in a closed space, convergence is difficult to achieve with steady-state simulations. The pseudo-transient under-relaxation method was used to enhance the diagonal dominance of the coefficient matrix of the equations by introducing an artificial time derivative to each cell, leading to an improvement in the numerical stability (Ferziger & Perić, 2002). A convergence criterion of 10^{-3} has been set for the normalised residues of the energy equation, while the continuity, momentum, and turbulence equations have a convergence criterion of 10^{-5}. Besides the convergence criteria, several properties in the flow field, such as...
average room temperature and surface-averaged heat flux of the interior surfaces of the exterior walls, were monitored to ensure that the convergence of the solution was reached.

**Boundary conditions**

The outer surfaces of the two exterior walls were assigned an external heat transfer coefficient of 25 W/m²K under a constant outdoor temperature of -5 °C. The internal building components were assumed to be adiabatic. However, their thickness was still modelled to account for the effect of thermal mass. For the surfaces of building elements and the white radiator paint an emissivity of 0.9 and 0.79 was assumed, respectively.

There are several ways to define the boundary condition of the radiator as a heat source, e.g., replacing the radiator with a convective heat source and defining a uniform heat flux on the radiator surfaces. In this study, the front and back panels were maintained at set temperatures of 30 °C / 35 °C / 40 °C / 45 °C. The U-values were varied as described above. The investigated cases are shown in Table 2.

**Grid generation and independency**

In addition to using the higher-order discretisation schemes, truncation errors can be reduced when the airflow is aligned with the mesh. Therefore, a structured conformal hexahedral grid was created. Although it will not be guaranteed that all the hexahedral cells will be aligned with the flow because of the complex flow in the enclosure, it is in any case advantageous as it can reduce the total number of cells compared to tetrahedral mesh. To fully resolve the boundary layers near the wall, the height of the first layer of cells vertical to the fluid-solid interface was refined to achieve y+ (dimensionless distance from the wall) values of around 1, indicating that the cell lies in the viscous sublayer. The thickness of further layers vertical to the interface is based on the thickness of the previous layer with an increment factor of 1.2 until it reaches the set maximum element size of 0.06 m. Figure 3 shows the final generated mesh of the central x-y-plane from the air domain vertically sliced from the centre of exterior wall 1. The panel radiator is depicted as a white rectangle. Figure 4 shows the mesh on the vertical and horizontal central plane of the panel radiator.

To reduce truncation errors and discretisation errors further the mesh elements can be refined. To accomplish this, a grid independence study was conducted. The grid convergence index (GCI) calculation method described by Celik et al., 2008, was employed to evaluate the grid independency. The GCI is a measure of the grid convergence error in percentage resulting from the refinement of the mesh size, indicating the deviation of the computed solution from an asymptotic numerical solution. To determine the optimal mesh size, several simulations were conducted using different maximum global mesh sizes. The global mesh size was adjusted to create three series of grids with mesh sizes of 0.08 m, 0.06 m, and 0.04 m, respectively. The GCI values of the surface temperature T_st and the convective and radiative heat flux q_con and q_rad were calculated for the position at the edge between the two exterior walls. In addition to the GCI of the local solutions, the GCI values for the surface-averaged solutions on the exterior walls 1 and 2 and the average room air temperature T_{ar} were also calculated. The results show that the GCI values of the chosen data points were all under 1 % except with a mesh size of 0.08 m, where the maximum GCI of q_con and q_rad for the solution at the edge was 4.09 % and 2.55 %, respectively. With the refinement of the mesh size to 0.06 m, the values were reduced to 1.12 % and 1.64 %, respectively. Therefore, a global mesh size of 0.06 m was used to obtain better local solutions at the edge with acceptable computational cost.

**Results and Discussion**

**Flow characteristics**

To better understand the flow field in the studied cases and its effect on the CHTC, it is helpful to analyse the flow pattern. Figure 5 shows the selected planes for the illustration. Plane-1 and plane-2 are the central x-y-plane and z-y-plane of the air domain, respectively. Additionally, plane-3 is a z-y-plane that passes through the centre of the radiator. As an example, streamlines tangential to the velocity vectors on these planes are shown for C8. The boundary on the left side represents the inner surface of the exterior wall for each plane.

It was found that the flow patterns on plane-1 and plane-3 were qualitatively remarkably similar across all studied cases. On plane-1, the air close to the floor was initially...
heated up by the undersides of the radiator surfaces, which then rose due to the density change. At the same time, the inner surface of exterior wall 1 cooled the airflow above the radiator, causing it to split into two recirculation regions near the ceiling. One small region was located in the upper left corner, while another stretched towards the opposite interior wall. The airflow near the interior wall recirculated back towards the exterior wall and the radiator, while near the floor, it detached and formed small recirculation regions. On plane-3, the rising airflow also was separated into two main regions. The first region was observed, where a counterclockwise recirculation moved towards exterior wall 2, while for the second region, a clockwise recirculation region moved towards the opposite interior wall occurred. Although these two regions exhibited a similar recirculation pattern, the airflow velocity in the first region was higher due to the stronger downward airflow from the extra cooling of exterior wall 2.

The flow patterns on plane-2 appeared less stable and had approximately one order of magnitude lower average air velocity than in plane-3. The airflow from the centre of the ceiling tended to move towards exterior wall 2 and the opposite interior wall. Across the cases with a different panel temperature or U-value, a stable pattern could only be observed close to the exterior wall 2, where the air was cooled and bent downwards towards the floor before recirculating back to the upper region at the height of around 2.1 m. An unstable pattern containing numerous recirculation regions was found in the other regions away...
from the exterior wall 2. These recirculation regions had no fixed locations across all the cases, which might be attributed to the natural instabilities of the buoyancy flow.

Above the radiator, where the air was rising, the maximum velocity occurred between heights of 2.0 m and 2.3 m for all cases with the maximum value observed in C10 at 0.65 m/s. Furthermore, the maximum velocity increased with higher panel temperatures or U-values. This is due to the buoyancy force increasing by the warmer airflow, which itself originates from a higher panel temperature or from a lower surrounding temperature around the rising airflow due to the stronger cooling effect of the exterior walls with a higher U-value.

Table 3 shows the average room air velocity and temperature for the studied cases. Increasing the panel temperature from 35 °C (C6) to 45 °C (C8) with the same U-value (0.35 W/m²K) caused a small increase of around 0.003 m/s in average velocity. Increasing the U-value from 0.35 W/m²K (C8) to 0.84 W/m²K (C10) with the same panel temperature (45°C) caused an increase of 0.01 m/s in average air velocity. In general, changing the U-value had a significant impact on the overall air flow velocity, while the change in the panel temperature primarily affected the air velocity locally around the thermal plume and recirculation regions near the panel radiator.

<table>
<thead>
<tr>
<th>CHTC</th>
<th>C1</th>
<th>C2</th>
<th>C3</th>
<th>C4</th>
<th>C5</th>
<th>C6</th>
<th>C7</th>
<th>C8</th>
<th>C9</th>
<th>C10</th>
</tr>
</thead>
<tbody>
<tr>
<td>τ ((\text{m/s}))</td>
<td>0.028</td>
<td>0.030</td>
<td>0.031</td>
<td>0.032</td>
<td>0.033</td>
<td>0.035</td>
<td>0.036</td>
<td>0.038</td>
<td>0.046</td>
<td>0.048</td>
</tr>
<tr>
<td>T ((\text{°C}))</td>
<td>23.9</td>
<td>28.2</td>
<td>22.5</td>
<td>26.6</td>
<td>30.8</td>
<td>25.0</td>
<td>29.0</td>
<td>33.0</td>
<td>21.9</td>
<td>25.1</td>
</tr>
</tbody>
</table>

Heat transfer characteristics

Figure 6 presents the derived local total, convective and radiative heat transfer coefficients \((h, h_{\text{con}}, h_{\text{rad}})\) as solid, dotted, and dashed lines, respectively, depending on the horizontal distance (L) from the edge of exterior walls 1 and 2 for C6, C8, and C10. The orange and green lines represent values at heights of 0.535 m (radiator centre) and 1.98 m (1 m above the radiator) for each of the exterior walls 1 and 2. Positive values indicate the same sign of the heat flux and the difference between the local surface and the average room temperature, while negative values indicate the opposite. The results showed that the local heat transfer coefficients had a consistent pattern at the same height across all cases. However, there were noticeable differences between the two exterior walls.

On the exterior wall 1 where the radiator was located, directly above the radiator at a height of 1.98 m, the CHTC was higher than in other regions due to the thermal plume, except where reverse flow occurred. This caused a slight drop in CHTC, especially in the middle upper part of the wall. The RHTC directly above the radiator was negative in regions where warm airflow heated the wall surfaces to a temperature higher than the surrounding room surfaces, causing heat to be transferred to the room by radiation. For a different reason, the RHTC at a height of 0.535 m behind the radiator showed strongly negative values, this is because the surfaces behind the radiator were strongly heated up by the back panel by radiation causing a much higher surface temperature than the room average temperature.

In regions not directly above the radiator at the height of 1.98 m, the CHTC remained nearly constant up to around 0.2 m from the edges. Convective heat transfer was suppressed due to conduction dominating in regions where laminar viscous sublayers of the airflow at both walls overlapped. The transition from a laminar viscous to turbulent region affected the convective heat transfer positively, resulting in a peak CHTC around 0.03 m from the edge between the two exterior walls and 0.07 m from edge between exterior wall 1 and the adjacent interior wall. As the boundary layer of the downward airflow developed along the edges, its thickness increased. The peak values were reduced, as can be seen at the height of 0.535 m. Notably, extreme values of CHTC and RHTC occurred on the wall regions close to the side panels of the radiator. These values resulted from the small difference between the local surface and average room temperature. Heat transfer processes were less complicated for exterior wall 2, with radiation heat transfer dominating. The CHTC was generally lower than on the exterior wall 1. The suppression of the convective heat transfer from the edge was also observed. However, peak values of CHTC at the edge between the two exterior walls were much higher than those at the edge adjacent to the interior wall. This can be explained by the strong counterclockwise recirculation moving towards exterior wall 2, which weakened as the distance from the radiator increased. Furthermore, the local temperature difference between the wall surface and the airflow was much higher at the edge between the exterior walls.

The similar distribution of heat transfer coefficients on the exterior walls suggested comparable flow characteristics near the exterior walls across all the cases, likely due to stable airflow patterns. However, the overall heat flux magnitudes differed slightly between the exterior walls in the same case and across the cases. When comparing C6 and C8, a stronger impact on the convective heat transfer at exterior wall 1 was found for C8. This is because changing panel temperature strongly affected the air velocity close to the panel radiator. However, a larger effect on the convective heat transfer at both exterior walls was discovered by changing the U-value of the envelope in C10 to reach a similar average room temperature as C6 with the same panel temperature as C8. This was due to the global impact on the air velocity. A strong impact on both walls was also observed for the radiative heat transfer, as the overall temperature differences between the wall surfaces and between the radiator and wall surfaces were impacted globally by changing the U-value.

Convective heat transfer contributed between 35 % (C2) and 40 % (C10) to the total heat transfer on exterior wall 1, and between 35 % (C2) and 44 % (C10) on exterior wall 2. The strong radiative heat transfer between the back...
Table 4: 1% percentile of the local total heat transfer coefficient ($h_{L,1%}$) in the thermal bridge region at the edge between the exterior walls 1 (W1) and 2 (W2), along with the surface-averaged total heat transfer ($h_t$) for the whole surface of W1 and W2.

<table>
<thead>
<tr>
<th>[W/m²K]</th>
<th>C1</th>
<th>C2</th>
<th>C3</th>
<th>C4</th>
<th>C5</th>
<th>C6</th>
<th>C7</th>
<th>C8</th>
<th>C9</th>
<th>C10</th>
</tr>
</thead>
<tbody>
<tr>
<td>K</td>
<td>W1</td>
<td>W2</td>
<td>W1</td>
<td>W2</td>
<td>W1</td>
<td>W2</td>
<td>W1</td>
<td>W2</td>
<td>W1</td>
<td>W2</td>
</tr>
<tr>
<td>$h_{L,1%}$</td>
<td>1.65</td>
<td>1.80</td>
<td>1.75</td>
<td>1.89</td>
<td>1.67</td>
<td>1.83</td>
<td>1.74</td>
<td>1.87</td>
<td>1.83</td>
<td>1.99</td>
</tr>
<tr>
<td>$h_t$</td>
<td>9.23</td>
<td>4.90</td>
<td>10.35</td>
<td>5.09</td>
<td>9.86</td>
<td>4.99</td>
<td>10.26</td>
<td>5.18</td>
<td>10.64</td>
<td>5.34</td>
</tr>
</tbody>
</table>

To apply the results in practical situations, different scenarios must be considered, e.g., protection against moisture and investigation of energy demand. For preventing mould growth and condensation on room surfaces, the critical surface temperature is of interest, which requires simulating the thermal bridge to obtain precise results. The simulation typically includes a region with a length of 1 m to the central position of a thermal bridge (DIN EN ISO 10211) and assumes a room-side surface temperature lower than the average room temperature through the change only from 1-10, respectively. By comparing the values points from low to high ΔT correspond to the case number from 1-10, respectively. By comparing the values between C6, C8 and C10, it was found that to reach the same average room temperature, through the change only in the U-value, $h_{conv}$ of exterior wall 2 was impacted more compared to changing the panel temperature. As for exterior wall 1, the change in $h_{conv}$ was almost the same for changing the U-value or panel temperature.

Figure 7 presents the surface-averaged CHTC ($h_{conv}$) for exterior walls 1 (left) and 2 (right) over the absolute value of the difference between the surface temperature and the average room temperature (ΔT). The values obtained from the present study, Khalifa, Wallén, and DIN EN ISO 6946 are shown in blue, orange, green and black, respectively. Notably, ΔT increased with increasing U-value or panel temperature. Thus, the data points from low to high ΔT correspond to the case number from 1-10, respectively. By comparing the values between C6, C8 and C10, it was found that to reach the same average room temperature, through the change only in the U-value, $h_{conv}$ of exterior wall 2 was impacted more compared to changing the panel temperature. As for exterior wall 1, the change in $h_{conv}$ was almost the same for changing the U-value or panel temperature.

The findings also indicated that the $h_{conv}$ in studied cases had with up to 1.7, 1.5 and 3.8 times higher values than the standardised value and the values predicted by the correlations from Khalifa and Wallén, respectively. For exterior wall 2, the results were closer to the one from Khalifa than to the standardised value, with mean average percentage errors of 12%, 23%, respectively. The derived $h_{conv}$ from the studied cases were fitted with the curves in light blue. The correlations between $h_{conv}$ and |ΔT| are provided below:
For the exterior wall adjacent to the radiator:
\[ h_{\text{conv}} = 3.73|\Delta T|^{0.12} \]
For the exterior side wall vertical to the radiator:
\[ h_{\text{conv}} = 1.60|\Delta T|^{0.31} \]
The mean average percentage error and the root mean square deviation were 1.1 % and 0.05 for exterior wall 1, and 0.5 % and 0.01 for exterior wall 2, respectively. The correlation can be applied in a room with a similar range of the average room temperature between 22 °C to 33 °C and similar room geometry.

**Conclusion**

This study numerically investigated the heat transfer in a room equipped with a two-panel radiator with realistic wall construction and accounting for the thermal bridge effect. In each simulation, either the U-value of the building envelope or the panel temperature of the radiator was varied. The results showed that achieving the same average room temperature required a larger adjustment of only the U-value compared to adjusting only the panel temperature. Within this, adjusting only the U-value had a greater impact on both the convective and radiative heat transfer than changing the panel temperature alone.

The heat transfer coefficients on the exterior walls obtained from the study were analysed and compared with those from the literature and European/German standards. It was found that they varied up to 3.8 times from the existing values. Therefore, revised values were suggested for different application scenarios in rooms with similar room conditions. To identify the critical surface temperature of a thermal bridge to prevent mould growth and condensation, a total heat transfer coefficient of 1.5 W/m²K lower than the standardised value is recommended. For investigation of the energy demand, separate convective heat transfer coefficient correlations with the temperature difference between the surface and the room were developed for the exterior walls depending on their position (adjacent or vertical) to the radiator.

**Acknowledgement**

This research is funded by the German Research Foundation (DFG). Its support is gratefully acknowledged.

**References**


Peng, Z., Carrigan, S., & Kornadt, O. (2022a). Investigation of the influence of room-side convective heat transfer on the heating energy demand of a room with different insulation levels and wall constructions (in German). In: Proceedings Bauphysiktage, 131–133.


