Analysis of the impact of convector heaters operating at low supply temperature on the possibility of deterioration of thermal conditions

Published Version:
Zukowski, M., Semprini, G. (2024). Analysis of the impact of convector heaters operating at low supply temperature on the possibility of deterioration of thermal conditions. BUILDING AND ENVIRONMENT, 252, 1-12 [10.1016/j.buildenv.2024.111286].

Availability:
This version is available at: https://hdl.handle.net/11585/973026 since: 2024-09-02

Published:
DOI: http://doi.org/10.1016/j.buildenv.2024.111286

Analysis of the impact of convector heaters operating at low supply temperature on the possibility of deterioration of thermal conditions

Mirosław Zukowski

Department of HVAC Engineering, Faculty of Civil Engineering and Environmental Sciences, Bialystok University of Technology, Wiejska 45E Street, 15-351 Bialystok, Poland, m.zukowski@pb.edu.pl

Giovanni Semprini

DIN - Department of Industrial Engineering, University of Bologna, Viale Risorgimento 2, 40136 Bologna, Italy, giovanni.semprini@unibo.it

Abstract

The increasing use of renewable energy sources reduces the operating parameters of district heating networks. As a consequence, it is necessary to reduce the supply temperature of already existing building central heating systems. Therefore, there is a doubt whether such operations will not cause deterioration of the thermal conditions of rooms, in particular with a large glazing area. A balance of jet momentum fluxes was developed between the cold down draught and warm stream generated by the heater. A mathematical correlation was formulated based on this comparison. It allows to determine the minimum average velocity of convective currents of warm air generated by the heater, which protects the occupants from the undesired local cooling. The article also includes computer code to perform this calculation in a simple and quick way. A number of experimental studies and CFD modelling were performed to determine the characteristics of the convective currents rising above the typical convector heater with widths of 96 mm, 146 mm, 186 mm, and 236 mm. The type and area of the glazing, air temperature conditions, as well as the supply temperature and the size of the convector were analysed. The results of these experiments and computer simulations were presented in the form of graphs showing the impact of the most important parameters on the minimum warm air velocity value.

Keywords: low temperature heating; convector heater; CFD, air jet momentum flux; draught risk

Nomenclature

Roman letters

e_Q – relative error of the energy balance [%],

g – gravitational acceleration [m/s^2],

Gr – Grashof number [-],

n, m – power exponents [-]

$\dot{m}$ - mass flow rate [kg/s],

$\dot{M}$ – jet momentum flux [kg·m/s^2],

R – thermal resistance [m^{2}K/W],

RME – relative maximum error [%],

RMSE – root mean square error [ºC or m/s],

T – temperature [ºC],

v – velocity [m/s].

Greek letters

β – coefficient of volumetric expansion [1/K],

ρ – density [kg/m^3],

δ_c – thickness of the cold stream layer [m],

ΔT – temperature difference [ºC]

ρ – density [kg/m^3],

ν – kinetic viscosity [m^3/s].

Subscripts

a – air,

c – cold,

δ – convective boundary layer,

e – external,

g – glass,

se – exterior surface,

si – interior surface,

w – warm,
W – window,
x, y, z – Cartesian coordinates.

1. Introduction

People spend 65–90% of their time in indoor environment and this percentage rises to above 90% in industrialized countries [1]. Due to this reason, it is essential that the thermal conditions, besides air quality, visual and acoustic aspects, inside buildings remain comfortable. Both International standards ASHRAE 55 [2] and ISO 7730 [3], as well as EN 16798-1 [4], define assessment criteria for indoor thermal comfort. While many published works report simulations and measurements of the global index PMV in different case studies, few researches have been conducted on local thermal discomfort (draught, radiant temperature asymmetry, floor temperatures, vertical air temperature differences) due to the complexity of experimental measurements and of simulations – generally requiring a CFD approach.

Indoor global and local comfort conditions mainly depend on the surface temperature of building envelope components (walls, windows) and on the characteristics and operational conditions of HVAC systems. A very important parameter, which is often omitted in the HVAC design phase, is the effect of cold draft which occurs when a convection heating system is close to large cold surfaces. Tian et al. [5] made a review on the application of BES and CFD for indoor environment applications and Hajdukiewicz et al. [6] made a simulation with a validated CFD model for a highly-glazed window room to analyse comfort conditions. Detailed analysis must be done for convector emitters depending on the configuration and operational temperature.

The strong impulse promoted by the European EPBD Directive to reduce the energy consumption of buildings has led in the last years to an increase in thermal performances of the building envelope and the adoption of efficient HVAC systems. These factors, together with the effects of climate change, lead to a reduction in winter heat loads [7] with the consequent possibility of using low-temperature heating systems, thus reducing their thermal losses and global efficiency.

Among thermal energy supply systems, fourth generation district heating (4GDH) networks are becoming more and more popular, especially in north European countries. 4GDHs are characterized by supply temperatures below 70°C, increased efficiency of the distribution system and reduced fossil fuel consumption [8]. One of the biggest advantages is the easier integration with renewable energy sources (RES), including, among others, thermal solar collectors, ground and air heat pumps, and waste heat. The use of the 4GHD network is also associated with the modernization of systems to provide space heating and domestic hot water in existing buildings. An issue that should be given special attention is the effect of low supply temperature on the characteristics of central heating terminals.

Different types of heat emitters, theoretically powered by a ground source heat pump, were investigated by Hesaraki et al. [9]. Experimental tests of convective radiators and fan coils as well as underfloor heating were carried out in a thermal chamber. The authors of this experiment determined the supply conditions of the heat emitters at an internal air temperature of 22°C. The supply temperatures were as follows: 45°C for a conventional heater, 33°C for a ventilation heater and 30°C for a floor heating system.

Liu et al. [10] determined the heating efficiency of radiators operating at low medium temperature. The results of the output power measurements were compared with the results of calculations made based on standard mathematical correlations for this type of device. As it turned out, the deviations were about 5%. The authors developed a computational model of a radiant and surface heating system powered by an air heat pump. The TRNSYS software environment was used to perform energy simulations for both systems. The results of the calculations showed that the effect of low-temperature heating with convection heaters after correcting their number is like underfloor heating.

Heat losses of low-temperature and conventional heating systems in low-energy single-family and multi-family houses were analysed by Maivel and Kurnitski [11]. Simulation calculations were carried out in the conditions of two types of climates for Northern Europe and Central Europe. It turned out that in the case of conventional radiator heating, it generated energy losses in the amount of half the heat demand in the apartment building. While low-temperature systems resulted in heat losses of 6% to 12% in the climatic conditions of Northern and Central Europe, respectively. Based on the computational analysis, the authors proposed a heating curve of 45°C/35°C for detached houses and 40°C/30°C for multi-family buildings.

Hasan et al. [12] studied the operation of a heating system consisting of radiators in rooms and underfloor heating in bathrooms. The system was supplied with a medium with a supply temperature of 45°C and a return temperature of 35°C. Its characteristics were compared with the results of a computer simulation performed for three radiator and underfloor heating systems. The results of the analysis of thermal comfort conditions showed that the operating temperature variations in all four tested heating systems are within the limits recommended in the ASHRAE 55–2004 standard. It also turned out that in the conditions of low-temperature heating, the vertical air temperature profile differed only slightly and did not affect thermal discomfort.

The characteristics of a ventilation heater made of staggered convector panels were analysed using numerical simulations, according to EN 442-2, by Ploskić et al. [13]. The tested convector was supplied with water at a relatively low temperature of 45°C. It was verified that the temperature of the air flowing through it increased by 31°C, at a volumetric flow rate of 10 l/s. In addition, it was also shown that the convector plate has a slight effect on its heat output and on the vertical profile of the air temperature in the room.

Østergaard and Svendsen [14] considered the issue of the influence of the heating system model's accuracy on the water-cooling level in radiators. The IDA-ICE computer program was used to perform energy simulations of the case house equipped with conventional water heaters. The calculation results showed that the return temperature differed by up to 16°C, depending on the assumptions and simplifications. Therefore, only a very accurate model ensured the correct simulation of the operation of the heating system.

The Ansys/Fluent software environment was used by Jahanbin and Zanchini [15] to model a heat output of the thin plane radiator and the air circulation in a room measuring 4 m by 4 m by 3 m (height). It was assumed that the radiator worked in the normal and reverse configuration, and its distance to the wall was equal to 3, 5 and 10 cm. The computational fluid dynamics (CFD) simulation results showed that the optimal distance of the radiator from the wall is 10 cm, and the normal configuration gives better thermal performance than the reverse.

The heating system's supply temperatures, which enabled thermal comfort maintenance in four Danish single-family houses, were assessed by Østergaard and Svendsen [16]. This investigation included both building energy simulations using the IDA ICE program, as well as experimental research. The results of the study proved that houses from the 1980s can be supplied with district heating water at a temperature of 45°C in Danish climatic conditions.

An innovative design of the heating system using a radiant baseboard was proposed by Shobi et al. [17]. Experimental studies and numerical simulations were used to determine the temperature distribution in the room, as well as the share of radiation and free convection in the heat exchange process of the tested radiator. It turned out that the thermal power of the new design of the radiator increased by 34% compared to the previously used solution. This was due to a significant increase in radiant heat exchange and over 45% increase in convective heat transfer. In addition, the modified radiant baseboard requires a lower supply temperature, and its length can be reduced by 50% compared to a conventional baseboard heater.

Ploskić and Holmberg [18] determined the thermal efficiency of the heating system using low-temperature baseboard heaters with an integrated external air supply system. The authors developed three analytical models and used CFD simulations to estimate the increase in temperature of the air that flowed through the radiator. It turned out that the incoming external air was heated to the set internal temperature at a supply water temperature of 45°C. The heating system tested in this work had more than twice the thermal power of a conventional baseboard heater. In addition, CFD simulations showed that the baseboard heater with integrated air supply provides thermal comfort throughout the office space during the Swedish winter season.

Hesaraki and Holmberg [19] determined the actual energy consumption of newly built single-family houses in Stockholm (Sweden). The analysed facilities had low-temperature heating systems containing ventilation and underfloor heating, heat pumps, and exhaust ventilation. Energy simulations using the IDA Indoor Climate and Energy software and measurements of heat pump energy consumption were used in this research work. Based on the results of this analysis, it was found that the buildings subjected to the tests provided thermal comfort and met the Swedish requirements related to energy consumption.

A new type of naturally aspirated convector based on a heat pipe was tested by Kerrigan et al. [20]. This type of device should be an excellent solution for systems with low supply temperatures between 35°C and 55°C cooperating with renewable energy sources. It turned out that the power density of the heat pipe convector was almost 3 times higher compared to the traditional panel heater and the serpentine fin convector. An additional advantage of this solution is its very small water capacity, which reduces the thermal inertia of the entire heating system.

Bagheri et al. [21] analysed the possibility of increasing the thermal efficiency of water baseboard heaters. New shapes of fin-tube arrays and fin-clips were subjected to experimental tests. CFD simulations were used to assess the intensification of heat transfer in the modernized design of the radiator. The new convector was characterized by an increase in efficiency of 42% at low supply temperature and 94% at high temperature compared to the convector equipped with traditionally shaped fins.

The impact of lowering the operating temperature of district heating networks and heating terminals on thermal comfort in the occupied zones should be carefully analysed. The low temperature of the heat emitters affects the convective heat transfer and, within certain boundary conditions may cause a local discomfort due to draft risk. Lin et al. [22] compare the thermal comfort of convective and radiant heating terminals in office buildings showing that convective systems result in higher draught risk and more local discomfort in the feet region due to the more temperature variation and vertical temperature gradients.

Wu et al. [23] considered the draft risk in office rooms with a large glazing area. Experimental tests were carried out using thermal vision measurements of the temperature of a mannequin located near the cold surface of the window. It was found that the draft risk was at an unacceptable level when the surface temperature of the windows was below 15°C. In this work, a predictive control algorithm dedicated to the heating system was presented to minimize the risk of drafts.

The issue of thermal discomfort that may occur in an underfloor air distribution (UFAD) system was studied by Zukowski [24]. An analytical relationship was developed to determine the minimum air velocity from the UFAD system that prevents the draft risk. The new formula was created based on the balance of momentum and energy of air streams. The mathematical equation was positively validated with the results of CFD simulation results.

The influence of a cold air draft on thermal comfort in rooms with high windows was tested by Rueegg et al. [25]. The measurement of cold air downdraft and thermal comfort level was made on a real scale in a climate laboratory. The main conclusion was that, it is not the glazing, but the window frame is the critical element that has the greatest impact on thermal discomfort in the occupied zone.

An experimental study of the draft phenomenon caused by two different types of glass panes was done by Ge et al. [26]. A 3.8 m by 6.7 m section of a curtain wall with various glazing systems incorporated was tested in a climate chamber. The influence of thermal parameters of windows, frames and mechanical ventilation in winter climatic conditions was investigated. Test results have shown that fan operation can create a draft risk in accordance with ASHRAE requirements if people are closer than 2 m to the wall.

Wu et al. [27] analysed the airflow fields over a radiator placed against a wall with a window in a central heating system using thermal image velocimetry. The result of this work is a second-order polynomial correlation that can be used to judge whether overheating is occurring based on the calculated speed reference.

Embaye et al. [28] studied the impulse flow of the medium supplying the radiators in terms of increasing the heating power of the radiators. The air temperature distribution and its velocity profile in the room were determined using CFD simulations. Based on the analysis of the calculation results, it was found that a 25% increase in the heat output of the heater can be obtained at the frequency from 0.0083 Hz to 0.033 Hz. This method of supplying the central heating system did not adversely affect the reduction of thermal comfort as defined by the ASHRAE 55 and EN ISO 7730 standards.

As the review of the literature showed, the subject of low-temperature heating is up-to-date, and many research centres deal with it. As mentioned above, there are many advantages to lowering the temperature of the medium supplying the radiators. However, possible energy savings should not affect the thermal environment deterioration, the parameters of which are regulated by European Standard 16798. The main reason that may reduce the thermal comfort index is the unwanted local convective cooling of the human body caused by air movement.

Large length hot water convector radiators are commonly used, especially in cold regions, in office buildings, winter gardens, and above all, near shop windows, especially in spaces with large glazing areas. This type of heaters made as floor-mounted or hidden under the floor covering, has not been extensively analysed in previous literature. Problems arise with the danger of deterioration of the thermal environment quality due to the continuous trend of lowering the temperature of water-based heating systems. The purpose of this scientific study was to assess the operating conditions for different sizes of convector heaters at a supply temperature below 45°C in order to optimise local comfort conditions and to avoid draft risk due to cold air flowing down the window surface. This article presents the results of experimental tests and CFD simulations on the characteristics of the heater and analytical calculations of the movement of air streams near an external wall with a window.

The analysis presented here was aimed at developing a new formula that will allow the designer to check whether the characteristics of the convector will protect the room against the unwanted local convective cooling. To achieve the intended goal, both comprehensive experimental studies and CFD simulations were carried out. The results of this study contain elements of novelty that allowed for a better explanation of the impact of the low-temperature heating system on the conditions of the indoor thermal environment of rooms.

The novelty and originality of this paper are contained in several statements presented below:

- A three-dimensional profile of air temperature and velocity directly above the convector heater at low supply temperature was developed. As the literature review showed, no similar experimental research results have been published so far.
- A CFD model of a room with a convector as a heat source was created. An element of novelty in this type of modelling was the use of the substitute model of the heating coil and its calibration based on experimental results. This made it possible to reduce the density of

the computational mesh while maintaining the accuracy of mapping the studied physical phenomena.

- The analysis of the nature of the air flow through the convector heater allowed the authors to find a large irregularity of the velocity field at the inlet to the heating coil when reducing the supply temperature. This phenomenon may result in a decrease in thermal power. This is an interesting and so far unexplored phenomenon, which will contribute to further research on the modification of the radiator casing and the intensification of the heat transfer.
- A balance of the two jet momentum fluxes of a cold down draught and a warm air generated by the convector was made. On this basis, a new mathematical relationship was developed. It allows to estimate the minimum air velocity generated by the convector heating coil, which can protect the room against the risk of a decrease in thermal comfort.
- The applied aspect and the element of novelty in this paper is the development of charts enabling the determination of the minimum velocity of air flowing from the convector. A computer code that allows analysis of the operation of low-temperature convector heating, which can be used in the design of HVAC systems, was also written and attached.

2. Materials and methods.

In rooms with a large area of glazing, cold down-draughts are formed in the winter. A rising stream of warm air must be generated in order to protect the occupied zone from the risk of draught. The thicknesses of these convective air layers are small in relation to the dimensions of the room. Thus, when investigating this type of aerodynamic phenomena, we can treat it as free streams. At a certain height, the cold falling and warm rising streams are merging. The mixed air stream moves perpendicular to the wall and depending on the energy and mass balance, it can then go down or up. If the resulting jet is too cold and flows towards the floor, there may be a risk of local cooling. This is an unfavourable situation and the heating system should be designed in such a way that the kinetic energy of the hot stream compared to the kinetic energy of the cold stream should be at least equal.

This research, the results of which are presented in the current article, consisted of three main stages. The first of these was the development of an analytical model, which was preceded by an in-depth literature review. Then, experimental studies and CFD simulations were performed to determine the necessary coefficients for the mathematical correlation. In the last step, the parameters that should be met by convectors used in low-temperature heating systems were developed. The successive stages of this study are shown in the block diagram in Fig. 1.

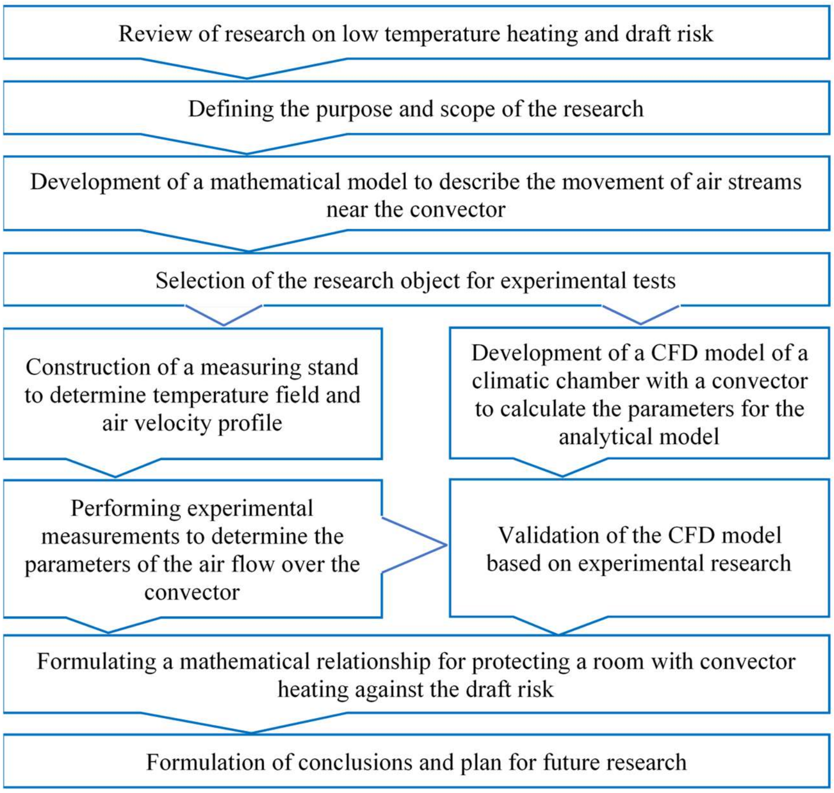

Figure 1. Research methodology used in this study.

2.1. Jet momentum fluxes balance - a mathematical model.

The risk of deterioration of the thermal environment was determined on the basis of comparing the values of two jet momentum fluxes $\dot{M}$, which are the product of the mass flow rate $\dot{m}$ and the average velocity $\bar{v}$. The names of the variables used in the modelling of air streams are shown in Figure 2.

The jet momentum flux for a cold down draught is equal to:

$$\dot{M}_c = \dot{m}_c \bar{v}_c. \tag{1}$$

The rest of this study will be conducted for 1 m width, so the mass flow rate can be expressed by the following relationship:

$$\dot{m}_c = \delta_c \rho \bar{v}_c. \tag{2}$$

where:

δ_c – thickness of the cold stream layer [m],

ρ – air density [kg/m^3] (assumed in the calculations:

$\rho = -0.0039T_a + 1.2831$ (correct in the temperature range from 10°C do 50°C). (3)

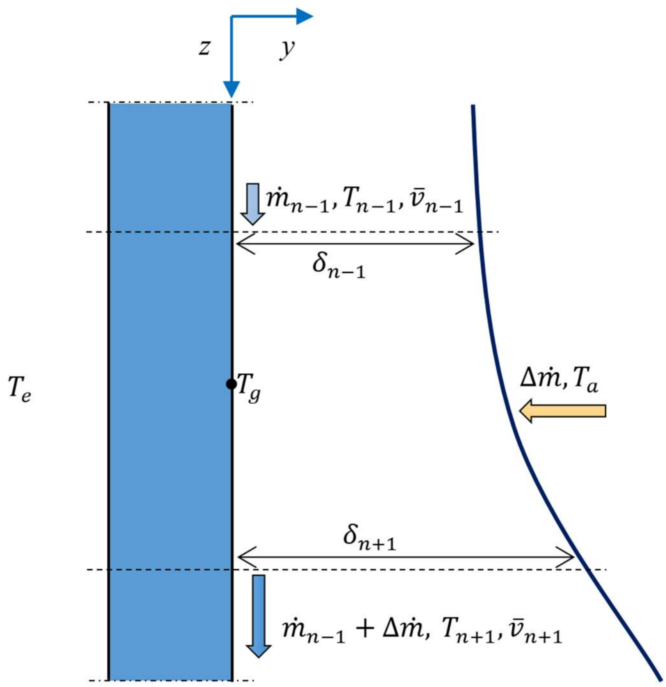

Figure 2. Axis system and specification of variables.

As a result of expressing a mass flow as a function of an average air velocity, we get the following equation:

$$\dot{M}_c = \delta_c \rho_c \bar{v}_c^2. \tag{4}$$

The value of v_c of the cold air stream moving by the window can be determined from Eq. 5, which describes the velocity distribution profile in the convective boundary layer.

$$v_c = v_\delta \left(\frac{y}{\delta_c}\right)^{1/n} \left(1 - \frac{y}{\delta_c}\right)^m, \tag{5}$$

where:

n=2, *m*=6 - power exponents estimated experimentally by Shia-hui and Peterson [29] for the Grashof number (Gr) ranging from 10^8 to 10^{10},

y – axis perpendicular to the wall [m].

The average air velocity can be obtained as a result of integrating the above power profile over the jet thickness from 0 to δ_z.

The characteristic thickness δ^*, determined by Eckert and Jackson [30], was used to calculate the thickness of the convective layer:

$$\delta_c = 3.138 \mathrm{Gr}_z^{-0.076} \delta^*, \tag{6}$$

where:

$\delta^* = z \mathrm{Gr}_z^{-0.1}$,

z – axis parallel to the window and directed to the floor [m],

$\mathrm{Gr_z} = \frac{g \beta \Delta T z^3}{\nu^2}$,

g – gravitational acceleration [m/s^2],

β – coefficient of volumetric expansion [1/K],

$\Delta T = T_a - T_g$ - the difference between the temperature of the air in the room T_a and the surface of the glass T_g,

ν – kinetic viscosity [m^3/s].

In order to calculate the temperature of the glazing on the side of the heated room, the following relationship was derived and applied in this analysis:

$$T_g = T_a - \frac{R_{si}}{R_{si}+R_W+R_{se}}(T_a - T_e), \tag{7}$$

where:

R_{si}, R_{se} – thermal resistance at interior and exterior surfaces [m²K/W],

R_W – thermal resistance of the window [m²K/W],

T_e - external air temperature [°C].

Another variable v_δ can be expressed as a function of the Grashof number and the characteristic velocity defined by Cheesewright [31].

$$v_\delta = 5.425\mathrm{Gr}_z^{-0.076}(g\beta\Delta Tz)^{1/2}. \tag{8}$$

The jet momentum flux for streams of a warm air generated by the convector can be determined as before, from the following relationship:

$$\dot{M}_w = \delta_w \rho_w \bar{v}_w^2. \tag{9}$$

This time, the values of $\bar{v}_w$ and δ_w of the warm air streams convectively rising upwards were determined by means of experimental studies. This was due to the fact that no description of a similar case was found in the scientific literature.

In order to minimize the risk of thermal discomfort, the value of the cold air stream should be lower than that of the warm air jet. In this study, it was assumed that the directions of the velocity vectors of both air streams are opposite so that the following math inequality can be written:

$$\dot{M}_w > \dot{M}_c, \tag{10}$$

or

$$\bar{v}_w > \left(\frac{\delta_c \rho_c}{\delta_w \rho_w}\right)^{1/2} \bar{v}_c. \tag{11}$$

The thickness of the cold air stream occurring in the above relationship will be equal:

$$\delta_c = 3.138\mathrm{Gr}_{H_w}^{-0.176} H_w. \tag{12}$$

Finally, the mathematical correlation for protecting a room with a convector heating against the draft risk is as follows:

$$\bar{v}_w > 1.77\mathrm{Gr}_{H_w}^{0.324}\left(\frac{H_w\rho_c}{\delta_w\rho_w}\right)^{1/2}\bar{v}_c. \tag{13}$$

Many factors influence the accuracy of the above mathematical correlation. First of all, it is dedicated primarily to convector heaters. The mathematical analysis was performed assuming the absence of any obstacles in the path of air streams, such as window sills. It was also assumed that the radiator is close to the wall or window. This is quite a general statement. Unfortunately, determining the maximum distance, at which the above formula will give correct results, requires extending this analysis. Additional CFD simulations or experimental tests require determining the influence of the radiator length on the uniformity of the generated warm air stream. In the case of short convectors, noticeable deformations of air streams may occur on both edges of the radiator. This type of phenomenon is definitely less important in the case of long convectors.

It was also assumed that both streams will meet at a height of 5 cm above the convector, and the falling stream flows down the window pane with a height of H_w. The choice of the height of 5 cm was determined primarily by the diameter of the wire cover of the sensors (10 cm), which protected device against damage. In the authors' opinion, the most unfavourable conditions will occur when both air streams meet at the level of the convector outlets. In this case, cold drafts will have the greatest thickness and the jet momentum flux of the warm air will be the lowest. Therefore, this analysis concerned the most unfavourable conditions. Apart from that it was assumed that the window height H_w is measured from the level of the convector outlet. Eq. 13 can be used to analyse air flows in rooms for which thermal conditions and geometric dimensions can be described by a Grashof number ranging from10^8 to 10^{10}. In the case of mechanical ventilation and convectors equipped with fans, the relationship developed above should not be used.

Manufacturers of such devices do not provide information on the velocity of the air stream over the radiator as a function of the supply temperature. The size (heat power) of the convectors is determined by designers based on the heat load of the occupied zone. However, in the case of large glazed areas and a low supply temperature, it is necessary to ensure not only the design temperature of the indoor air, but also to protect the room against the risk of a local convective cooling. Therefore, a number of experimental studies were performed to determine the value of v_w, as described in the next section of this paper.

2.2. Experimental research on the characteristics of the warm air stream generated by the converter.

2.2.1. Description of the experimental test object.

In this study, the test object was a typical convector with external dimensions: length - 1600 mm, height - 140 mm and width - 236 mm. Its heat exchanger was made of copper tubes and aluminium fins. The dimensions of the heating coil were as follows: length - 1350 mm, height - 100 mm and width - 200 mm. The thickness of the fins was 0.1 mm, the diameter of the tube was 16 mm, and the spacing of the fins was 6.67 mm. The convector sheathing was made of steel sheet without contact with the heat exchanger. A photograph of the device that was tested is shown in Figure 3.

2.2.2. Description of the measurement stand and the scope of the experiment.

The aim of the experimental study was to determine the basic parameters of jet momentum flux ($\bar{v}_w$ and δ_w), which is generated by the convector. For this purpose, the heater was placed in a climatic chamber used to stabilize the constant air temperature with a very high accuracy of ±0.1°C. The AirDistSys5000 system was used to measure temperature and air velocity. Six SensoAnemo5100SF transducers with an omni-directional (spherical) air velocity sensor were used in this study. A special aluminium coating, which is applied under vacuum, reduces the impact of thermal radiation on the accuracy of the measurement and increases resistance to pollution circulating in the air. The manufacturer of these sensors provides the following parameters:

- Measurement velocity range: 0.05-5 m/s.
- Velocity measurement accuracy: ±0.02m/s ±1% of reading range.
- Automatic temperature compensation: lower than ±0.1%/K.
- Temperature measurement accuracy: ±0.2°C.

SensoBee 485 wire-less transmitter was used for serial connection of sensors and radio data transmission to SensoBee receiver with USB port for connection to a computer. Measurements were recorded every 10 seconds. Two sets of sensors, shown in Figure 3 (variant 1 - left side and variant 2 – right side), were used to spatially determine the velocity and temperature field above the convector. In the case of the first arrangement, the sensors were moved along the convector, collecting measurements in 9 positions. In the second set of sensors, the measurement was made along the axis of the convector. The supply temperature of the heater was stabilized on three levels 35°C, 40°C, and 45°C with an accuracy of ±0.2°C.

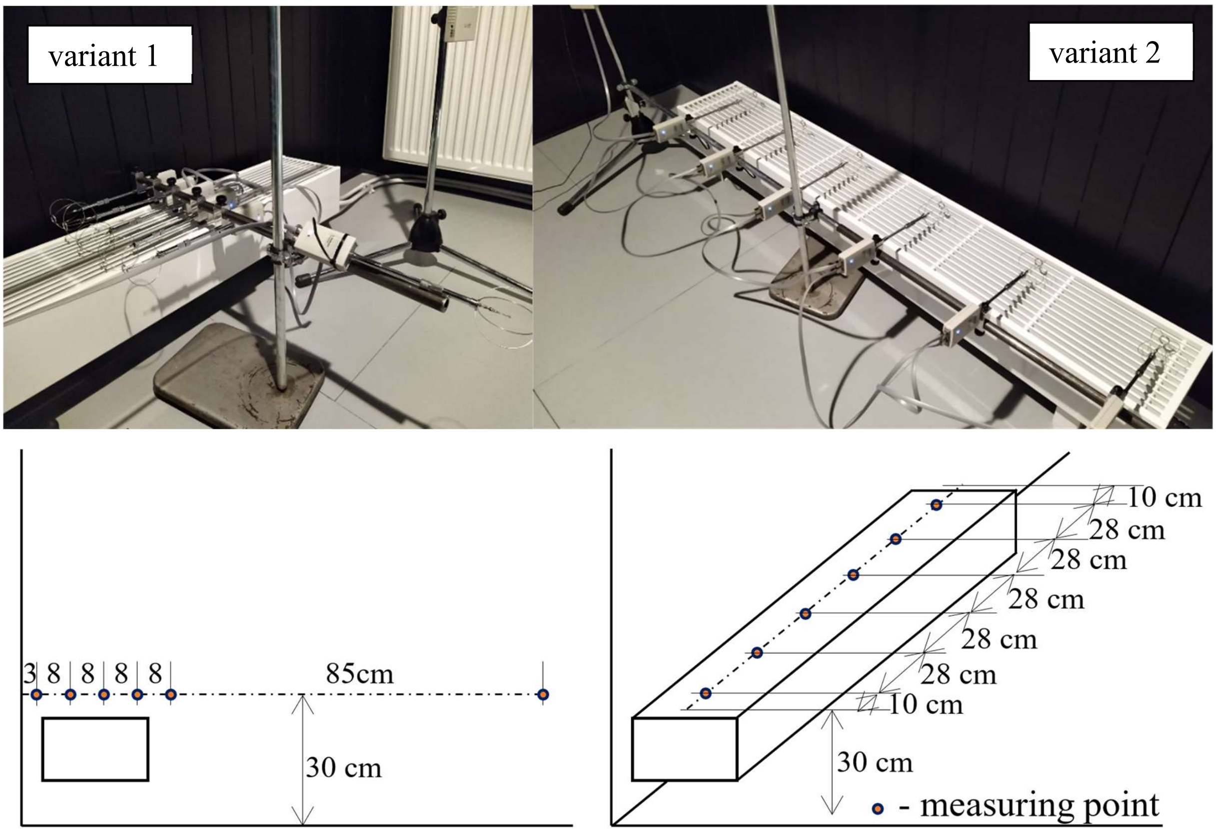

Figure 3. Two variants of setting the measurement sensors.

2.3. Development of computational fluid dynamics model.

The distribution of air movement in the climatic chamber caused by the operation of the convector heater was modelled using the Ansys/Fluent 2021 R2 software. The CFD model reflects the actual size of the tested object, the dimensions of which are shown in Figure 4. Due to the complicated shape of the heating coil, a substitute model of this element was used. It was developed in such a way as to obtain a positive validation result. It was decided to create a two-dimensional model, because in this analysis the main element to be determined is the profile of temperature and air velocity directly above the convector.

Four models were developed, covering the following range of convectors with widths of 96, 146, 186 and 236 mm. The widest unit was subjected to experimental research.

The mesh of the model was generated as regular in the area of the heat exchanger and as unstructured in the places where the heater connects to the air region. Three types of mesh with different density were generated, the parameters of which are shown in Table 1.

Table 1. Parameters of computational meshes developed to test their accuracy.

Mesh No.	Cells	Faces	Nodes	Temperature [ºC]	Velocity [m/s]	e_Q [%]
1	320132	641923	321792	21.214	0.04612	4.3
2	797591	1597903	800278	21.121	0.04580	1.2
3	1280194	2563710	1282963	21.115	0.04534	0.9

The change in average temperature and the air velocity in the Fluid computational subdomain as well as the relative error of the energy balance e_Q (Total Heat Transfer Rate) after 1000 iterations were used to assess the quality of the mesh. It was decided to use mesh number 3 for further computational analysis.

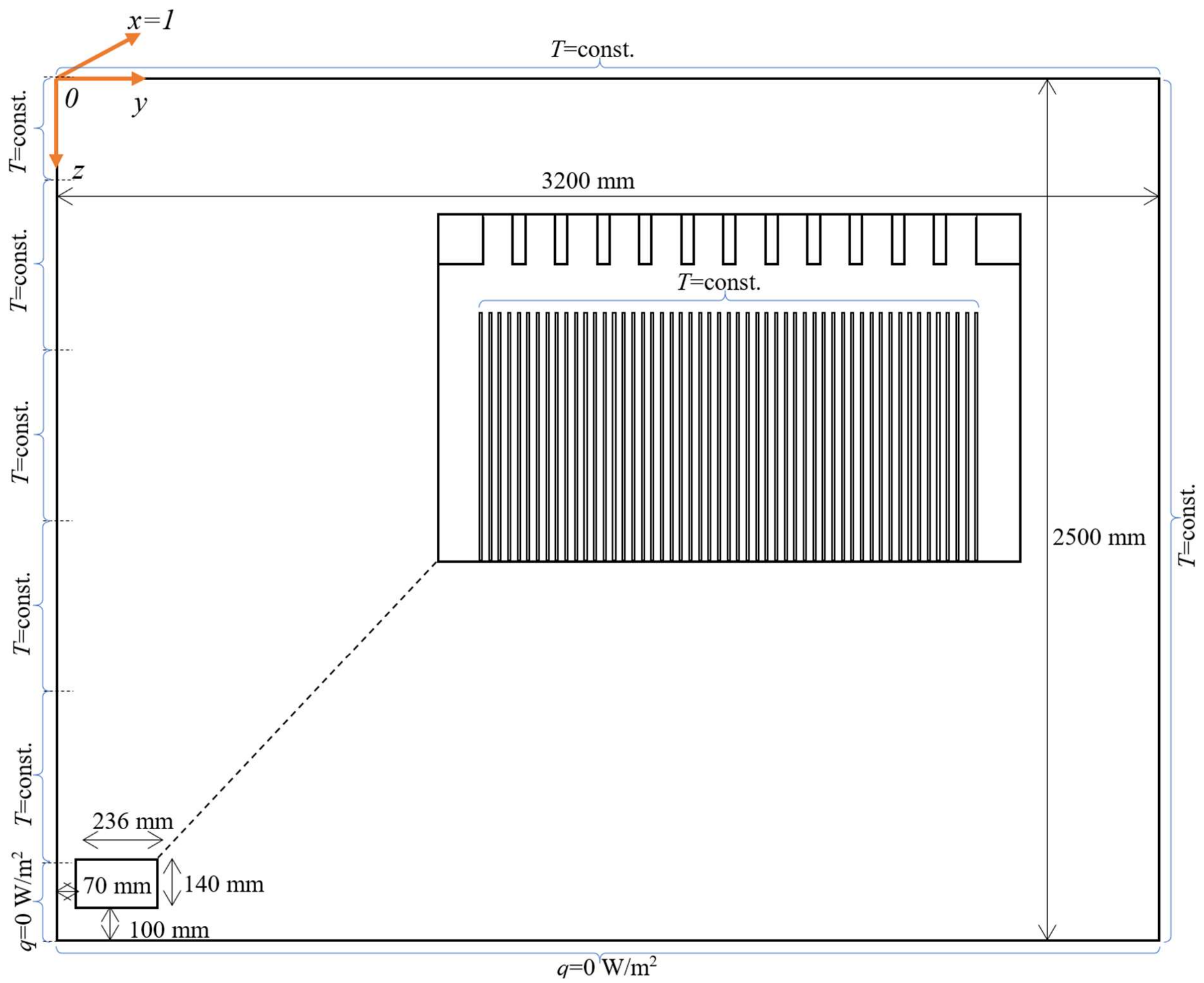

Figure 4. The calculation domain and main dimensions of the model elements.

The mechanism of heat exchange is free convection because the convector used for the tests was not equipped with a fan. This phenomenon was analysed under steady-state conditions, due to the assumed scope of this study. The Grashof number, which describes the phenomenon of free convection occurring in the tested chamber, is greater than 10^9, so a turbulent character of the fluid motion was assumed. The k-ε turbulence model was adopted, as well as the Realizable and Wall Treatment options. Air was treated as an ideal gas in an incompressible form. A constant wall temperature was the basic boundary condition at the domain boundaries as well as at the surface of the heating coil. The unit heat flux equal to 0 was declared on the floor of the chamber. The Pseudo Transient algorithm using the sub-relaxation method was adopted to solve difference equations. Pressure discretization was performed using the Body Forced Weighted scheme, and the QUICK algorithm was assumed for the remaining variables.

2.4. Validation of CFD simulations.

The assessment of the accuracy of CFD simulations was performed by comparison with experimental data. An illustration of this validation is shown for a supply temperature of 40°C in Figure 5 (temperature) and Figure 6 (velocity).

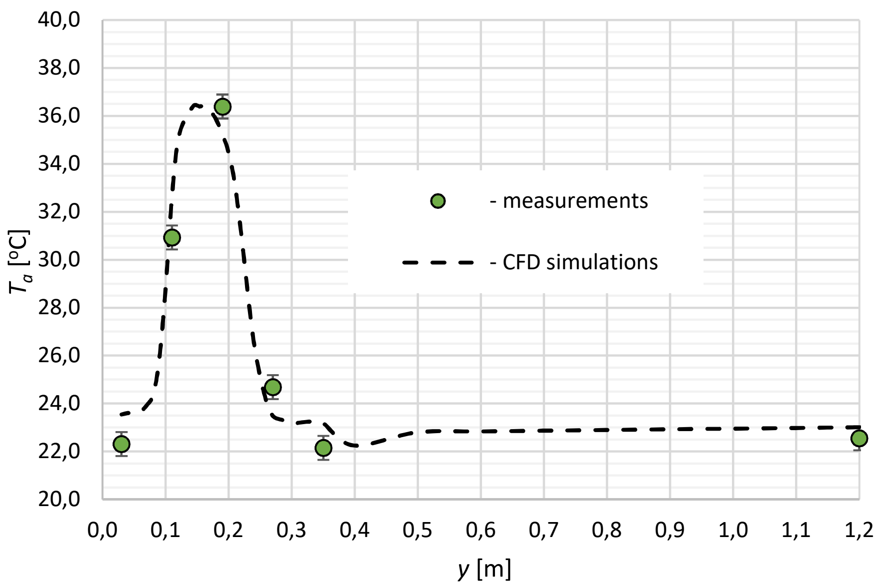

Figure 5. Comparison of air temperature measurements and the results of computer simulations at the convector supply temperature of 40ºC.

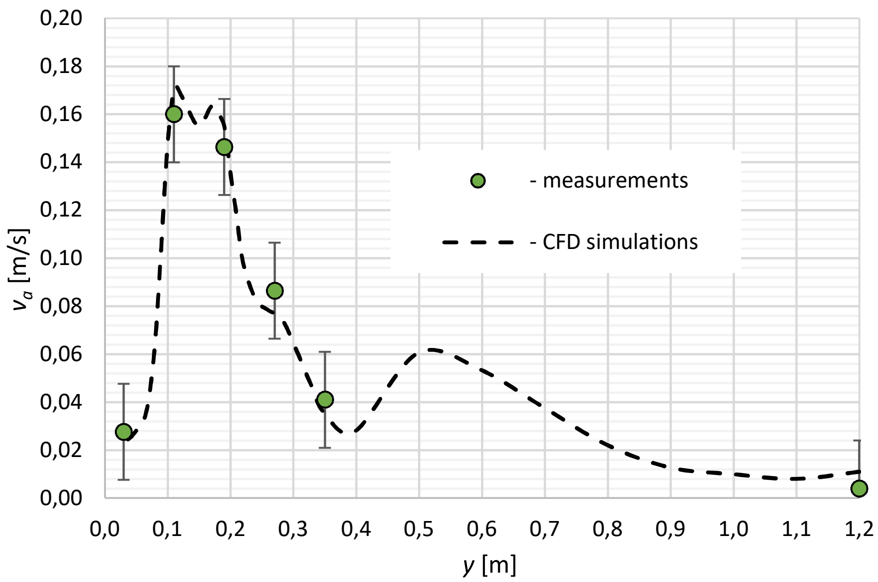

Figure 6. Comparison of air velocity measurements and the results of computer simulations at the convector supply temperature of 40ºC.

It can be said that the simulation results quite accurately reflect the velocity profile and temperature distribution of the air stream flowing out of the convector. The calculated air velocity values are within the measurement accuracy range. To estimate the degree of convergence of CFD simulations and measurements in a measurable way, the following two relationships were used:

Relative maximum errors:

$$\mathrm{RME}_{T_a} = \max_{i=5}\left\{\left|\frac{T_{a,m}-T_{a,CFD}}{T_{a,m}}\right| \cdot 100\%\right\}, \tag{14}$$

$$\mathrm{RME}_{v_a} = \max_{i=5}\left\{\left|\frac{v_{a,m}-v_{a,CFD}}{v_{a,m}}\right| \cdot 100\%\right\}, \tag{15}$$

Root mean square errors:

$$\mathrm{RMSE}_{T_a} = \sqrt{\frac{\sum_{i=1}^{5}\left(T_{a,m}-T_{a,CFD}\right)^2}{5}}, \tag{16}$$

$$\mathrm{RMSE}_{v_a} = \sqrt{\frac{\sum_{i=1}^{5}\left(v_{a,m}-v_{a,CFD}\right)^2}{5}}. \tag{17}$$

The relative maximum error was 6%, 6.5%, and 8% for the temperature calculations and 15.2%, 15.4%, and 17.7% for the air velocity estimation at the convector supply temperature of 35°C, 40°C, and 45°C, respectively. Whereas the average accuracy described by the RSME error was 1.1°C, 1.4°C, and 1.5°C for the temperature calculations and 0.0064 m/s, 0.008 m/s, and 0.0104 m/s for the air velocity estimation at the convector supply temperature of 35°C, 40°C, and 45°C, respectively. Two regularities were observed. The discrepancy between measurements and calculations increased with the increase of the supply temperature and the highest relative errors of velocity estimation appeared at the boundaries of the air stream due to their very low value.

Summing up, it can be concluded that the prediction of air velocity and temperature distribution using CFD simulations is at an acceptable level.

3. Results and discussion.

3.1. Results of experimental studies and CFD simulations.

The results of air velocity and temperature measurements were presented in two forms. Figures 7 and 8 show profiles perpendicular to the wall in the middle of the length of the radiator for three supply temperatures. The spatial distribution of the velocity and temperature fields of the warm air, 5 cm above the radiator outlet, are demonstrated in Figures 9 and 10.

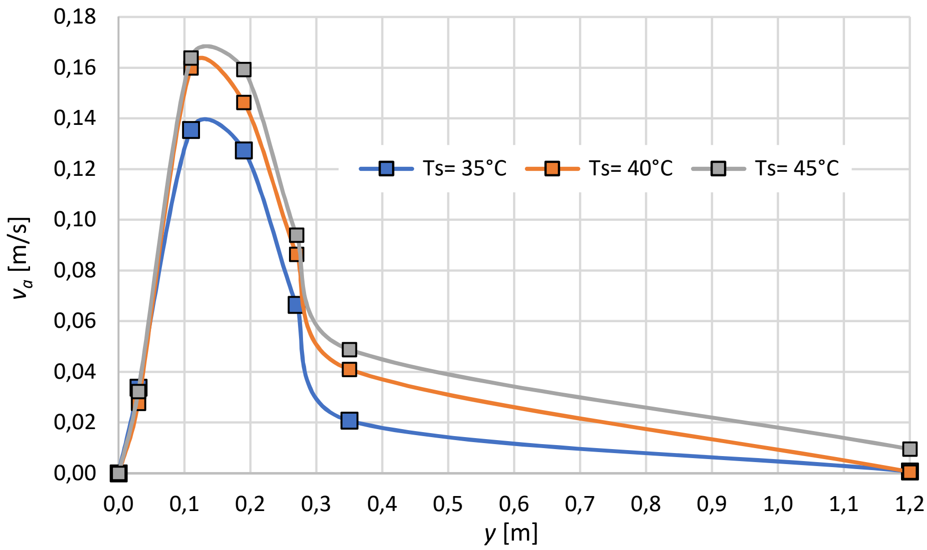

Figure 7. Perpendicular to the wall profile of air velocity distribution in the middle part of the radiator.

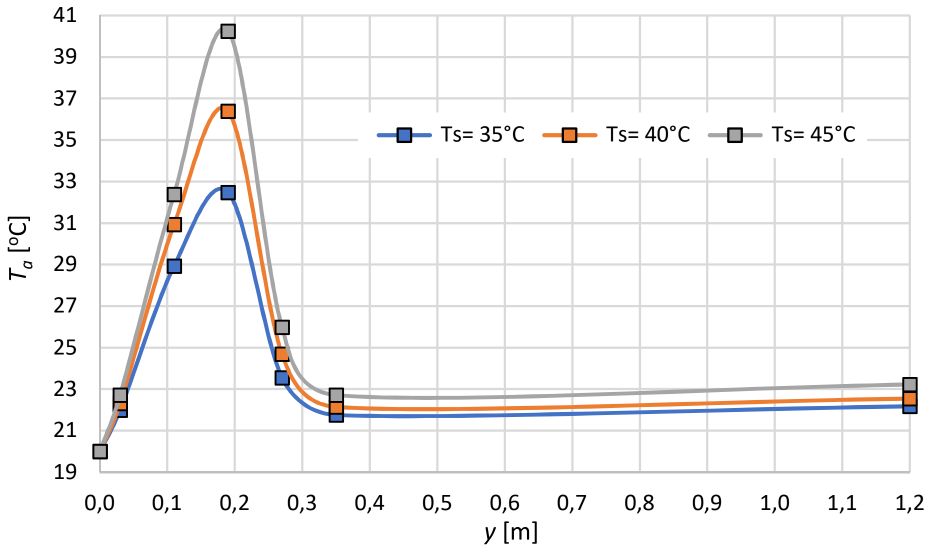

Figure 8. Perpendicular to the wall profile of air temperature distribution in the middle part of the radiator.

Based on the analysis of the air velocity field above the convector (Figure 9), it can be concluded that its distribution is relatively uniform along the radiator. However, the shape of the temperature field was slightly different (Figure 10), as the highest values appeared in the middle part of the convector. Both graphs were prepared in the Statistica software using the spline method. It should be noted that these fields have been "flattened" as a result of polynomial interpolation of the measurement results.

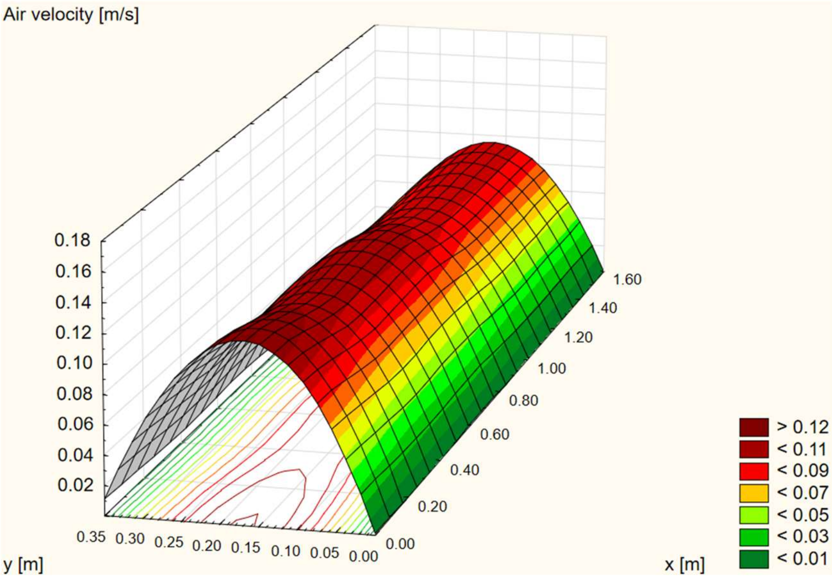

Figure 9. Two-dimensional velocity field over a convector at a water supply temperature of 35°C.

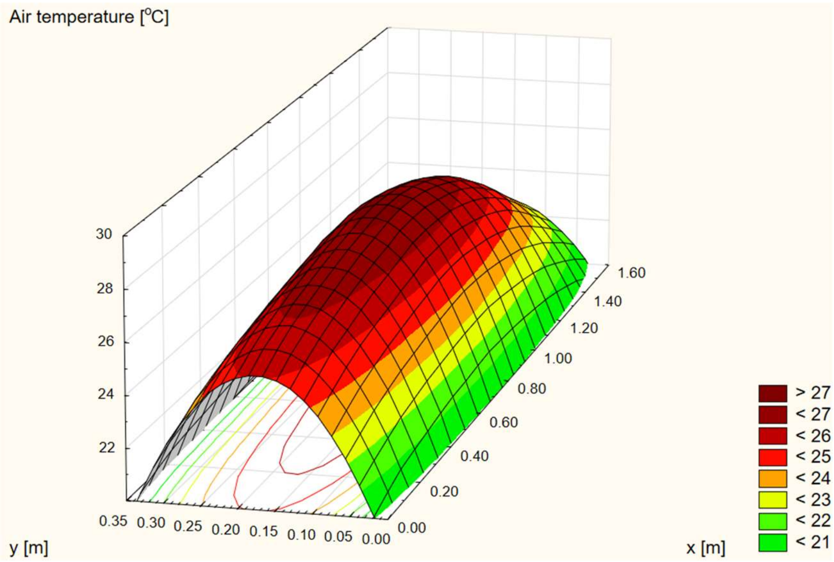

Figure 10. Two-dimensional temperature field over a convector at a water supply temperature of 35°C.

The average values of air velocity and temperature were calculated from all measurement points and shown in Table 2.

Table 2. Average values of velocity and air temperature above the tested convector.

Convector supply temperature T_s [°C]	Average air velocity $\bar{v}_w$ [m/s]	Average air temperature $\bar{T}_w$ [°C]
35	0.10	27.9
40	0.12	30.3
45	0.13	32.8

The analysis of the results of the CFD simulation was used to determine the average value of air velocity and temperature (Table 3) for the remaining convectors.

Table 3. Average values of velocity and air temperature over the other convectors.

Convector supply temperature [°C]	Average air velocity $\bar{v}_w$ [m/s]	Average air temperature $\bar{T}_w$ [°C]
Convector with a width of 96 mm		
35	0.08	24.2
40	0.09	25.0
45	0.10	26.3
Convector with a width of 146 mm		
35	0.09	25.3
40	0.10	26.7
45	0.11	28.4
Convector with a width of 186 mm		
35	0.09	26.5
40	0.11	28.3
45	0.12	30.3

Conventional limits of the range of the stream can be determined on the basis of the analysis of the direction of air outflow from the convector's grill. Figure 11 shows exemplary velocity vector profiles for the widest (236 mm – left side) and narrowest (96 mm – right side) convectors in order to show the effect of the casing width on the character of the air flow. In addition, there are other influences that affect the width δ_w of the air flow coming out of the convector. These factors include, first of all, the design of the grill, the temperature of the water supplying the radiator, the distance of the convector from the floor, and wall as well the temperature of the surface of these partitions. Therefore, the value δ_w was assumed to be equal to the casing width.

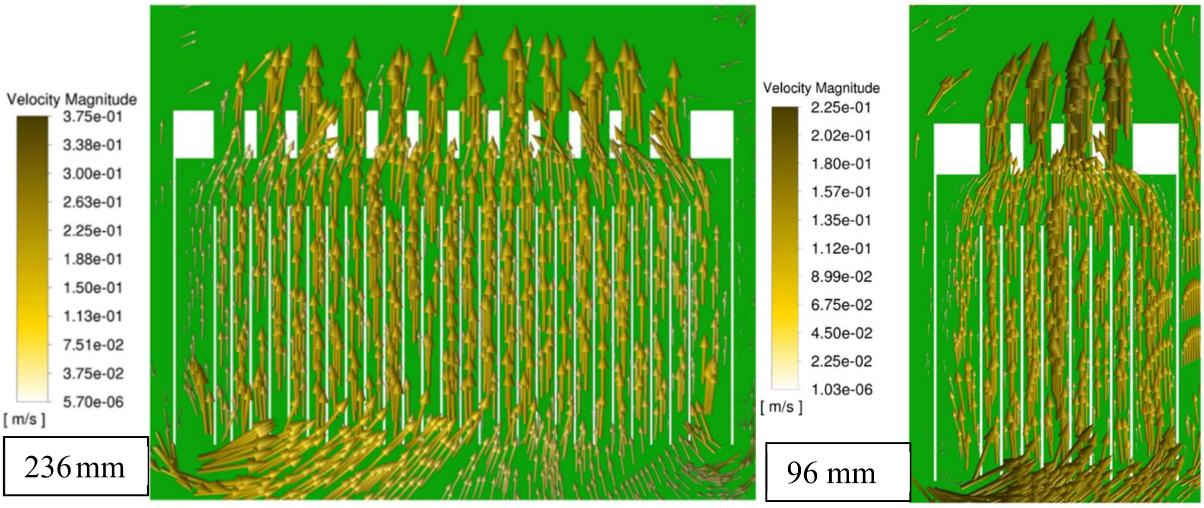

Figure 11. Velocity vector profile for a convector for a supply temperature of 35°C.

3.2. Calculation results of the recommended minimum velocity of the warm air stream.

In order to graphically illustrate the formula for the minimum velocity of the warm air stream (Eq. 13), a number of calculations were made, and the results of which may be of practical importance.

A specially dedicated C++ code using console I/O streams for this purpose was written. It uses the numerical integration technique based on the rectangle method to determine the average velocity of a cold air stream moving by the window (Eq. 5). A listing of this computer program is given in Appendix A of this article. After appropriate modifications, it can be used for a detailed analysis of all parameters affecting the interaction of warm and cold air streams.

The scope of calculations included three types of glazing: double-pane, triple-pane, and quadruple-pane windows. The thermal resistance of these windows was: 0.612, 0.933, and 1.241 m^2K/W, respectively [32]. The velocity of hot air above the convector v_w (dashed lines) was compared with the minimum value (solid lines) determined from Eq. 13, as shown in the graphs below. Four values of the temperature difference between the air inside the room and the outside air: 15°C, 20°C, 30°C, and 40°C, and three values of the convector supply temperature: 35°C, 40°C, and 45°C were included in the analysis.

As can be seen from the Figure 12, a convector with a width of 96 mm can only protect a room with double glazing windows against cold air flowing down the window surface at a minimum supply temperature of 45°C. However, if we use a quadruple glazing system, it will be possible to reduce the supply temperature of the radiator to 35°C.

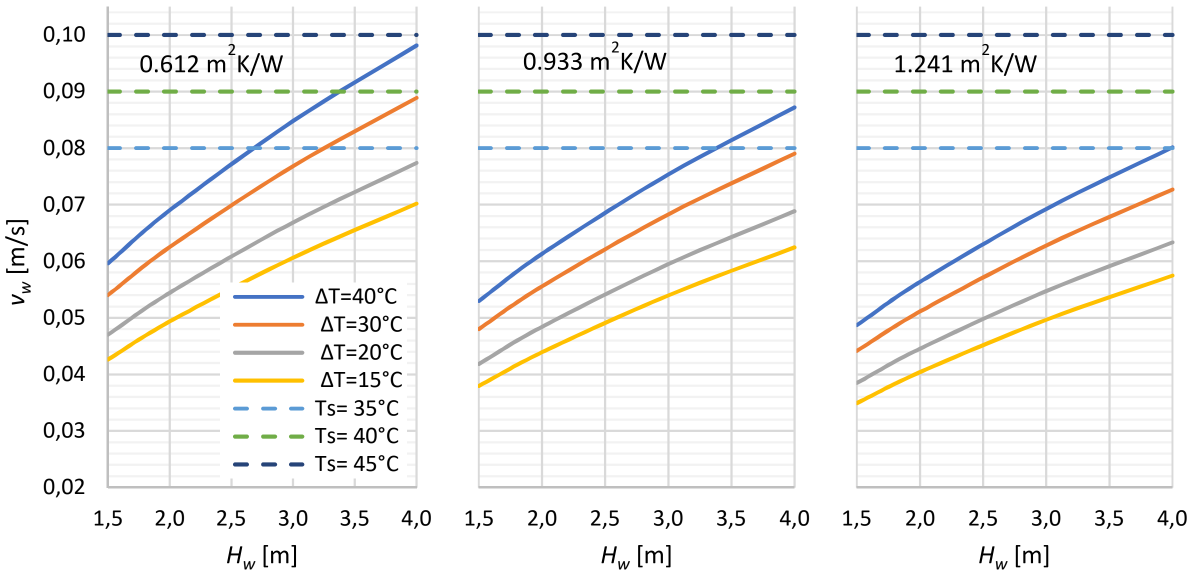

Figure 12. Comparison of the velocity of warm air over the convector with a width of 96 mm and the thermal resistance of the window 0.612 m^2K/W, 0.933 m^2K/W, and 1.241 m^2K/W.

For the purpose of comparison, an analogous graph (Figure 13) was prepared for the widest convector with a size of 236 mm. It should be noted that this type of radiator can operate at a low supply temperature without fear of deterioration of thermal comfort in a room with a large glazing area.

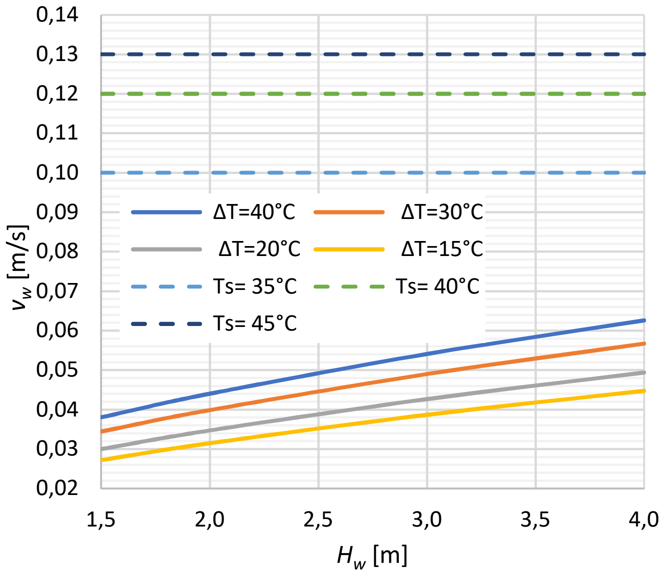

Figure 13. Comparison of the velocity of warm air over the convector with a width of 236 mm and the thermal resistance of the window 0.612 m^2K/W.

Figure 14 shows the influence of the thermal resistance of the glazing on the minimum required air velocity over the convector. The use of double-glazed windows instead of triple-glazed windows increases the minimum average velocity v_w by about 11%, and in the case of quadruple-pane windows, the v_w value increases by about 18%. More noticeable differences appear when we consider the influence of the width of the convectors (Figure 15). The decrease in the value of v_w is about 19%, 28%, and 36% when instead of the narrowest convector we use radiators with a width of 146 mm, 186 mm, and 236 mm, respectively.

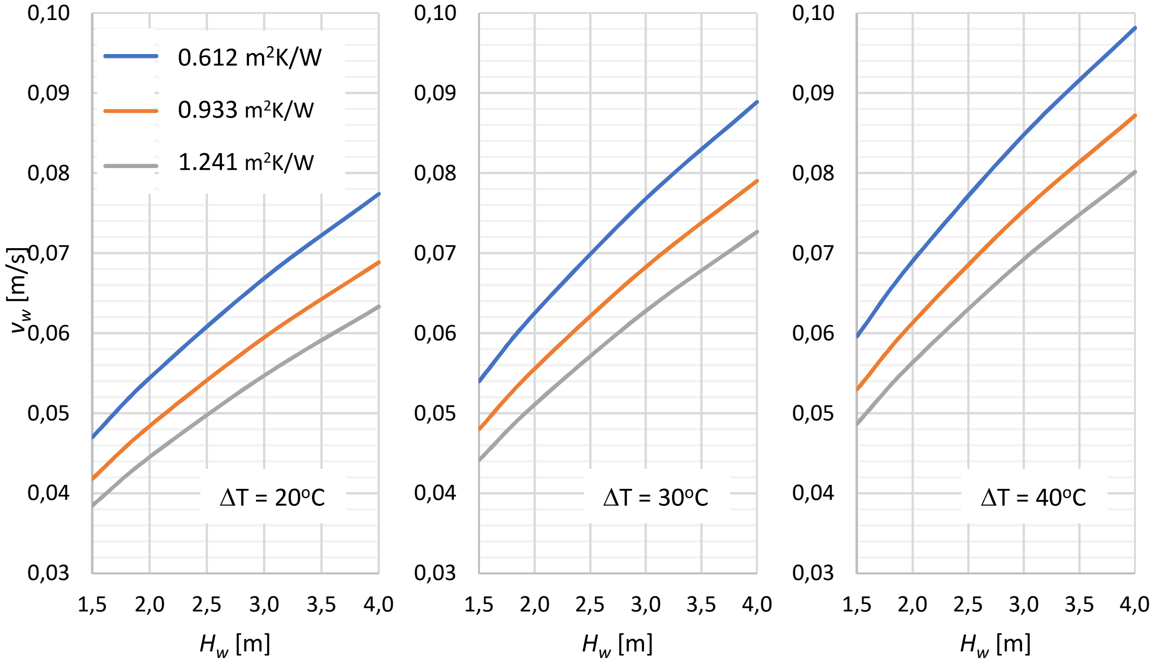

Figure 14. Comparison of the minimum velocity of warm air over the convector with a width of 96 mm for a temperature difference of the air inside the room and the outside air equal to 20°C, 30°C, and 40°C.

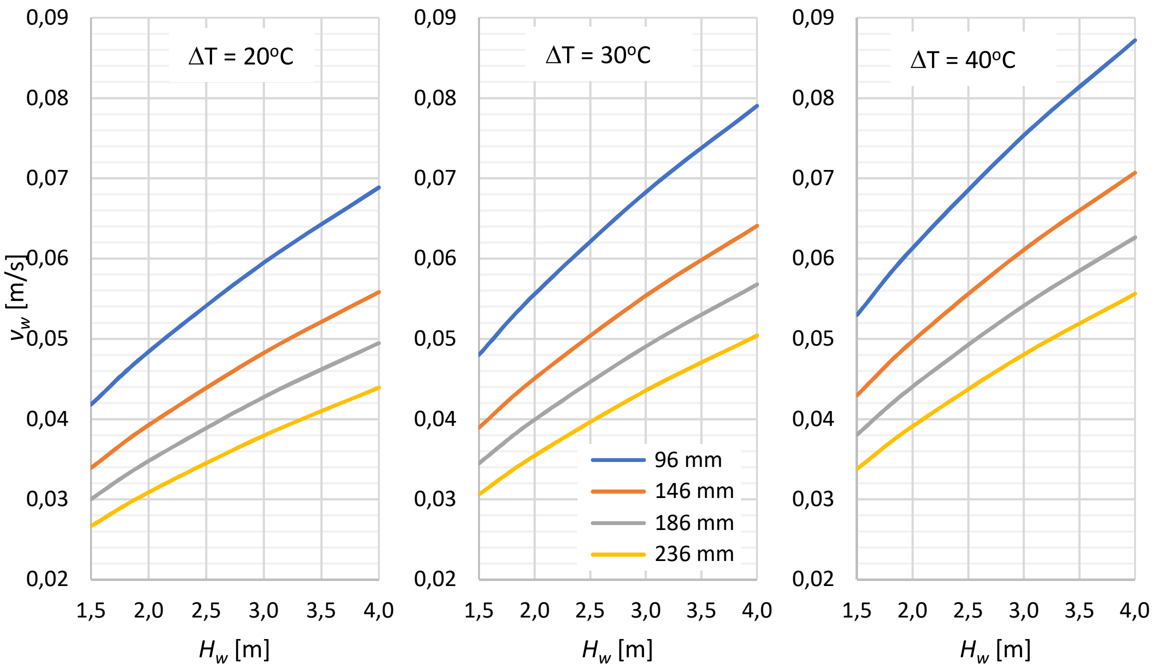

Figure 15. Comparison of the minimum velocity of warm air over the convector of different widths for a temperature difference of the air inside the room and the outside air equal to 20°C, 30°C, and 40°C.

4. Summary and conclusions.

The main purpose of this study was to determine selected operating parameters of a convector heater located in a room with a large glazing area, in terms of the possibility of a risk of local deterioration of thermal conditions. The reason for undertaking this analysis was the continuous trend of lowering the operating temperature in district heating networks. The consequence of this is also the reduction of the supply temperature of central heating systems in buildings.

A balance of jet momentum fluxes of the cold down draught and warm stream flowing from the radiator was made. An element of novelty is the development of an analytical relationship (Eq. 13), which allows estimation of the minimum air velocity v_w generated by the heating coil of the convector. This parameter allows a correct design of convectors and their operating mode in order to protect the room against undesired local convective cooling caused by air movement. Computer code (Appendix A) was written to quickly perform calculations using the formula (Eq. 13) proposed in this article.

A series of experimental studies and CFD simulations of a typical convector heater supplied with water at 35°C, 40°C, and a maximum of 45°C were carried out. Experimental determination of the temperature and velocity of air distribution above the convector and the development of a substitute CFD model of convectors of different widths should be considered as elements of novelty.

The results of measurements and computer calculations provided the necessary data to assess the convectors' operating parameters in conditions of low supply temperature. The article contains graphs that allow to determine whether the selected type of convector will meet the criterion of the minimum velocity of the warm air stream generated by its heating coil. The height of the windows, three types of glazing (double, triple and quadruple glazed windows), the temperature difference between the outside and inside air, as well the width and supply temperature of the convector are the parameters that were considered for this analysis.

It was confirmed that, four-pane and three-pane windows, compared to double-glazed ones, reduce the minimum value v_w by about 18% and 11%, respectively. As it turned out, the width of the convector showed the greatest influence on the minimum warm air velocity. The v_w value can be reduced by approximately 19%, 28%, and 36% by increasing its size from the smallest (96 mm) to 146 mm, 186 mm, and 236 mm. Moreover, the convector with the smallest width can protect a room with a double-glazed window against cold air down-draughts only at a supply temperature of 45°C, and the lowest supply temperature, i.e. 35°C, can be used in rooms with four-pane windows. As the results of the analysis showed, the safe solution is to use convectors with a width of 236 mm. In this case, the room will be protected against the cold down-draughts formed on large glazing surfaces at the lowest supply temperature and even low thermal resistance.

Future works

The results of experimental tests showed an irregular temperature distribution in the lower part of the heating coil. Therefore, future work on this topic will focus on developing some modifications in the design of convector heaters to increase their efficiency.

Funding

This work was supported by the Bialystok University of Technology and financed by the Ministry of Science and Higher Education of the Republic of Poland [grant number WZ/WBIIS/ 8/2023].

Acknowledgment

Experimental tests were carried out in a climatic chamber made as part of the project "Innovative research and didactic centre for alternative energy sources, energy efficient construction and environmental protection" called in short INNO–EKO–TECH. This project was implemented by the Bialystok University of Technology (BUT) and was co-financed by the European Regional Development Fund.

References

[1] Kelly F.J., Fussell J.C. Improving indoor air quality, health and performance within environments where people live, travel, learn and work, Atmosferic Environment 200, 2019, 90-109.

[2] ASHRAE STANDARD 55. Thermal Environmental Conditions for Human Occupancy. Atlanta, 2013, USA.

[3] ISO 7730, Ergonomics of the thermal environment - Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria, 2005, Switzerland.

[4] 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, 2019, Brussels.

[5] Tian W., Han X., Zuo W., Sohn M.D. Building energy simulation coupled with CFD for indoor environment: A critical review and recent applications, Energy and Buildings, 164, 2018, 184-199

[6] Hajdukiewicz M., Geron M., Keane M.M. Calibrated CFD simulationto evaluate thermal comfort in a highly-glazed naturally ventilated room, Building and Environment, 70, 2013, 73-89.

[7] D'Agostino D., Parker D., Epifani I., Crawley D., Lawrie L. How will future climate impact the design and performance of nearly zero energy buildings (NZEBs)? Energy 240, 2022, 122479.

[8] Dorte Skaarup Østergaard, Kevin Michael Smith, Michele Tunzi, Svend Svendsen. Low-temperature operation of heating systems to enable 4th generation district heating: A review. Energy, Volume 248, 2022, 123529.

[9] Arefeh Hesaraki, Eleftherios Bourdakis, Adnan Ploskic, Sture Holmberg. Experimental study of energy performance in low-temperature hydronic heating systems. Energy and Buildings 109, 2015, 108–114.

[10] Zongjiang Liu, Wei Xu, Zhong Li, Linhua Zhang, Ji Li, Aisong Li, Airong Feng. Research on heating performance of heating radiator at low temperature. Journal of Building Engineering, Volume 36, 2021: 102016.

[11] Mikk Maivel, Jarek Kurnitski. Low temperature radiator heating distribution and emission efficiency in residential buildings. Energy and Buildings 69, 2014: 224-236

[12] Ala Hasan, Jarek Kurnitski, Kai Jokiranta. A combined low temperature water heating system consisting of radiators and floor heating. Energy and Buildings 41(5), 2009: 470-479.

[13] Adnan Ploskić, Qian Wang, Sasan Sadrizadeh. A holistic performance evaluation of ventilation radiators – An assessment according to EN 442-2 using numerical simulations. Journal of Building Engineering 25, 2019, 100818.

[14] Dorte Skaarup Østergaard, Svend Svendsen. Case study of low-temperature heating in an existing single-family house—A test of methods for simulation of heating system temperatures. Energy and Buildings 126, 2016: 535-544.

[15] Amin Hossein Jahanbin, Enzo Zanchini. Effects of position and temperature-gradient direction on the performance of a thin plane radiator. Applied Thermal Engineering 105, 2016: 467-473.

[16] Dorte Skaarup Østergaard, Svend Svendsen. Space heating with ultra-low-temperature district heating - a case study of four single-family houses from the 1980s. Energy Procedia 116, 2017: 226–235.

[17] Mostafa Omran Shobi, Hesamoddin Salarian, Ali Lohrasbi Nichkoohi, Majid Eshagh Nimvari. Experimental and numerical investigations of a modified designed baseboard radiator using an air gap enhancing free convection heat transfer. Journal of Building Engineering 32, 2020, 101535.

[18] Adnan Ploskić, Sture Holmberg. Low-temperature baseboard heaters with integrated air supply – An analytical and numerical investigation. Building and Environment 46(1), 2011: 176-186.

[19] Arefeh Hesaraki, Sture Holmberg. Energy performance of low temperature heating systems in five new-built Swedish dwellings: A case study using simulations and on-site measurements. Building and Environment 64, 2013: 85-93.

[20] K. Kerrigan, H. Jouhara, G.E. O'Donnell, A.J. Robinson. A naturally aspirated convector for domestic heating application with low water temperature sources. Energy and Buildings 67, 2013: 187-194.

[21] N. Bagheri, A. Moosavi, M.B. Shafii. Thermal enhancement of baseboard heaters using novel fin-tube arrays: Experiment and simulation. International Journal of Thermal Sciences 151, 2020, 106285.

[22] Lin B., Wang Z., Sun H., Zhu Y., Ouyang Q. Evaluation and comparison of thermal comfort of convective and radiant heating terminals in office buildings. Building and Environment, 106, 2016, 91-102.

[23] Yuxin Wu, Aleksi Mäki, Juha Jokisalo, Risto Kosonen, Simo Kilpeläinen, Sonja Salo, Hong Liu, Baizhan Li. Demand response of district heating using model predictive control to

prevent the draught risk of cold window in an office building, Journal of Building Engineering 33, 2021, 101855.
[24] Zukowski, M. A new formula for determining a minimum recommended value of inlet air velocity from UFAD system to prevent occupants from draught risk. Building and Environment 42(1) 2007: 171-179.
[25] Rueegg, T.; Dorer, V.; Steinemann, U. Must cold air down draughts be compensated when using highly insulating windows? Energy and Buildings 33(5) 2001: 489-493.
[26] Hua Ge, Paul Fazio. Experimental investigation of cold draft induced by two different types of glazing panels in metal curtain walls. Building and Environment 39(2) 2004: 115-125.
[27] Qing Wu, Zhaojun Wang, Jiankai Dong, Jing Liu. A method for judging the overheating of the radiator in the compensation of window downdraught based on thermal image velocimetry. Building and Environment 197, 2021, 107858.
[28] Embaye, M., AL-Dadah, R.K., Mahmoud, S. Numerical evaluation of indoor thermal comfort and energy saving by operating the heating panel radiator at different flow strategies. Energy and Buildings 121, 2016: 298–308.
[29] Shia-hui P., Peterson F. Convection from a cold window with simulated floor heating by means of a transiently heated flat unit. Energy and Buildings 23, 1995:95–103.
[30] Eckert, ERG; Jackson, TW. Analysis of turbulent free convection boundary layer on flat plate. NACA TR 1951;1015.
[31] Cheesewright R. Turbulent free convection from a vertical plane surface. Journal of Heat Transfer, Transactions of ASME 90,1968:1–8.
[32] Jezierski W., Zukowski M. Evaluation of the Impact of Window Parameters on Energy Demand and CO_2 Emission Reduction for a Single-Family House. Energies 16, 2023, 4429. https://doi.org/10.3390/en16114429

Appendix A

```
#include <iostream>
#include <cstdlib>
#include <fstream>
#include <stdio.h>
#include <conio.h>
#include <stdlib.h>
#include <math.h>

using namespace std;

ofstream outfile;
ifstream infile;

int main()

{
   float z0, zend, h, Gr, ni, average_vc, U, Tg, rc, rw, v_min;
   float vd, z, dc, vc, Te, Ti, convector_w, R_window, Rsi, Rse, Hw, Beta, d_star, v_star;
   float Hw_M[5], Te_M[5], Tg_M[5], vc_M[5], vmin_M[5], dc_M[5];
   int n_rect, n, m, i, iH, iTe;

   /*  DATA READING      */

   infile.open("DATA_1.txt");

   infile>>Te;
   infile>>Ti;
   infile>>Hw;
   infile>>convector_w;
   infile>>R_window;

   infile.close();

   outfile.open("RESULTS_1.txt");

   Hw_M[1]=1.5;
   Hw_M[2]=2.;
   Hw_M[3]=3.;
   Hw_M[4]=4.;
   Te_M[1]=-20.;
   Te_M[2]=-10.;
   Te_M[3]=0.;
   Te_M[4]=5.;

   ni=0.0001506;
   n=2;
   m=6;
   Rsi=0.13;
   Rse=0.04;
   U=1./(Rse+R_window+Rsi);

   for (int iH=1; iH<=4; iH++)
   {
   Hw=Hw_M[iH];

   Tg=Ti-Rsi*U*(Ti-Te);
   rc = -0.004*(Ti+Tg)/2 + 1.2831;
   rw = -0.0039*30 + 1.2855;
```

```
    Beta=1./((Ti+Tg)/2.+273.15);
    v_star=sqrt(9.81*Beta*(Ti-Tg)*Hw);
    Gr=9.81*Beta*(Ti-Tg)*Hw*Hw*Hw/ni/ni;

    vd=5.425*pow(Gr,-0.076)*v_star;
    d_star=Hw*pow(Gr,-0.1);

    dc=3.138*pow(Gr,-0.076)*d_star;
    dc_M[iH]=dc;

    // Integration range
    z0 = 0;
    zend = dc;

    // Number of intervals
    n_rect = 500;

    h = (zend-z0)/n_rect;
    cout << "step: h=" << h << endl;

    average_vc = 0.;

    for (int i=1; i<=n_rect; i++)
    {

      z=z0+ i*h;
      average_vc += vd*pow(z/dc,1./n)*pow(1-z/dc,m)*h;

      vc= vd*pow(z/dc,1./n)*pow(1-z/dc,m);

    }

    average_vc = average_vc /dc;
    v_min=sqrt(dc*rc/convector_w/rw)*average_vc;

    cout << "Result of integration by the rectangle method: " <<  average_vc << endl;

    vc_M[iH]=average_vc;
    vmin_M[iH]=v_min;
    Tg_M[iH]=Tg;

  }

infile>>Te;
    infile>>Ti;
    infile>>Hw;
    infile>>convector_w;
    infile>>R_window;

    outfile<<" Te= "<<Te<<"  R_window= "<<R_window<<"  convector_w= "<<convector_w<<" Tg  "<<Tg_M[1]<<endl;
    outfile<<" Hw     "<<Hw_M[1]<<"   "<<Hw_M[2]<<"   "<<Hw_M[3]<<"   "<<Hw_M[4]<<endl;
    outfile<<" vc  "<<endl<<vc_M[1]<<endl<<vc_M[2]<<endl<<vc_M[3]<<endl<<vc_M[4]<<endl;
    outfile<<" dc  "<<endl<<dc_M[1]<<endl<<dc_M[2]<<endl<<dc_M[3]<<endl<<dc_M[4]<<endl;
    outfile<<" vmin  "<<endl<<vmin_M[1]<<endl<<vmin_M[2]<<endl<<vmin_M[3]<<endl<<vmin_M[4]<<endl;

    outfile.close();

    system("PAUSE");
    return 0;
}
```