2_C21

Fan-assisted Trench Heating in Extreme Outdoor Temperatures

Olga Yakimchuk

ABSTRACT

Comfortable heating of rooms with large areas of external glazing is one of the most challenging issues in cold climate zones. The rule of thumb, in this case, is locating the heating unit under the window. Modern architectural trends lead to an increase in the number of facilities with panoramic glazing. The common practice is to locate trench units in close proximity to glass for such facilities. This analysis shows that even fan-assisted trench heating cannot always cope with cold air screening of large windows. A huge volume of cold air from the surface of the glass "outweighs" the warm rising flows, displacing the air from the convector towards the internal space areas. This can cause unwanted surface temperatures and uncomfortable conditions for people due to increased air velocity, i.e., draughts. Also, too low temperatures can lead to condensation and ice formation on glazing, especially in rooms with high humidity, such as swimming pools. In the present work, CFD analysis was used to evaluate equipment performance with increasing glazing height. Research results showed examples of trench heating implementation, leading to adverse consequences. The recommendation of using additional air supply along the glazing was tested. The simulation results showed that slot diffusers, accelerating the flow from trench heating, play a crucial role in the heating of floor-to-ceiling glazing and prevent draughts at floor level. In this work, the features of CFD modeling of trench units were considered.

INTRODUCTION

The appearance of buildings is always of great importance. Due to architectural trends, developers wish to use panoramic glazing more and more. Fortunately, modern materials and progress in the construction field allow implementing the most incredible projects. However, with these features, a new challenge shows up for engineers in terms of comfort in high-glazed facilities. This issue is especially topical for sharply continental climate zones, where temperatures in winter can drop to -25 ˚C or lower, and in summer exceed 35 ˚C. In winter, high glazing creates a large area cooled by thermal conductivity, which chills the air near the surface. Gravity causes cold air to move down along the glass, and this can produce uncomfortable conditions in the lower part. The common practice is to locate the trench units in close proximity to glass in order to minimize the engineering equipment in usable space and to get the ability to place the heating device along the entire surface of the glass. The modern trench unit can have a fan in their design, which helps to increase the heating power and the airspeed at the outlet of the convector. Fan-assisted models are often used for high glazing. As it turned out in practice, the use of such devices does not always guarantee compliance with the criteria of comfort. In case of very high glazing, a massive volume of cold air from window surface "outweigh" the warm rising flows, displacing the air from the convector towards the internal space areas. This can cause unwanted temperatures, surface condensation in facilities with high humidity, uncomfortable radiant temperature, and draughts. Manufacturers publish detailed equipment selection tables depending on the supply-return temperature, size, and fan speed. The scientific literature does not contain much information about the trench units. There is a study of its heat exchanger and efficiency (Frana et al. 2012), as well as the experimental studies of rising convective flows with numerical simulation (Pukhkal 2016, 2017). In the last work, the distance from the natural convection trench unit to glazing was varied, and it was concluded that to form an upward flow of warm air, it is necessary to install the device no more than 0.4 m from the glass. However, recommendations were not found on the limit of use floor convectors depending on the height of the glazing.

Olga Yakimchuk is a computer simulation engineer at APEX project bureau, Moscow, Russian Federation.

There is variability in the location of the heating element inside the trench unit with natural convection and, as a result, a different way of heating the room to the required temperature. Either the heating element is closer to the cold window surface, then the warm air stream should rise along the glass, or the heating element is located on the room side of the unit, so the cooled air stream passes through the convector and heats up, which reduces the human discomfort. This issue was analyzed by Pukhkal 2016. But for high glazing, the rate of falling air can be too high to comply with the comfort criteria. It is better to prefer forced convection for such solutions; therefore, in the present work, only the fan assisted case of the location of the heating element closer to glass was analyzed.

For the systems under consideration, field experiments are limited by the available types of buildings, applied engineering equipment, and weather conditions at the measurement moment. Numerical modeling allows applying a multivariate approach to this issue and studying the dependence of heating efficiency on parameters of interest. In this work, the glazing height was considered as such a parameter. First, the features of computer modeling of trench units, its geometry, and boundary conditions are reviewed. Next, the validation of CFD and selected option for modeling parameters are presented. After that, a study of the trench unit efficiency dependence on the glazing height was carried out. In the end, additional air supply along the glazing is offered as an option to maintain the desired temperature near the glass for rooms with high humidity.

CFD FEATURES FOR TRENCH UNITS

Numerical simulation has been used for a long time in various fields of physics, including indoor airflows and assessment of human comfort. Computational fluid dynamics (CFD) modeling quantitatively predicts thermal and fluid physical phenomena in an indoor space. It is included as an auxiliary tool in the construction standards of many countries. The international community has not yet developed precise criteria for the extends of CFD's capability, however, there are recommendations on the use and verification of this method (Chapter 13 of ASHRAE 2013, Chapter 6.4 of CIBSE 2015). It can increase the likelihood of getting a reasonable result with appropriate computing time. One of the main steps while developing a CFD model is the consideration of important components. Despite the increase in computer performance, there is still a need to neglect unnecessary details that do not significantly contribute to the result.

A CFD model is a combination of a geometric model of a room, a physical model of the processes under consideration, and a mathematical model, which can be solved by developed computer code. The physical model, in turn, includes the properties of air, the parameters of the heating unit operation, the conditions of heat exchange with the environment, radiation, and other phenomena. The mathematical model is set up, taking into account the peculiarity of important for solution physical phenomena and geometry of the space. It is a system of Navier-Stokes equations describing the change of parameters in space and in time in case of the time-dependent problem. Its solution allows us to obtain the distribution of the desired parameters and analyze them.

For a more accurate solution to this problem, it is necessary to take into account the flow turbulence. This complex three-dimensional nonstationary phenomenon is the fluctuation of environmental parameters in space and time. Additional terms and equations are added to the system of Navier-Stokes equations to consider for turbulence. At present, there is a large number of turbulence models. Some of them are studied in detail and has proven itself well, such as the standard "kε" configuration. Some of them are being investigated, often give a better resemblance, but require much more computational power, such as LES. The specifics of the problem determines the final choice of the turbulence model. For this study, two of them have been tested.

The finite volume method is used in most software systems to simulate the fluid flow. An essential stage of modeling with this method is the creation of a finite element mesh, which is a set of elementary volumes. These volumes fill up built geometry, and the equations are solved for each of them. The accuracy of the model, the calculation results, and computational time depend on the number of elements, the shape, and the regularity of the grid. Inflate boundary layers should be used to make the first step of the grid near the wall small enough for accurate accounting of heat exchange and resolution of the flow near the surfaces of windows and heating devices. The project determines room geometry with the presence of different sources of airflow and energy exchange. Since it is currently impossible to set all the geometric features in CFD analysis of the real room with equipment, it is necessary to simplify engineering systems to acceptable options. The question is how to represent the geometry and physics of a trench unit correctly.Figure 1 shows a schematic illustration of possible geometry assignments. There are some works in which it is considered as a heating element in the cavity (Pukhkal and Bulgakov 2017), Figure 1a.

This option is suitable for natural convection. The advantage of this approach is the absence of necessity to set the flow ratebecause the air rise automatically. On the other hand, this method requires a detailed drawing of the heating surface. Also, for correct air distribution, it is necessary to take into account grill's free area. Another option for representing a trench unit is a narrow surface at floor level with an area equal to the effective area of the convector outlet to ensure the correct balance of airspeed and flow rate. For example, Frana et al. 2013 went this way (Figure 1b).

This method neglects the convector's air intake. On the one hand, the explicit resolve of the air outflowing from the calculated volume should not significantly affect the result because of the relatively low air velocity at the convector's intake. On the other hand, using a separate surface for convector air intake may be useful for a more accurate presentation of the problem. In the present work, the trench unit is set as two surfaces on the floor. Air is supplied from a narrow and closer to the window one with a given temperature and velocity vector directed to the glazing surface. Air is exhausted from the room through a wide one (Figure 1c). The last two options are in a worse position relative to option a) in terms of accurate velocity accounting. It is necessary to take the flow rate and velocity spatial distribution according to data from the experiment or the convector's manufacturer, which is not always possible. A cold wall can be set as a wall with a fixed temperature or as a surface with a heat transfer coefficient and outdoor temperature. The second option is better to represent window and heat exchange with the environment.

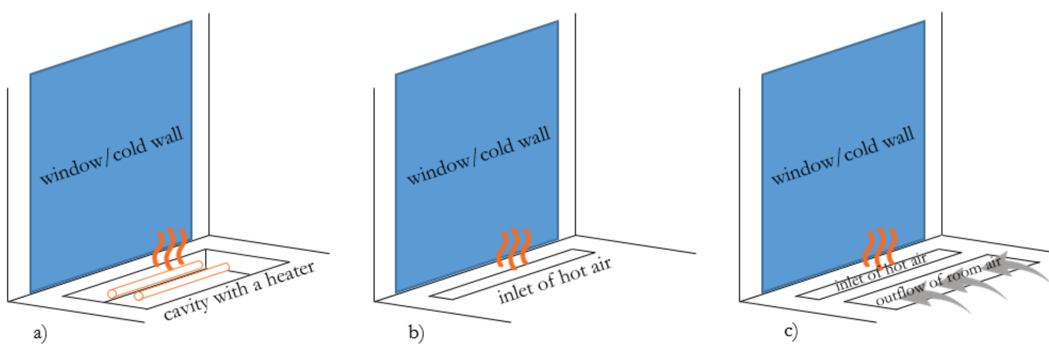


Figure 1
Schematic illustration of possible geometry to represent a trench unit: (a) as a cavity with a heater, (b) as an inlet of hot air, (c) as an inlet and outlet.

This work is done using SimScale. It is a cloud-based, online engineering simulation platform. This product offers various simulation capabilities for fluid dynamics, most of which are based on the use of open-source software OpenFOAM. The most suitable for indoor air exchange and comfort criteria assessment is “Convective heat transfer” (BuoyantPisoFoam for transient and BuoyantSimpleFoam for steady). This analysis type is used when temperature changes in the fluid lead to density variations and movement of the fluid due to gravity. It is better to prefer compressible fluid flow analysis for studies with trench heating; however, Boussinesq approximation suits in case of small temperature differences (less than 10 ˚C) that significantly speed up the calculation. This tool complements the standard OpenFOAM solvers with the ability to "enable" passive species transport if necessary. It contains built-in meshers; one of them based on the OpenFOAM SnappyHexMesh hex-dominant meshing algorithm was used in present work.

CFD VALIDATION

Due to a broad set of parameters that must be taken into account in numerical simulation and the influence of selecting options on the result, validation is required. "Validation demonstrates the ability of both the user and the code to accurately predict representative indoor environmental applications for which some sort of reliable data are available." (ASHRAE 2013) Chapter 13 of this document contains recommendations on how to avoid errors. The basic idea of validation is to identify suitable experimental data, to make sure that all important phenomena in the problem are correctly modeled, and to quantify the error and uncertainty in the CFD simulation. Comparison with the experiment is one of the important steps. Here, the paper of Muller et al. 2013 was chosen for comparison, where the measurements of the influence of the wall temperature on the flow from a convector were realized using the Particle Image Velocimetry (PIV) method. It turned out to be almost the only published experimental data on floor convectors with quantitative flow characteristics.

Figure 2 shows the computational domain in accordance with the experimental design with dimensions (Figure 2a) and the generated mesh (Figure 2b) with higher density around the objects of interest. All unnecessary elements are excluded from the calculation: there are only walls, a cold surface, a trench unit as an inlet and outlet surfaces, and an open boundary (door) to compensate pressure changes and avoid instability. The "Natural Convection Inlet-Outlet" on a door surface defines an open surface to an external environment considered infinite. The most suitable for our case option for comparison is the lowest of two wall surface temperature in the experiment, namely, 12.5 ˚C. There is no detailed description of the trench unit parameters in the article. Leaving air temperature was taken 50 ˚C, velocity 0.5 m/s tilted slightly towards the glass, flowrate at outflow 0.06 m^3/s. Convector's width and length was taken 0.36 m and 2 m correspondingly, with inlet surface width of 0.06 m, and outlet surface width of 0.24 m.

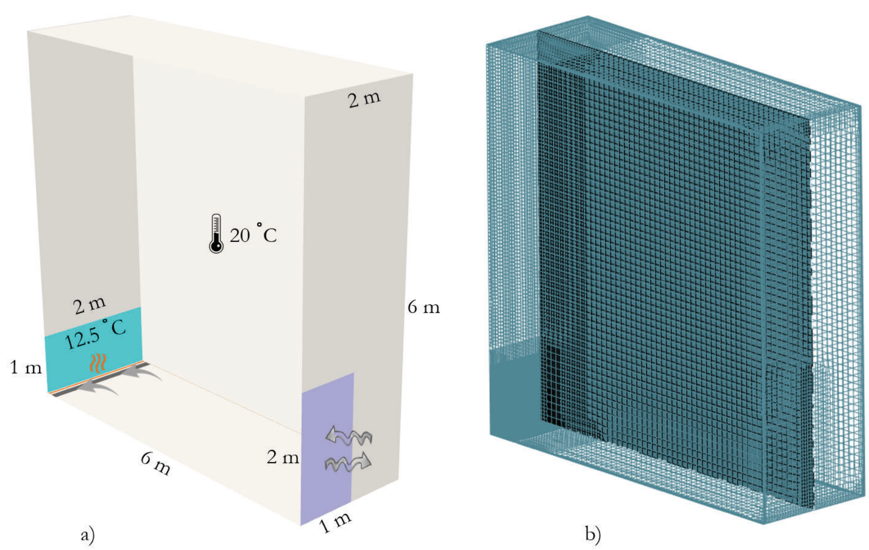


Figure 2 (a) Computational domain in accordance with the experimental design and (b) the mesh.

Variable parameters were the turbulence model (k-e or k-ω SST) and mesh step (0.014 m, 0.007 m, 0.0034 m). Result of comparison experimental velocity profile along the horizontal line at the position 0.34 m above the convector with the calculated one (Figures 3, 4) allow concluding, that k-ω SST for turbulence and 0.007 m as a mesh size around the heating unit a nd near the window surface suit for conducting analysis, described in the next section.

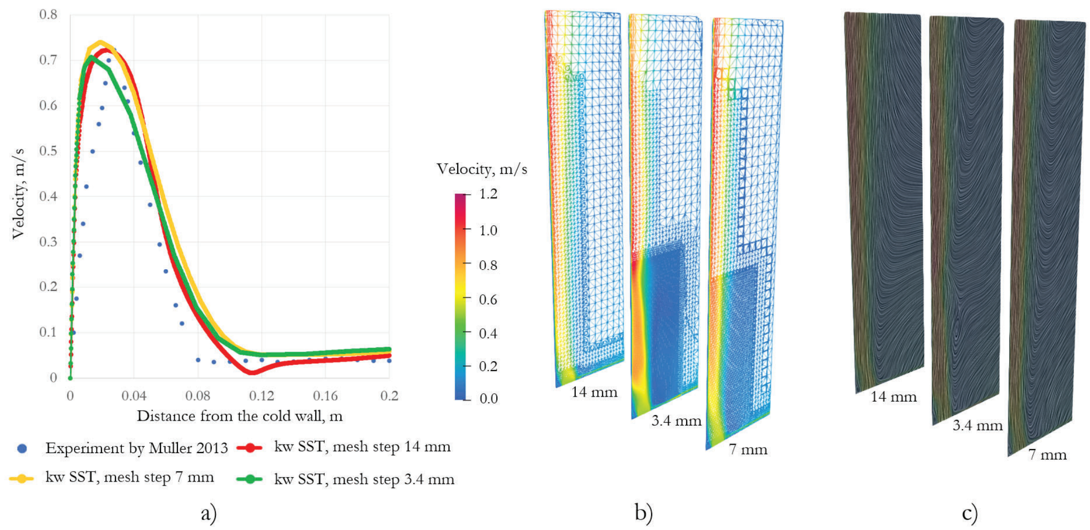


Figure 3 Comparison of results for k-ω SST turbulence model and different mesh steps (a) by velocity profile along the horizontal line at the position 0.34 m above the convector, (b) by velocity magnitude on mesh edges in vertical cross-section and (c) by the line integral convolution (LIC) vector field visualization technique.

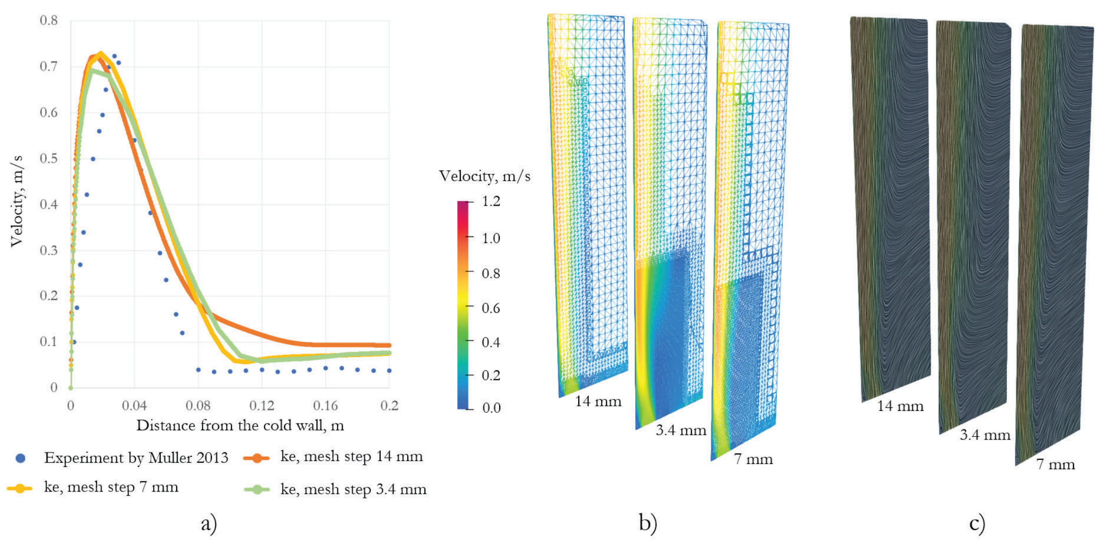


Figure 4 Comparison of results for k-e turbulence model and different mesh steps (a) by velocity profile along the horizontal line at the position 0.34 m above the convector, (b) by velocity magnitude on mesh edges edges in vertical cross-section and (c) by the line integral convolution (LIC) vector field visualization technique.

THE GRADUAL INCREASE IN GLAZING HEIGHT

Simulation parameters were selected according to the conclusions made above. Numerical experiments were carried out with a stepwise increase in the area of the glazing. The height of the geometry from a previous section increased from 6 to 12 meters. The surface of the cold wall consists of 2 m sections with heat exchange to the environment. The computational domain, its dimensions, boundary conditions, and the mesh are presented in Figure 5.

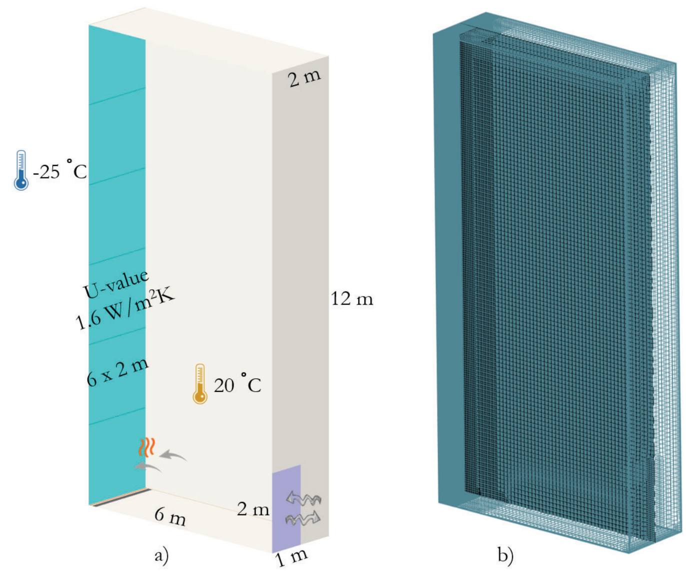


Figure 5 (a) Computational domain for the increase in glazing height study and (b) the mesh.

Heat gain must compensate heat loss to maintain a constant temperature in the room. With an increase in the height of the glazing, the area of the cold surface grows, so the heating device's power should increase. In this case, cooling through the glazing is balanced by the trench unit heat output. Two strategies exist to increase the power of the device: increase the leaving air temperature or the fan speed, in other words, air velocity at the unit's outlet. Two series of experiments were carried out to evaluate the influence of these factors. In one set, the outlet temperature was fixed at 50 ˚C, and the outlet velocity was a changing parameter. It increased with increasing of glazing surface from 0.07 m/s for 2 m height to 0.4 m/s for 12 m height. A constant speed of 0.7 m/s was maintained in another set, where the air temperature grew from 23 ˚C to 37 ˚C.

The analysis showed that the warm air from the convector does not rise to the glazing's full height, but only to its part, for all cases considered. The top of the glass remains cold. Figure 6a shows the dependence of the heated part fraction on the glass height. Blue points display the results for a fixed convector temperature (50 ˚C), yellow triangles - for an air speed of 0.7 m/s. Thus, with the increasing height of the glazing the moment comes when the floor convector, compensating only window heat loss, cannot cope with the glass's heating. The jet of warm air deviates into the room, creating uncomfortable conditions (Figure 6d). For reviewed configuration and fixed output temperature, such a moment has come for a height of 12m. The strategy of fixing output velocity and the increasing temperature was unsuccessful. It turned out that fast but not very warm air rises low or can't overcome the cold falling air.

Figure 6b shows the temperature distribution in the cross-section of the computational domain and on the wall surface, and Figure 6c shows the vertical velocity component and the direction of the vectors for 10 m glazing height.

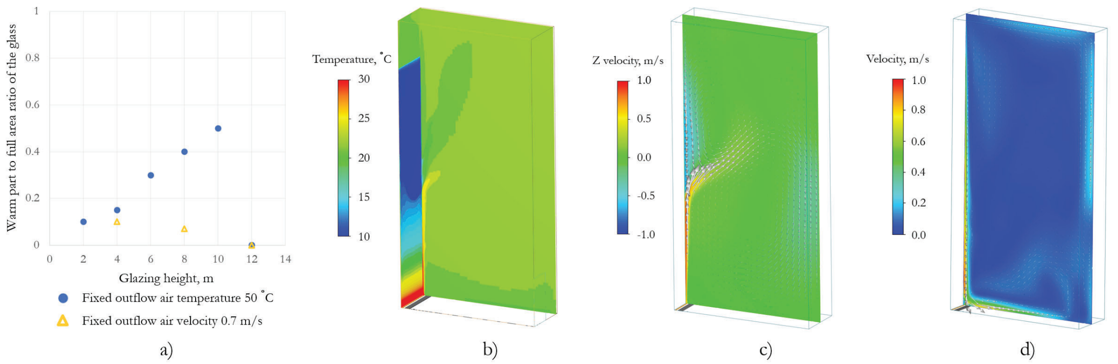


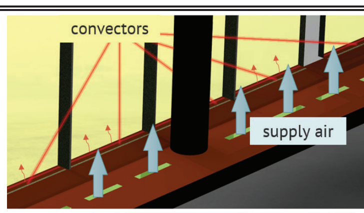
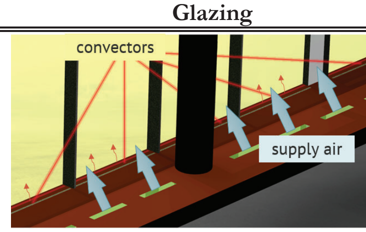
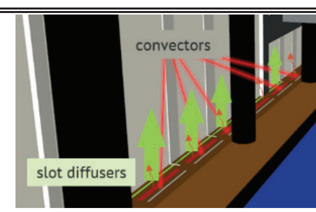
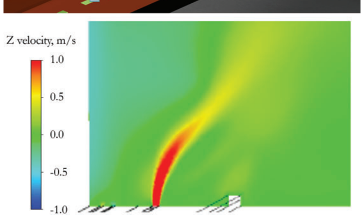
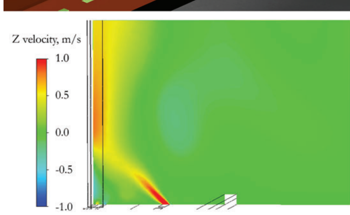
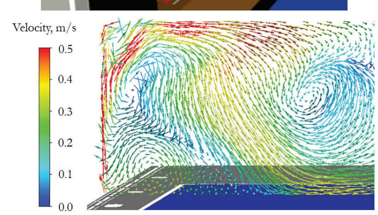
Figure 6 Results of glazing height study. (a) Chart of the ratio of warm glass part to full height in dependence on glazing height. (b) Temperature distribution in section plane and on the wall surface for 10 m height. (c) Vertical velocity component and veloc ity vectors in the middle cross-section for 10 m glass height. (d) Velocity value and vectors in the middle cross-section for 12 m glass height.

This is a typical appearance when the cold part of glazing remains, but no condensation occurs most often in practice due to low humidity in winter. The next section is an example of rooms with high humidity, where protecting the glass from condensation is a high-priority engineering task.

ADDITIONAL AIR SUPPLY ALONG THE GLAZING

In our practice (APEX project bureau), there were cases where the initial proposed solution with fan-assisted trench heating did not provide full heating of the glazing. In situations where cold glazing can lead to undesirable consequences, for example, moisture condensation, it is necessary to ensure its heating throughout the entire height of the room. For facilities such as swimming pools or water parks, where the humidity inside is maintained at a high level all year round, you can find recommendations to use an additional airflow along the glazing. The following is an example of considering various options for inflowing air into a swimming pool. It was not possible to provide the required conditions with the convector alone. It is necessary to use an additional airflow directed towards the glass surface to warm the glazing, as shown in columns 2 and 3 in Table 1.

Table 1. Options for Inflowing Air into a Pool

Supply Grill Angle 0°	Supply Grill Angle 45° Towards the Glazing	Slot Diffuser Along Glazing
		
		

Numerical model parameters were steady-state CFD, 5.6 million mesh cells, 40.3 m convector total length, 4 000 m^3/h convector supply air volume, 39 ˚C convector supply air temperature, 28 ˚C ventilation air temperature, -28 ˚C outdoor air temperature, 29 ˚C room air temperature. For cases in columns 1 and 2 ventilation airflow was 10 800 m^3/h. Convectors warm air rises along the glazing only when the ventilation supply air grilles are directed to the window. Otherwise, the cold air stream from the glass pushes the warm air, which results in the condensation on the glazing surface.

Another option was fan-assisted trench heating together with slot diffusers along the window surface before convectors. For case in column 3 of Table 1 ventilation airflow was 19 000 m^3/h. The use of slot diffusers, accelerating the flow from the convector, plays a crucial role in the heating of the full-height glazing of tall facilities with humid air.

CONCLUSION

In presented work, CFD modeling of heating facilities with glazing over the entire area of the facade by trench unit at extremely low temperatures was carried out. It was concluded that high panoramic glazing could not be heated well just by only floor convectors. Numerical experiments were carried out with a stepwise increase in the glazing height to support this assumption. This study has shown that the floor convector, compensating only window heat loss, cannot cope with the glass's heating starting at a certain height. When it happens, the trench unit's warm air deviates into the room, creating uncomfortable conditions. For the case considered, such a moment comes for a height of 12 m at fixed output temperature 50 ˚C. Fixing output velocity with the increasing temperature turned up an unsuccessful strategy.

These results cannot be applied directly to other parameters of glazing, convector, space geometry, and outdoor temperature. It is necessary to make new calculations when changing the configuration. An additional acceleration of the warm air due to the supply air increase the likelihood of obtaining the desired temperature on the glass, especially in rooms with high humidity.

Further studies will include the case of cooling the glazing zone with a trench unit in summer. At high outside temperatures, a significant amount of heat enters through the window, which can be compensated by the existing unit in a cooling regime. However, the cold airflow from the convector, displacing towards the room, can create a draft, which requires a separate study.

ACKNOWLEDGMENTS

The author would like to thank APEX project bureau for the support of the study.

REFERENCES

ASHRAE. 2013. *ASHRAE Handbook—Fundamentals*. Atlanta: ASHRAE.

CIBSE. 2015. *Building performance modelling.* London: The Chartered Institution of Building Services Engineers.

Fraňa, K., Müller, M., Lemfeld, F. 2012. An enhance of the energy effectiveness of the convectors used for heating or cooling. *International Journal of Mechanical and Mechatronics Engineering* 6(7):1247-1251.

Fraňa, K., Zhang, J.S., Müller, M. 2013. A numerical simulation of the indoor air flow. *International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering* 7(6):938-943.

Müller, M., Fraňa, K., Kotek, M., Dancova, P. 2013. The influence of the wall temperature on the flow from the floor convector (experimental results). *EPJ Web of Conferences* 45(01130):1-4.

Pukhkal, V. 2016 Studies of application conditions of in-floor convectors with natural air circulation in water heating systems. *Architecture and Engineering* 1(2):42-47.

Pukhkal, V., Bulgakov, V. 2017. Generation of natural convective air flows in rooms with the use of in-floor convectors with natural circulation. *Architecture and Engineering* 2(4):42-47.