

SIMULATION OF INDOOR ENVIRONMENT IN THE CONCERT HALL HOUSED IN A CONVERTED FORMER CHURCH

Petr Zelenský¹, Martin Barták¹, Jan L. M. Hensen^{1,2}

¹Czech Technical University in Prague, Department of Environmental Engineering,
Czech Republic

²Eindhoven University of Technology, Building Physics and Services, Netherlands

ABSTRACT

The paper presents an application of a new method to simplify models of heat sources in CFD simulations of large spaces exposed to variable occupancy patterns. The previously developed approach is applied in a real scenario of a recently refurbished former church built in 14th century, now used as a concert and conference hall with up to 350 visitors staying for different period during each day. The investor of the restoration was concerned about temperature fluctuations caused by variable occupancy and their 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 at the street level and roof windows is possible in order to preserve the original look of the building.

Study elaborated in the paper is based on the results of CFD simulations with simplified models of visitors acting as heat sources under two different occupancy scenarios.

INTRODUCTION

The former church of St. Anna is a 14th century gothic building located in the Old Town of Prague. It has been completely reconstructed (it was desecrated already in 19th century) and now it 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 St. Anna interior after reconstruction is displayed in Figure 1.



Figure 1 – Interior of St. Anna, after reconstruction

Adaptation of historical buildings to a different way of use 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 patterns and air temperature distribution in the vicinity of the internal wall surfaces (Barták, Drkal, Hensen et al., 2001). 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 at 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 (Barták, Drkal, Lain, & Schwarzer, 2001).

CFD simulation enables to predict a complex interaction of natural ventilation and test different scenarios. However, for its 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 of St. Anna. Their main impact on the airflow is caused by convective currents generated by them. Warm air is driven upwards by buoyancy forces and forms a rising thermal plume above each heat source.

Detailed CFD modeling 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 heat sources. The natural ventilation of the indoor space was emulated by prescribed pressure difference between internal space of the St. Anna and surrounding environment, which was pre-calculated using BES (Barták, Drkal, Hensen et al., 2001). Yet, the explicit modelling of heat sources is important, as they can be vital for appropriate air change (Skistad, Mundt, Nielsen, Hagstrom, & Railio, 2002; Xing, Hatton, & Awbi,

2001) and they can significantly influence air flow distribution indoors as well as indoor environment quality (Awbi, 2003; Zbořil, Melikov, Yordanova, Bozhkov & Kosonen, 2007; Zukowska, 2011)

The current work elaborates CFD simulations of indoor air flow inside the St. Anna with two different occupancy scenarios (65 and 304 visitors). The models of heat sources were simplified following the previously developed method based on the replacement of each heat source by simple boundary conditions of convective flow, which is determined in advance from a detailed simulation of thermal plume above a comprehensive model of the heat source (Zelenský, Barták, & Hensen, 2013). The influence of each heat source on the overall air-flow pattern is preserved, yet the computational demands of the simulations are lowered, enabling the variant study.

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

NUMERICAL MODEL OF THE FORMER CHURCH ST. ANNA

The St. Anna is a former single-nave church with the main enclosure of total volume approx. 9,630 m³ and basic external dimensions (width x length x height) approx. 11.4 x 43.5 x 29.2 m. The rear half of the nave is divided by second gallery in the height of 6 m, see Figure 2.

The St. Anna has 3 large window openings at the street level and 11 small window openings in the roof (6 of them at the south-facing side and 5 of them at the north-facing side). Only natural ventilation through these openings is possible.

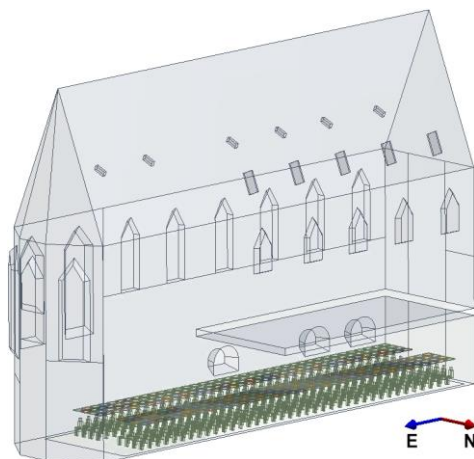


Figure 2 – Model of St. Anna, 304 seated visitors

Two real-scale numerical models of the St. Anna with different number of seated visitors were created in Ansys Design Modeller. Both models include 1.5 m thick layer of surrounding external space, in order to correctly simulate the process of natural ventilation. The first model represents almost fully occupied space, with 304 seated visitors, as displayed in Figure 2. The second model represents low occupied space, with 65 seated visitors, similarly distributed in two rows.

It was expected that a high number of visitors in the space of St. Anna, acting as heat sources, may have a significant effect on the indoor environment, as they are driving force for the natural ventilation. It was important to consider them in the CFD simulation, yet their models had to be simplified, in order to lower computational demands of the simulation. All the visitors were considered as identical (with the same heat output) and substituted by a repeatedly used simplified model, based on the previously developed methodology.

METHOD TO SIMPLIFY MODELS OF HEAT SOURCES

The models of visitors in the CFD simulations were simplified according to previously developed methodology (Zelenský, Barták, & Hensen, 2013), based on the replacement of each heat source by simplified geometrical object and simple boundary conditions of convective flow. The preparation of simplified model of heat source was done in two steps.

Model of a sitting thermal manikin with detailed geometry and boundary conditions of constant heat flux of 57.3 W/m² 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 modeled using dense boundary layer mesh (i.e. without wall functions), which is accurate but significantly increases demands on the computer memory and prolongs computational time. CFD simulation was performed with the detailed model to obtain velocity and temperature fields around the manikin, with focus on the generated thermal plume.

The results of detailed simulation were used to determine 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 – see (Zelenský, Barták, & Hensen, 2013) for more details. The obtained profiles were parameterized using Curve Expert software and programmed in CFD solver by User Defined Function (UDF).

The heat sources in the numerical model of the St. Anna were substituted by geometrically simplified object with adiabatic surface, acting only as an obstacle to the air flow. Thermal plume rising above each heat source is induced artificially, by velocity

and temperature profiles calculated and prescribed by the UDF as a boundary conditions at subsidiary zones created 2 m above the ground, see Figure 3.

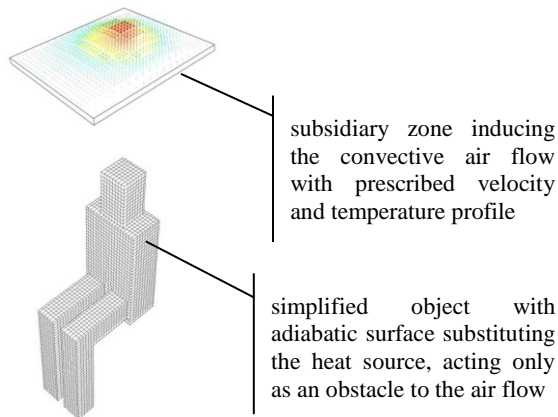


Figure 3 – Simplified model of sitting occupant

The velocity and turbulence parameters 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 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 reference temperature;
4. prescribe the boundary condition of temperature;
5. prescribe the boundary condition of vertical velocity;
6. prescribe the boundary conditions of turbulence parameters k and ε .

The previous development and testing of the method to simplify models of heat sources is described more in detail in Zelenský et al. (2013).

CFD SIMULATION

The building interior (approx. $9,630 \text{ m}^3$), including the external space surrounding the building, was divided by an unstructured 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. Boundary conditions of the internal surfaces of St. Anna were defined in the form of temperatures, on the basis of previous multi-zonal ESP-r simulation, performed during the first stage of the project preparation (Barták, Drkal, Lain, et al., 2001). Boundary conditions of the surfaces facing surrounding environment were specified as free boundary with zero pressure gradient. The external air temperature was $-7 \text{ }^\circ\text{C}$, as winter scenario was considered.

CFD simulations were solved using the software ANSYS Fluent 16.0 as non-isothermal flow of incompressible air with the influence of thermal expansion (so called Boussinesq approximation). The flow in the proximity of the walls was solved using wall functions, considering the influence of temperature and buoyancy on the turbulence. Two equation $k-\varepsilon$ turbulence model by Launder and Spaldin (Launder & Spalding, 1974) – so called *standard* – was used. This model has been previously found as the most suitable for CFD simulations of indoor spaces with prevailing effect of natural convection (Zelenský, Barták, Hensen, & Vavříčka, 2013).

The *Body Force Weighted* scheme was chosen for the discretization of pressure term as it is recommended for solving buoyancy driven flows (ANSYS Inc., 2013). The convective terms were solved using second order upwind scheme. A coupled and steady-state solver was used to obtain pressure and velocity fields.

The evaluation of the results targeted at the influence of heat source on the indoor environment inside the St. Anna, especially on the airflow patterns, temperature stratification and air change rates. The result of simulations with 65 and 304 visitors were compared mutually and also with the simulation of the space without modelled visitors (Barták, Drkal, Lain, et al., 2001).

RESULT ANALYSIS AND DISCUSSION

CFD simulation of indoor air flow with 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 5,000 iteration in the case of the simulation with 65 visitors and more than 4,000 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} . The convergence has been proved on the basis of the total mass flux balance in the whole computational domain, which reached the order of 10^{-3} kg/s in both simulated cases. This is reasonable, considering the absolute values of mass flux in the computational cases. Also the balance of the total mass flux through the building internal space was evaluated to prove the convergence. It was 0 kg/s for both simulated cases, i.e. perfectly balanced.

Images of simulated velocity and temperature fields in the St. Anna were evaluated after the convergence of each calculation. The velocity vectors, isolines of velocity magnitude and contours of temperature were recorded in 1 vertical plane $y-z$ (side view) intersecting the geometrical center of the building and 5 vertical planes $x-y$ (front view) intersecting the church with spacing of 7 m.

Velocity fields

The CFD simulation without explicitly modelled heat sources performed during the restoration works in 2001 (Barták, Drkal, Lain, et al., 2001) was considered as the reference case for the result analysis of the velocity fields. The velocity vectors and isolines of velocity magnitude in the vertical plane y - z intersecting the centre of St. Anna (side view) are displayed in the Figure 4 for the case with no explicitly modelled heat sources, in the Figure 5 for the case with 65 visitors and in the Figure 6 for the case with 304 visitors.

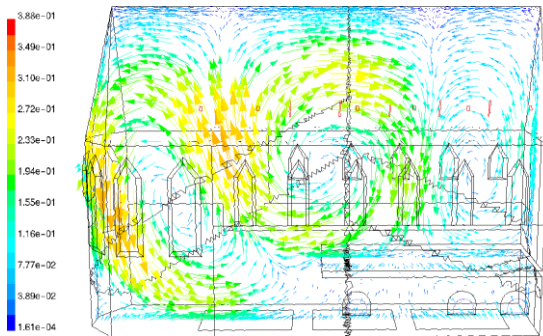


Figure 4 – Velocity field in the case without heat sources (velocity vectors [m/s])

Comparison of the velocity fields in all the three CFD cases shows that the large windows in the front of the St. Anna (on the left side of the displayed intersections in the plane y - z) strongly influence the indoor air flow patterns. The air is cooled down in their vicinity and falls down towards the floor.

The falling flow does not reach the floor in the two simulations with visitors, as it merges with the rising convective flow, unlike in the case without heat sources. In the case with 65 visitors it spreads atop the gallery in the rear part of the St. Anna and attached circumferential circulation is formed along the walls. In the case of 304 visitors the falling flow is turned up and two circulation flows are formed, one in the front part and one in the rear part of the former church. Maximum velocities nearby walls are 0.42 m/s, which is sufficient for preservation of the original stucco decorations.

The visitors have a strong influence on the environment in their close vicinity. There is very low air circulation under the raised gallery in the rear part of the St. Anna in the simulation without any visitors. The presence of the visitors in this space causes air flow with strong mixing in the other two simulations.

The air flow velocities did not exceed 0.6 m/s in any of the three simulated cases. Maximum velocities are in the space under the gallery, where the ventilation air enters the building through the three window openings. This may be the main concern, as the fast cold flow enters directly the space with sitting visitors, which could cause their discomfort. The visitors acting as heat sources consequently influence the distribution of this flow in the rest of the space.

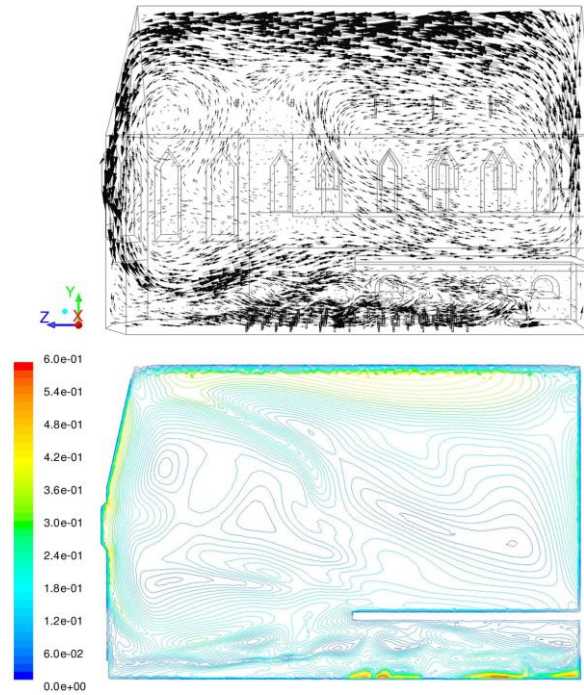


Figure 5 – Velocity field in the case with 65 visitors
- Upper: velocity vectors
- Lower: velocity magnitude isolines [m/s]

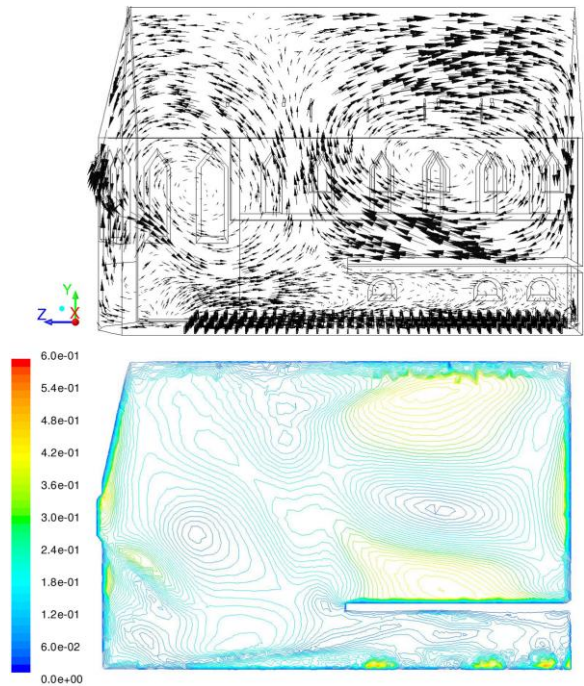


Figure 6 – Velocity field in the case with 304 visitors
- Upper: velocity vectors
- Lower: velocity magnitude isolines [m/s]

Velocity vectors and isolines of velocity magnitude were evaluated also in the 5 vertical planes x - y (front view), in order to get more complete information about velocity fields under the roof, where the original wooden trusses are exposed to the air flow. It

has been confirmed that the maximum velocity near the roof sides does not exceed 0.42 m/s, which is sufficient for the preservation of the wooden trusses. See for example Figure 7, showing velocity field 21 m from the rear wall of the former church. Vertical sections also show apparent thermal plumes rising above the sitting visitors.

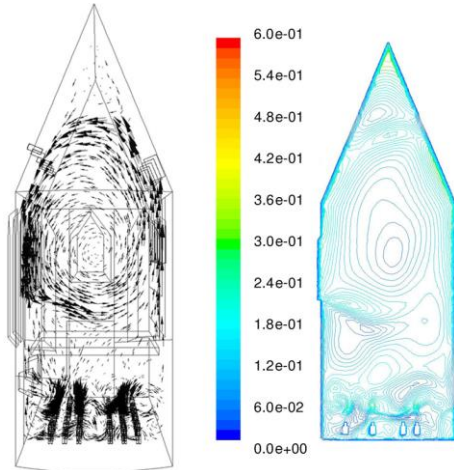


Figure 7 – Velocity field in the case with 65 visitors
 - Left: velocity vectors
 - Right: velocity magnitude isolines [m/s]

Temperature fields

The stratification and temperature distribution in the St. Anna were evaluated on the basis of the temperature contours recorded in 1 vertical plane y-z (side view) and 5 vertical planes x-y (front view). The selected intersections for the cases with 65 and 304 visitors are displayed in Figure 8 and Figure 9.



Figure 8 – Temperature field, case with 65 visitors
 (contours of temperature [K])



Figure 9 – Temperature field, case with 304 visitors
 (contours of temperature [K])

The influence of the visitors acting as heat sources is apparent in the lower levels above the floor. In the case with 65 visitors the space below the raised gallery in the rear part of the St. Anna 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 St. Anna are strongly influenced by visitors, the remaining space shows a better thermal stability. The temperatures fields in the higher regions are uniform in both cases with heat sources, without significant stratification. The only disturbance is the rising warmer plume in the simulation with 304 visitors, with temperature higher by 0.2 °C than the surrounding air.

The average temperature in the St. Anna increased by 0.6 °C due to the increase of the number of visitors from 65 to 304. This relatively small temperature change shall not have a negative effect on the preserved internal constructions of the building. However, the average temperatures inside the St. Anna are in both cases close to 3 °C, which is not acceptable considering thermal comfort of visitors. A heating system should be used during winter days.

It should be considered that the used methodology to simplify indoor heat sources does not perform well in the low levels above the ground. It can be expected that the temperature above the ground (up to the height approx. 2 m) would be in reality higher, with more gradual thermal stratification. The temperature fields in the higher regions (above approx. 2 m) shall be accurate (Zelenský, Barták, & Hensen, 2013).

Ventilation flow rates

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

Table 1 – Ventilation rates

Simulation	Ventilation flow rate	Flow rate per visitor
65 visitors	1,60 m ³ /s	24.6 L/s
304 visitors	1,76 m ³ /s	5.8 L/s

The supplied volume of fresh air is more than sufficient for this type of space in both simulated cases. The recommended value of the volume flow rates for auditorium seating area is 2.7 L/s per person (ASHRAE, 2003) and the volume of fresh air per visitor in both simulated cases exceeds the recommended rate. The natural ventilation shall be sufficient for the case of St. Anna.

It can even be advised to reduce the area of the lower windows openings and lower the air flow rates towards 2.7 L/s per visitor. In the current adjustment the cold supply air flow enters the space of sitting visitors with velocity reaching 0.6 m/s, which could cause discomfort. Partial closing of the low window openings would reduce the risk of draught in the auditorium. Also the internal temperature shall increase.

CONCLUSION

Two computational models of the former church of St. Anna 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 of thermal plume. The CFD simulation were compared mutually and also with the simulation without explicitly modelled heat sources. The presence of the visitors influence velocity patterns in the whole space and temperature fields especially in the levels close to the floor. The higher regions of the church show a good thermal stability.

The simulation results show that it is possible to use the St. Anna as a cultural space with high number of visitors, without negatively affecting the preserved building structures, especially stucco decorations on the walls and original wooden roof trusses. The indoor air velocities in the close vicinity of these structures do not exceed 0.42 m/s, which is favorable for the constructions. The building thermal environment shows good resistance to the occupancy fluctuation too, especially in the higher regions, at the location of the preserved building structures. The increase of the number of visitors from 65 to 304 caused rise of the average temperature in the space only 0.6 °C, with only small change in the temperature fields in the higher regions.

The main concern may be comfort of the visitors, as the supply air flow from the low window openings enters directly the space of auditorium, with velocity reaching 0.6 m/s. This fast cold flow may be reduced by partial blocking of the low window openings. The ventilation air flow rates exceed the recommended values, so this solution is possible. Also it is necessary to use a heating system during winter days, as the indoor temperature is very low even in the case with 304 visitors.

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

This research will continue with additional unsteady CFD simulation considering the thermal mass of the building constructions. It will target the process of the indoor environment stabilization after the visitors leave the church. The change in velocity fields and temperature decrease in time will be studied.

NOMENCLATURE

k	=	turbulence kinetic energy	[m ² /s ²]
ε	=	turbulent dissipation rate	[m ² /s ³]

ACKNOWLEDGEMENT

This project was supported by research grant of CTU Prague nr. SGS12/179/OHK2/3T/12.

REFERENCES

- ASHRAE. 2003. *Ventilation for Acceptable Indoor Air Quality*. ANSI/ASHRAE Standard 62-2001.
- Awbi, H. B. 2003. *Ventilation of Buildings*. 2nd ed. London: Spon Press.
- Barták, M., Drkal, F., Hensen, J. L. M., Lain, M., Matuška, T., Schwarzer, J., & Šourek, B. 2001. Simulation to Support Sustainable HVAC Design. *The 18th International Conference on Passive and Low Energy Architecture*, Florianópolis, Brazil, 7–9.
- Barták, M., Drkal, F., Lain, M., & Schwarzer, J. (2001). *Obrazy proudění v kostele Sv. Anny v Praze 1*. [report no. 01002]. Czech Technical University in Prague. [in Czech]
- ANSYS Inc. 2013. *ANSYS Fluent User's Guide*. USA: ANSYS Inc.
- Lauder, B. E., & Spalding, D. B. 1974. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Energy*, (3), 269–289.
- Skistad, H., Mundt, E., Nielsen, P. V., Hagstrom, K., & Railio, J. 2002. *Displacement ventilation in non-industrial premises*. REHVA Guidebook No. 1. Trondheim: Tapir.
- Xing, H., Hatton, A., & Awbi, H. B. 2001. A study of the air quality in the breathing zone in a room with displacement ventilation. *Building and Environment*, 36(7), 809–820.
- Zbořil, V., Melikov, A., Yordanova, B., Bozhkov, L., & Kosonen, R. 2007. Airflow Distribution in Rooms with Active Chilled Beams. *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007*, Helsinki, Finland, 1–7.
- Zelenský, P., Barták, M., & Hensen, J. L. M. 2013. Simplified Representation of Indoor Heat Sources in CFD Simulations. *Proceeding of the 13th International Conference of the International Building Performance Simulation Association*, Chambéry, France.
- Zelenský, P., Barták, M., Hensen, J. L. M. & Vavříčka, R. 2013. Influence of turbulence model on thermal plume in indoor air flow simulation. *Proceedings of the 11th REHVA World Congress „Energy Efficient, Smart and Healthy Buildings“ – Clima 2013*, Prague, Czech Republic.
- Zukowska, D. 2011. *Airflow interactions in rooms - Convective plumes generated by occupants* [PhD thesis]. Technical University of Denmark.