Ing. Petr ZELENSKÝ[1),2)]
Ing. Martin BARTÁK, Ph.D.[1),2)]
prof. Dr. Ir. Jan L.M. HENSEN[1),3)]

[1)]CTU in Prague, Faculty of Mechanical Engineering
[2)]CTU in Prague, University Centre for Energy Efficient Buildings
[3)]Eindhoven University of Technology, Building Physics and Services

Simulation of Indoor Environment in the Concert Hall Housed in a Former Church

Simulace vnitřního prostředí v koncertní hale umístěné v objektu bývalého kostela

A previously developed approach to simplify the numerical modelling of heat sources based on the replacement of a heat source by a simple boundary condition (see VVI 5/2012) is applied in the real scenario of a recently refurbished former church built in the 14th century. The building is now used as a concert and conference hall with up to 350 visitors staying for different time periods during each day. The investor of the restoration was concerned about temperature fluctuations caused by the variable occupancy and the possible negative impact on the historical stucco decorations and on the original wooden trusses of the former church. Moreover, only natural ventilation through window openings on the street level and the windows in the roof is possible in order to preserve the original look of the building. The study elaborated upon in the paper is based on the results of the CFD simulations with simplified models of visitors acting as heat sources under two different occupancy scenarios.

Keywords: *CFD (Computational Fluid Dynamics), indoor environment, heat sources, simplified model, convectional flow, thermal plumes interaction*

Dříve vyvinutá metoda zjednodušení numerických modelů zdrojů tepla založená na náhradě zdroje tepla jednoduchou okrajovou podmínkou (viz VVI 5/2012) je aplikována na reálnou případovou studii nedávno zrekonstruovaného bývalého kostela ze 14. století. Budova je nyní vyžíván jako koncertní a konferenční prostor s obsazeností až 350 návštěvníků, kteří se zdržují proměnnou dobu během každého dne. Investor restauračních prací projevil obavu z teplotního kolísání způsobeného proměnnou obsazeností kostela, které by mohlo mít negativní vliv na historické štukové dekorace a na originální dřevěný krov bývalého kostela. Navíc je z důvodu zachování původního vzhledu budovy možné využít pouze přirozenou ventilaci okenními otvory na úrovni ulice a střešními okny. Studie vypracovaná v tomto příspěvku vychází z výsledků CFD simulací se zjednodušenými modely návštěvníků, kteří působí jako tepelné zdroje v rámci dvou různých scénářů návštěvnosti.

Klíčová slova: *CFD (Computational Fluid Dynamics), vnitřní prostředí, zdroje tepla, zjednodušený model, konvekční proudění, interakce konvekčních proudů*

INTRODUCTION

The former church of St. Anna is a 14th century gothic building located in the Old Town of Prague (hereinafter referred to as the church). It was desecrated at the end of the 18th century, recently completely reconstructed and it now serves as a community centre and universal space suitable for events such as concerts, conferences, etc., with the total capacity up to 350 visitors. The front part of the church interior after reconstruction is displayed in Figure 1.

Figure 1 Interior of the church, after reconstruction

Adaptation of historical buildings to a new function always brings a question about the influence of the indoor environment change on the building structures. This question arose also during the restoration works in 2001. The biggest concerns were indoor airflow velocities and air temperature distribution in the vicinity of the internal wall surfaces [1]. The design team had to take measures to protect original stucco decorations on the walls and the original wooden roof trusses, but at the same time only natural ventilation through window openings on the street level and through openings in the roof was possible, so that the original look of the building was preserved. Building energy simulation (BES) and computational fluid dynamics (CFD) were used to tackle this uneasy task [2].

CFD simulations can predict the influence of heat sources on natural ventilation and test different scenarios. However, for their effective use, it is especially vital to reduce the time to be spent on the model set-up as well as on the simulations. One of the possible ways to achieve this is to simplify computational models providing that the results accuracy will not be significantly lowered.

The high number of visitors acting as heat sources may have a crucial effect on the environment inside the former church. The main impact on the airflow is caused by convective currents generated by human bodies. Warm air is driven upwards by buoyancy forces and forms a rising thermal plume above each visitor.

Detailed CFD modelling and simulations of natural convective flows generated in ventilated and air-conditioned spaces are quite demanding for computing power and time. The original CFD simulation used by the design team in 2001 did not contain explicitly modelled indoor heat sources. The natural ventilation of the indoor space was emulated by the prescribed pressure difference between the internal space of the church and the surrounding environment, which was pre-calculated using BES [1]. The floor of the model was heated in order to compensate the thermal gain from the occupants. Yet, the explicit modelling of heat sources is important, as they can be vital for the appropriate air change [3], [4] and they can significantly influence air flow distribution indoors as well as the indoor environment quality [5]–[7].

The current work elaborates on the CFD simulations of the indoor air flow inside the church with two different occupancy scenarios: 65 and 304 visitors. The models of heat sources were simplified according to the previously developed method based on the replacement of each heat source by a simple boundary condition of convective flow, which is determined in advance from a detailed simulation of a thermal plume above a comprehensive model of the heat source [8]. The influence of each heat source on the overall air-flow pattern is preserved, while the computational demands of the simulations are lowered, enabling, e.g., variant studies.

The target of this research was to investigate the influence of the visitors acting as heat sources on the stucco decorations and the wooden roof trusses of the church. The results of the simulations with 65 and 304 visitors are mutually compared and also with the simulation without the explicitly modelled visitors. The effect of the heat sources on the indoor air flow, temperature stratification and ventilation rates are studied. The paper also demonstrates the usability of the previously developed method to simplify indoor heat sources for real scenarios.

NUMERICAL MODEL OF THE CHURCH

The modelled building is a former single-nave church with the main enclosure of approx. 9,630 m^3 total volume and the basic external dimensions (width x length x height) of approx. 11.4 x 43.5 x 29.2 m. The rear half of the nave is divided by a gallery 6 m above the floor, see Figure 2.

The church has 3 large window openings on the street level and 11 small window openings in the roof (6 of them on the south-facing side and 5 of them on the north-facing side). As it is possible to use natural ventilation through these openings only, all of them were opened.

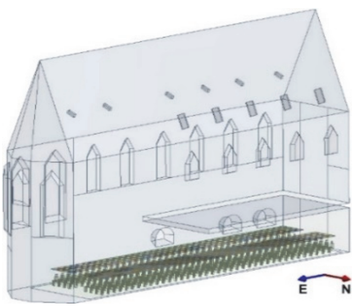

Figure 2 Model of the church, 304 seated visitors

Two numerical models of the church with a different number of seated visitors were created in the ANSYS Design Modeller. Both models included a 1.5 m wide region of external space surrounding the church, in order to correctly simulate the process of natural ventilation. The first model represents the almost fully occupied space, with 304 seated visitors, as displayed in Figure 2. The second model represents the low occupied space, with 65 seated visitors, distributed in two rows.

It was expected that the high number of visitors in the space of the church may have a significant effect on the indoor environment, for they act as heat sources driving natural ventilation. It was important to consider them in the CFD simulation. However, their models had to be simplified, in order to lower the computational demands.

METHOD TO SIMPLIFY MODELS OF HEAT SOURCES

The models of the visitors in the CFD simulations were simplified according to previously developed methodology [8], based on the replacement of each heat source by a simplified geometrical object and simple boundary conditions generating convective flow. All the visitors were considered as identical (with the same heat output). The preparation of the simplified model of the heat source was undertaken in two steps.

The model of a sitting thermal manikin with detailed geometry and boundary conditions of constant heat flux of 57.3 W/m^2 from its body was created as the first step. The total sensible heat output of the manikin was 90 W. Heat and momentum transfer near the manikin's surface were modelled using a dense boundary layer mesh (i.e., without wall functions), which is accurate but significantly increases the demands on the computer memory and prolongs the computational time. The CFD simulation was performed with the detailed model to obtain the velocity and temperature fields around the manikin, with the focus on the generated thermal plume.

The results of the detailed simulation were used to determine the temperature, velocity and turbulence profiles of the thermal plume at the height 0.7 m above the manikin, where the convective flow is considered as fully developed. The obtained profiles were parameterised using Curve Expert software and programmed in ANSYS Fluent by the User Defined Function (UDF). The velocity and turbulent quantities were prescribed as absolute, the temperature of the convective flow was prescribed as relative to the reference ambient temperature. The work flow of the programmed UDF was as follows:

1. set the local coordinate system with the origin in the geometrical centre of the subsidiary zone;
2. get the reference temperature in the vicinity of the subsidiary zone;
3. calculate the temperature profile of the convective flow, on the basis of the reference temperature;
4. prescribe the boundary condition for the temperature;
5. prescribe the boundary condition for the vertical velocity;
6. prescribe the boundary cond. for the turbulent quantities k and ε.

The heat sources were substituted by a geometrically simplified object with adiabatic surface, acting only as an obstacle to the airflow. The thermal plume rising above each heat source is induced artificially, by the velocity and temperature profiles calculated and prescribed in the UDF as boundary conditions at subsidiary zones created 2 m above the ground, see Figure 3.

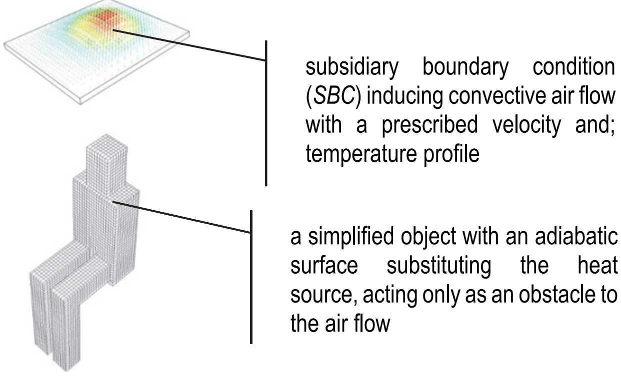

Figure 3 Simplified model of the sitting occupant

The previous development and testing of the method to simplify models of heat sources is described in the paper published in VVI 5/2012 [8]. Two additional studies were concluded prior to the simulation of the church. These studies targeted the optimal positioning of the subsidiary zone above the thermal manikin and the influence of the simplification on the merging of thermal plumes above multiple heat sources placed close to each other.

The CFD simulations were performed in the same manner as the simulations in the previously published studies [8]. The surface temperature of the room's walls was 19 °C, their emissivity was 0.94 and the emissivity of the thermal manikin surface was 0.98. The two equation k-ε turbulence model by Launder and Spalding [9] was used, with modification to account for the full buoyancy effects. This model has been found as the most suitable for CFD simulations of indoor spaces with the prevailing effect of natural convection [10]. Heat radiation was simulated using the surface-to-surface (*S2S*) model [11].

The *Body Force Weighted* scheme was chosen for the discretisation of the pressure equation as it is recommended for solving buoyancy driven flows [11]. The convective terms of the equation were solved using a second order upwind scheme. Pressure and velocity fields were coupled by the *SIMPLE* algorithm and the flow was considered as unsteady. 10 iterations for each time step of 0.1 s were computed in all the simulations. The residuals were in the order of magnitude of 10^{-5} or lower.

All the computational cases were simulated for a start-up period of at least 480 s after which the flow was considered fully formed and the results were then recorded for a further 120 s of the simulated time with a time step of 1 s. 120 data files were the outcome of each simulation. The values of temperature and velocity were recorded at appropriate points and averaged over the 120 s.

Optimal position of the subsidiary zone

The influence of the subsidiary zone position on the rising thermal plume was studied in the case of thermal manikin situated in the middle of a room. Four computational cases with different vertical positions of the subsidiary zone above the heat source were solved and compared to the reference case with a detailed model of the thermal manikin, see Figure 4.

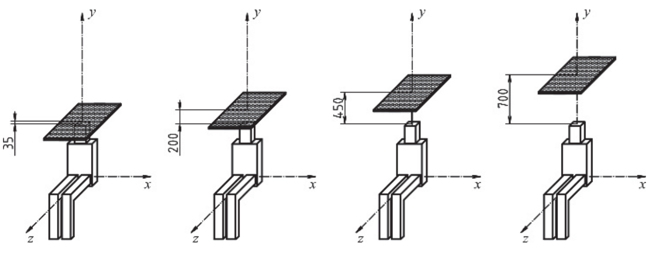

Figure 4 Vertical position of the subsidiary zone above the manikin's head (from left to right: 35 mm, 200 mm, 450 mm and 700 mm)

The velocity and temperature profiles determined for all the simulations with simplified models are close to the reference detailed case, especially in the higher regions of the room, i.e., above the subsidiary zones, see Figure 5 for example. The exception is the case with the subsidiary zone positioned 200 mm above the manikin's head. In this case, the comparison shows a slower velocity and lower temperature of the convection flow induced above the manikin. The most similar case corresponding to the reference case are the simulations with *SBCs* in the heights 450 and 700 mm above the manikin's head. The case with the *SBC* positioned 35 mm above the manikin reasonably corresponds to the reference case as well.

It can be advised to position the *SBC* rather higher above the heat source, where the thermal plume is fully developed, if the problem on hand allows it (i.e., in the simulation of a room with sufficient space above the heat sources, such as the church). If it is, for some reason, necessary to position the *SBC* in a closer vicinity to the heat sources (for example, rooms with a low ceiling), a CFD study should be performed, in order to test the correctness of the boundary condition defined from the reference detailed simulation.

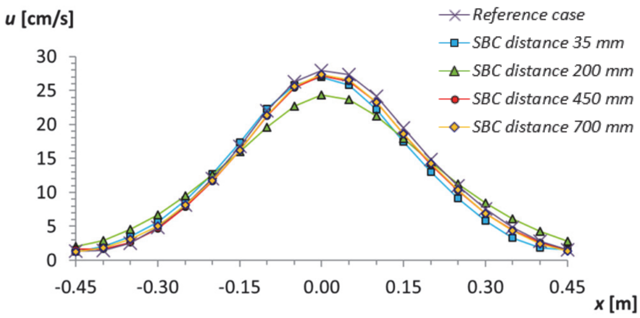

Figure 5 Velocity profiles at the height y = 1.975 m above the floor

Influence of the simplification on merging of thermal plumes

In the simulations with multiple heat sources, the merging of individual thermal plumes in higher regions must be considered. There was a concern how the developed method influences this physical process, especially as it is targeting simulations of spaces with a large number of heat sources possibly positioned close to each other, such as in the case of the church.

The numerical study was elaborated upon with the focus on the thermal plumes merging above multiple heat sources [12]. It compares a set of simulations with numerical models of four sitting thermal manikins in an enclosed room, which were modelled either in detail or simplified using the developed methodology, with the subsidiary zone placed 0.7 m above the thermal manikin. The group of manikins was placed in the middle of an enclosed room, with dimensions (width x length x height) 5 x 5 x 5.6 m, see Figure 6.

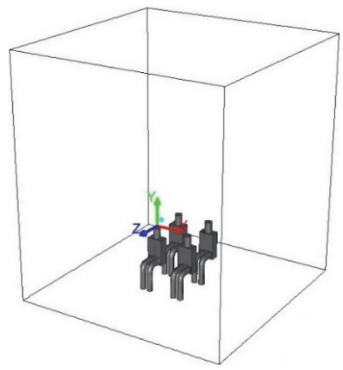

Figure 6 Thermal plumes merging simulation setup

The room dimensions and no obstacles around and above the manikins provided enough space for thermal plumes to develop sufficiently and merge. The height of the room was chosen according to the height of the space under the gallery, which divides the rear half of the church's nave, see Figure 2.

The results of the simulation with the detailed models were compared to the results with the simplified models of thermal manikins. The evaluation was focused on the phenomenon of the thermal plumes merging above the heat sources and the influence of the simplification of the heat source models on this physical process.

In both the simplified and detailed simulations, the thermal plumes are developing in a similar way, see Figure 7. Merging of the four thermal plumes is obvious in both cases, although it is more noticeable above the reference detailed models. Especially in the side view, it is possible to see that the thermal plume above the front seated manikin is, in this case, deflected more from the vertical direction and rises more towards the rear manikin than in the case with the simplified models. In the higher regions, the thermal plumes merge completely in both simulations, and adhere to the ceiling of the room and spread further to the vertical walls in a very similar way.

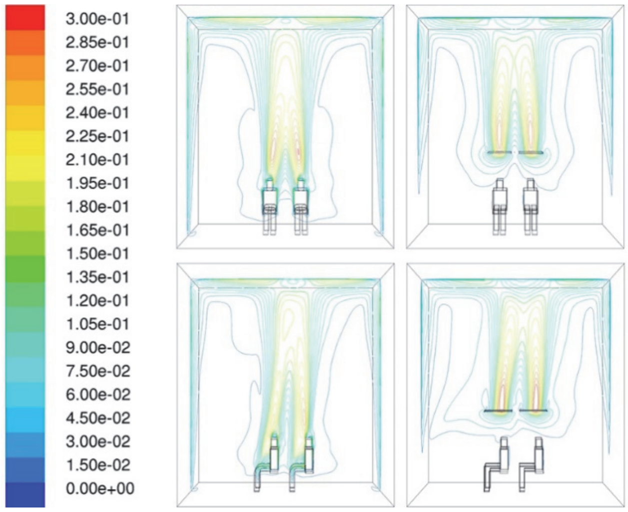

Figure 7 Velocity isolines [m/s], detailed model (left) and simplified model (right); top – front views; bottom – side views

It has been shown that the simplification affects the development of the thermal plumes considering their merging above the heat sources. However, the difference of the simplified simulation from the reference simulation decreases with the increasing height above the heat sources. The higher it is above the manikins, the more the thermal plumes resemble each other in the two simulated cases, see Figure 8 for example, presenting velocity profiles nearby the room's ceiling. Although the plumes in the simulation with the simplified models are not fully merged yet, they are very close to the plumes from the simulation with the reference models. The velocities, in both cases, are comparable and it can be expected that full merging in the simplified case is achieved if the space is higher. This deficiency could be solved by placing the prescribed boundary condition lower above the thermal manikins' heads, so the merging would be achieved sooner in the case with the simplified models also. However, in the case of the church, this was not necessary, as the primary interest was to simulate airflow high above the floor.

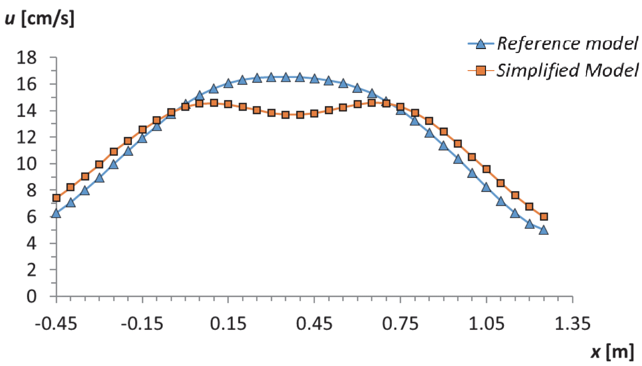

Figure 8 Velocity profiles 3.6 m above the floor (front view)

CFD SIMULATION OF THE CHURCH

The building interior (approx. 9,630 m^3), including the external space surrounding the building, was divided by an unstructured tetrahedral grid into more than 17.9 × 10^6 control volumes in the model with 65 visitors and more than 28.6 × 10^6 control volumes in the model with 304 visitors. The minimum size of a cell was in both cases 25 mm, the maximum was 250 mm.

The boundary conditions of the surfaces facing the surrounding environment (see Figure 9) were specified as a free boundary with zero pressure gradient. The external air temperature was –7 °C, as a winter scenario was considered. The church is partially, from two sides, surrounded by other occupied buildings with the indoor temperature being considered as 20 °C. The temperature of the ground under the floor was 5 °C. The composition of the building constructions was prescribed in the model and the solver calculated the temperature of the internal surfaces. Thus, the thermal conduction through the walls was taken into account. All the walls of the church are made from marlstone, the concrete floor is covered by tiles, the windows are single glazed.

Figure 9 Boundary surfaces facing the surrounding environment (red line)

The CFD simulations were solved in the software ANSYS Fluent 16.0 as a non-isothermal flow of incompressible ideal gas (air). The flow in the proximity of the walls was solved using wall functions and the dimensionless wall distance *y*+ for the first cell near the walls of the church was in the range from 80 to 110. The two equation k-ε turbulence model by Launder and Spalding [9] – the so called *standard* – was used considering the influence of the temperature and buoyancy on the turbulence. This model has been previously found to be the most suitable for CFD simulations of indoor spaces with the prevailing effect of natural convection [10].

The *Body Force Weighted* scheme was chosen for the discretisation of the pressure term as it is recommended for buoyancy driven flows [11]. The convective terms were solved using a second order upwind scheme. A coupled and steady-state solver was used to obtain the pressure and velocity fields. Radiation was not simulated, as the visitors are sitting very close to each other and the balance of heat transfer among them should be close to 0. Radiation heat transfer should be more significant for the visitors sitting at the end of the rows only, which was neglected.

The evaluation of the results targeted at the influence of heat source on the indoor environment inside the church, especially on the airflow patterns, temperature stratification and air change rates. The results of simulations with 65 and 304 visitors were mutually compared and also against the simulation of the space without the modelled visitors [2].

RESULT ANALYSIS AND DISCUSSION

A CFD simulation of the indoor air flow with a prevailing effect of natural convection caused by low temperature differences is always challenging and the calculation tends to be slow and unstable. In our case, the convergence of the calculation was achieved after more than 4,000 iterations in the case of the simulation with 65 visitors and more than 5,000 iterations in the case of 304 visitors. All the residuals reached the order of 10^{-4} or less, excluding the residuals of continuity, which reached the order of 10^{-3}. This could have been caused by the instability of the convective flow. The convergence has been proven on the basis of the total mass flux balance in the whole computational domain, which approx. reached $4 \cdot 10^{-2}$ kg/s in both simulated cases. This is reasonable, considering that it is approximately 1 % of the absolute value of the mass flux in the computational case. The imbalance occurred on the boundary facing the surrounding environment. The balance of the total mass flux through the building's interior was 0 kg/s for both simulated cases, i.e., perfectly balanced.

Images of the simulated velocity and temperature fields in the church were evaluated after the convergence of each simulation. The velocity vectors, the isolines of velocity magnitude and the contours of temperature were studied in 1 vertical plane *y-z* (side view) intersecting the geometrical centre of the building and 5 vertical planes *x-y* (front view) intersecting the church with a spacing of 7 m.

Velocity fields

The velocity vectors and isolines of the velocity magnitude in the vertical plane *y-z* intersecting the centre of the church (side view) are displayed. See Figure 10, for the simulation with no explicitly modelled heat sources performed during the restoration works in 2001 [2] and Figure 11 and Figure 12 for the new simulations with 65 visitors and with 304 visitors, respectively.

Comparison of the velocity fields in the three presented simulations shows that the explicit modelling of the visitors acting as heat sources significantly affects the results of the simulation. In the case without the explicit models of visitors (Figure 10), there are two large circulation flows above the raised gallery in the rear part of the church and one large circulation flow in the front of the building. However, in both cases with the models of visitors (Figure 11 and Figure 12), the vortexes above the gallery merge in one large circulation flow. This may have been caused by a strong thermal plume, which is formed above the models of the visitors and rises from the space under the gallery. Also, it is possible to see that, in the case with 304 visitors, there is stronger air circulation in the space of the church than in the case with 65 visitors, although the airflow trajectories resemble each other.

The visitors have a strong influence also on the environment in their close vicinity. There is very low air circulation under the gallery in the simulation without the explicitly modelled visitors. The presence of the visitors in the space causes the airflow with strong mixing in the other two cases.

The large windows in the front of the building (on the left side of the displayed intersections) have another strong influence on the airflow in the church. The air is cooled down in their vicinity and falls down towards the floor. The falling flow does not fully reach the auditorium in the two simulations with the visitors, as it is turned upward by the convective flow rising from the space under the gallery, unlike in the case without the explicitly modelled heat sources. The air flow velocities did not exceed 0.65 m/s in any of the three simulated cases. The maximum velocities in both simulations with the explicit models of the visitors are in the space under the gallery, where the ventilation air enters the building through the three window openings on the street level. This may be the main concern, as the fast, cold flow directly enters the space with the sitting visitors, which could cause their discomfort. The visitors acting as heat sources consequently influence the distribution of this supplied air in the rest of the space. Nearby the walls, the airflow velocity does not exceed 0.45 m/s.

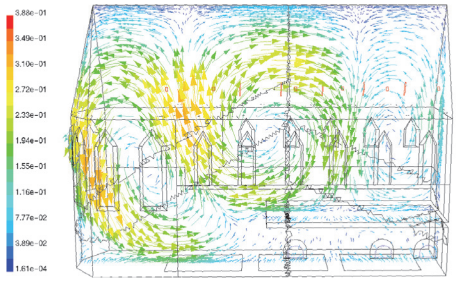

Figure 10 Velocity field in the case without heat sources, velocity vectors [m/s] [2]

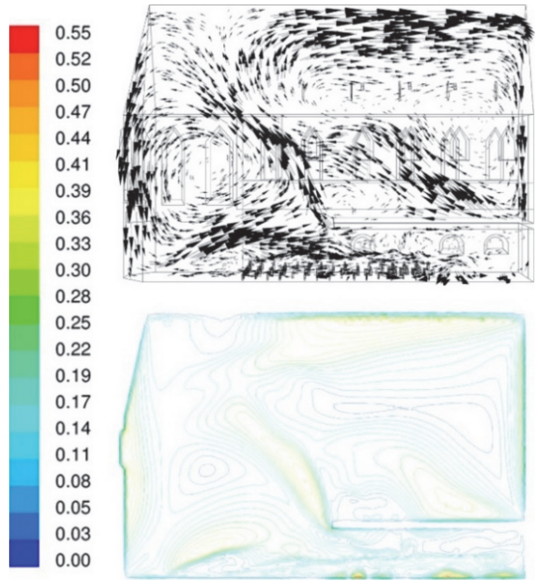

Figure 11 Velocity field in the case with 65 visitors, upper: velocity vectors, lower: velocity magnitude isolines [m/s]

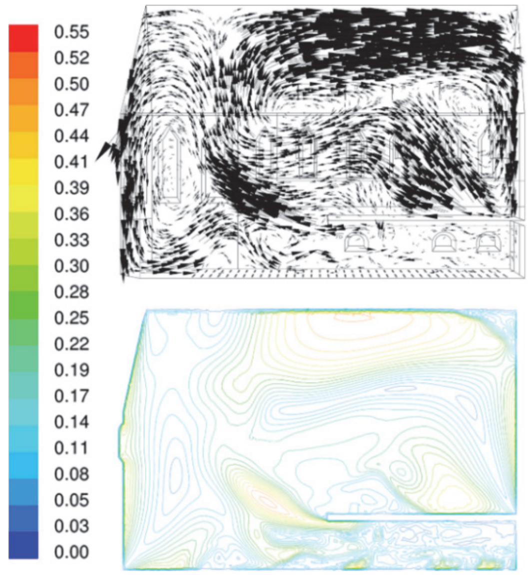

Figure 12 Velocity field in the case with 304 visitors, upper: velocity vectors, lower: velocity magnitude isolines [m/s]

The velocity vectors and isolines of the velocity magnitude were evaluated in 5 vertical planes x-y (front view) also in order to get more complete information about the velocity fields under the roof where the original wooden trusses are exposed to the air flow. See Figure 13 for example, showing a velocity field 21 m from the rear wall of the church. It has been confirmed that the maximum velocity near the sides of the roof does not exceed 0.45 m/s.

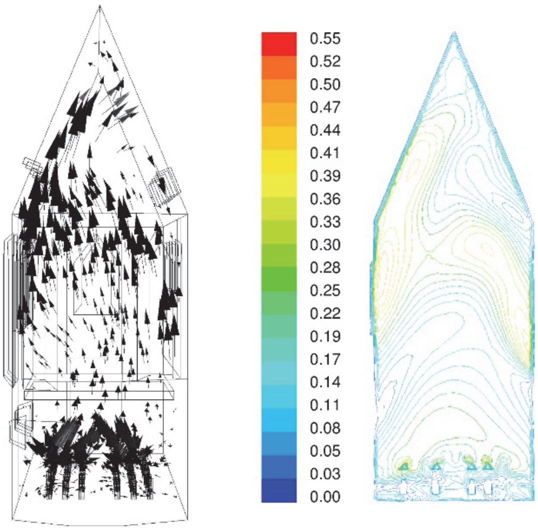

Figure 13 Velocity field in the case with 65 visitors - left: velocity vectors, right: velocity magnitude isolines [m/s]

Temperature fields

The temperature distribution in the church was evaluated on the basis of the temperature contours in 1 vertical plane *y-z* (side view) and 5 vertical planes *x-y* (front view). The selected cross-sections for the cases with 65 and 304 visitors are displayed in Figure 14.

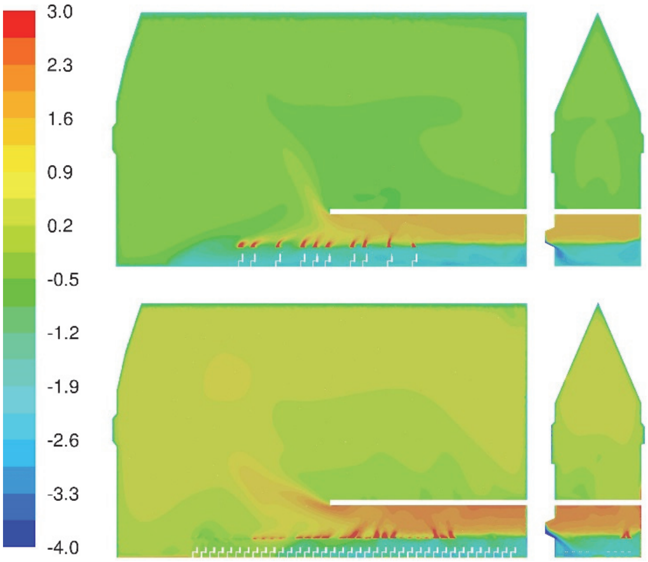

Figure 14 Temperature field, contours of the temperature [°C] 65 visitors (top), 304 visitors (bottom)

The influence of the visitors acting as heat sources is obvious. In the case with 65 visitors, the space below the gallery in the rear part of the church is colder, with more gradual stratification. In the case with more visitors, the supply air is heated up faster and the influence of the cold air flows from the low window openings is decreased.

While the lower regions of the church are strongly influenced by the visitors, the remaining space shows better thermal stability. The temperature fields in the higher regions are almost uniform in both simulations, without any significant stratification. The only disturbances are the warmer plumes rising from the space below the gallery, with the temperature higher by approx. 0.5 K than the surrounding air.

The average temperature in the church increased by 1.1 K due to the increase in the number of visitors from 65 to 304. This relatively small temperature change should not have a negative effect on the preserved internal constructions of the building. However, the average temperatures inside the church are, in both cases, just slightly above 0 °C, which is not acceptable considering the thermal comfort of visitors. A heating system should be used during winter.

It should be considered that the methodology used to simplify indoor heat sources does not perform well in the low regions above the ground. It can be expected that the temperature above the ground (up to a height of approx. 2 m) would be higher in reality, with a more gradual thermal stratification.

Ventilation flow rates

The ventilation flow rates have been evaluated for both simulated cases with the explicitly modelled visitors by summing the volume flow rates through the low window openings, see Table 1. The supplied flow rate per visitor was calculated.

Table 1 Ventilation rates

Simulated occupancy	Ventilation flow rate	Flow rate per visitor
65 visitors	1.07 m^3/s	16.5 L/s
304 visitors	1.35 m^3/s	4.4 L/s

The supplied volume of fresh air should be sufficient for this type of space. The minimum volume flow rate for the auditorium seating area is 2.7 L/s per person according to the ASHRAE standards [13]. The volume of fresh air per visitor in both simulated cases exceeds this minimal rate. However, according to the Czech standards, the minimal recommended volume flow rate for occupied spaces is 5 L/s [14]. Although this recommendation is indicated for forced ventilation systems only, it should be considered.

In situations with less visitors in the church, it can be advised to reduce the area of the window openings and reduce the airflow rates. In the simulated adjustment, the cold airflow supply enters the space of the seated visitors with the velocity reaching 0.65 m/s, which could cause discomfort. Partial closing of the window openings may reduce the risk of draught in the auditorium. In addition, the internal air temperature will increase.

CONCLUSION

Two CFD models of the former church with two different occupancy scenarios (65 and 304 visitors) were created in order to study the influence of the indoor heat sources (visitors) on the environment inside the building. The models of the heat sources were simplified following the previously developed methodology of substitution by simple boundary conditions generating thermal plumes. The CFD simulations were mutually compared and with the simulation without explicitly modelled heat sources also.

The presence of the visitors influences velocity patterns in the whole space and the temperature fields, especially in the regions close to the floor. The higher regions of the church show good thermal stability.

The simulation results show that it is possible to use the former church of St. Anna as a cultural space with large number of visitors, without negatively affecting the preserved building structures. The indoor air velocities in close vicinity to these structures do not exceed 0.45 m/s. The building's thermal environment shows good resistance to the occupancy fluctuation as well, especially in the higher regions, at the location of the preserved building structures and the original wooden roof trusses. The increase in the number of visitors from 65 to 304 caused a rise in the average temperature in the space of only 1.1 °C, with only a small change in the temperature fields in the higher regions.

It has been shown that natural ventilation of the building in the winter period is possible. However, it is recommended to use a heating system during winter, as the indoor air temperature is very low even in the case with 304 visitors, which would have a negative effect on the comfort of the visitors. Another concern may be the cold air supply from the low window openings that directly enters the space of auditorium, with the velocity reaching 0.65 m/s. This fast, cold air flow could be reduced by partial blocking of the window openings. However, in the situation with 304 visitors this should be accompanied by the installation of a heating system in the church, so the driving force of the natural ventilation will increase. Otherwise, the volume flow rate per visitor may not meet the values recommended by the Czech standards.

It has been shown that the previously developed methodology [8], [10] to simplify the models of the indoor heat sources is suitable for this type of study. The proposed method enabled the simplification of the heat sources' geometry and of the computational mesh around them also, but it preserved the rising thermal plumes patterns. Thus, the possibility of the variant CFD simulation study was enhanced.

REFERENCES

[1] BARTÁK, M., DRKAL, F., HENSEN, J.L.M., LAIN, M., MATUŠKA, T., SCHWARZER, J., ŠOUREK, B. Simulation to Support Sustainable HVAC Design, in *Proceeding of the 18th International Conference on Passive and Low Energy Architecture*, 2001, Floriánopolis, Brazil, pp. 7–9.

[2] BARTÁK, M., DRKAL, F., LAIN, M., SCHWARZER, J. *Obrazy proudění v kostele Sv. Anny v Praze 1* [in Czech]. Czech Technical University in Prague, Report no. 01002, 2001.

[3] XING, H., HATTON, A., AWBI, H.B. A study of the air quality in the breathing zone in a room with displacement ventilation. *Building and Environment*, 2001, 36(7), pp. 809–820.

[4] SKISTAD, H., MUNDT, E., NIELSEN, P.V., HAGSTROM, K., RAILIO, J. *Displacement ventilation in non-industrial premises*. REHVA Guidebook No. 1. Trondhaim: Tapir, 2002.

[5] AWBI, H. B. *Ventilation of Buildings*. 2nd ed. London: Spon Press, 2003.

[6] ZBOŘIL, V., MELIKOV, A., YORDANOVA, B., BOZHKOV, L., KOSONEN, R. Airflow Distribution in Rooms with Active Chilled Beams, in *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007*, Helsinki, Finland, pp. 1-7.

[7] ZUKOWSKA, D. *Airflow interactions in rooms - Convective plumes generated by occupants* [PhD thesis]. Technical University of Denmark, 2011.

[8] ZELENSKÝ P., BARTÁK M., HENSEN J.L.M. Model sedící osoby jako zdroje tepla ve vnitřním prostředí [in Czech]. *Vytápění, větrání, instalace,* 2012, vol. 5, pp. 22–26.

[9] LAUNDER, B.E., SPALDING, D.B. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Energy*, 1974, (3), pp. 269–289.

[10] ZELENSKÝ P., BARTÁK M., HENSEN J.L.M. Faktory ovlivňující CFD simulaci konvekčního proudu nad zdrojem tepla ve vnitřním prostředí [in Czech]. *Vytápění, větrání, Instalace*, vol. 5, pp. 2–8, 2013.

[11] ANSYS Inc., *ANSYS Fluent User's Guide*. USA: ANSYS Inc., 2013.

[12] ZELENSKÝ, P., BARTÁK, M., HENSEN, J.L.M., VAVŘIČKA, R. Influence of turbulence model on thermal plume in indoor air flow simulation, in *Proceedings of the 11th REHVA World Congress "Energy Efficient, Smart and Healthy Buildings" – Clima 2013*, Prague, Czech Republic.

[13] ASHRAE. 2003. *Ventilation for Acceptable Indoor Air Quality*. ANSI/ASHRAE Standard 62-2001.

[14] EN 13 779 *Ventilation for non-residential buildings - Performance requirements for ventilation and room-conditioning systems.* 2010.

NOMENCLATURE

c	specific thermal capacity [J/kg·K]
d	thickness [m]
k	turbulence kinetic energy [m^2/s^2]
$y+$	dimensionless wall distance [-]
ε	turbulent dissipation rate [m^2/s^3]
λ	thermal conductivity [W/m·K]
ρ	density [kg/m^3]
SBC	subsidiary boundary condition