Proceedings of BS2013:
13th Conference of International Building Performance Simulation Association, Chambéry, France, August 26-28

SIMPLIFIED REPRESENTATION OF INDOOR HEAT SOURCES IN CFD SIMULATIONS

Petr Zelenský[1], Martin Barták[1], and Jan L. M. Hensen[1,2]
[1]Czech Technical University in Prague, Department of Environmental Engineering, Czech Republic
[2]Eindhoven University of Technology, Building Physics and Services, Netherlands

ABSTRACT

The paper deals with simplified modeling of heat sources in the indoor environment based on the replacement of heat source by simple boundary conditions. The aim of this approach is to capture the influence of a real heat plume on overall air flow pattern in the room using a model which is easy to set up and can quickly produce simulation results. The replacement boundary conditions can be determined from a detailed CFD simulation (or from a measurement) of convective flow above a comprehensive model of the heat source.

A sitting thermal manikin in enclosed room is used as an example to demonstrate the influence of different computational model aspects on the simulation results. This includes position of the replacement boundary conditions, type of turbulence model and air temperature in the room. Study elaborated in this paper is based on the results of CFD simulations both with detailed and simplified models, which are mutually compared.

INTRODUCTION

For an effective use of computational fluid dynamics (CFD) in HVAC design 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.

Heat sources are very common elements of the indoor environment. Their main impact on the surrounding airflow is caused by convective currents occurring around them. Warm air is driven upwards by buoyancy forces and forms a rising thermal plume above the heat source. CFD modeling and simulations of natural convective flows generated in ventilated and air-conditioned spaces are quite demanding for computing power and time. Yet, they have to be considered as their momentum may be comparable with momentum of air flows from ventilation diffusers and they can significantly influence air flow distribution indoors as well as indoor environment quality, possibly in a negative way (Awbi, 2003; Zbořil et al., 2007; Zukowska, 2011). Correct CFD simulations can also contribute to more effective removal of heat gains from the indoor heat sources which can lower energy consumption of ventilation and air-conditioning systems.

The paper evaluates the results of CFD simulations of thermal plumes above a simplified model of heat source (sitting thermal manikin) with different layouts of replacement boundary conditions and different ambient conditions. The results are mutually compared and discussed in order to evaluate influence of the parameters on the simulated thermal plume rising above the heat source.

METHOD DESCRIPTION

A new method to simplify models of heat sources has been designed and tested. To test this approach we used a model of sitting thermal manikin as an example of indoor heat source. In general, occupants are usually present in indoor environments and it is not possible to effectively reduce their heat output. Their relative contribution to the total heat gains in various buildings is rising (Zukowska, 2011). Moreover, the surface of human body has relatively complex geometry which must be usually simplified in CFD simulations. Our choice was also driven by the possibility to use the same manikin in laboratory experiments for further testing and improvements of our approach.

Model of a thermal manikin with detailed geometry and boundary conditions of constant heat flux from the manikin body was created as the first step. Heat and momentum transfer near the manikin surface was modeled using dense boundary layer mesh (i.e. without wall functions), which is accurate but it significantly increases computational demands on the computer memory and prolongs computational time. CFD simulations were 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 and velocity profiles in three heights of the thermal plume. These profiles were subsequently used as the replacement boundary conditions inducing an artificial thermal plume rising around a substitutive, geometrically simplified object with adiabatic surface.

A simplified model of the heat source can be repeatedly used in different simulations (e.g. with variation of boundary conditions), or as a multiple object in a single simulation (multiple occurrence of an identical heat source in the computational model). The proposed method is also aiming to lower computational time of the simulation compared to the case with a detailed model of the heat source. The influence of heat sources on the surrounding environment shall be still reflected. However, it is necessary to capture the thermal plume rising above the heat source in the detailed simulation as precisely as possible.

GEOMETRY OF THE MODELS

The computational models of thermal manikin resembling a sitting occupant were based on previous work of Koiš (2009) who assembled a metal thermal manikin according to the previous prototype of the Centre for Indoor Environment and Energy at DTU in Lyngby, Denmark. The geometry of the manikin differs from the shape of a human body. However, such simplified models of occupants are frequently used in indoor environmental studies. Previous experiments of Zukowska et al. (2010a) indicated that simplification of thermal manikin geometry such as used by Koiš (regarding the shape of human body) shall not significantly affect character of the thermal plume formed above the manikin. The results of CFD simulation with detailed model of thermal manikin were consequently used as the basis for design of more simplified computational models.

Four computational models of a seated manikin were designed: a basic (detailed) model of thermal manikin, a model with the heat source substituted by a single simplified boundary condition and two models with the heat source substituted by two simplified boundary conditions defined at two different locations.

Geometrical models for all computational tasks were created in the preprocessor Gambit 2.4, the numerical grids were generated in the preprocessor TGrid 13.0.

Model of thermal manikin

Geometry of the basic model is a detailed copy of the thermal manikin constructed by Koiš (2009) – see Figure 1. The model of the manikin forms an empty shell, its inner volume is not defined.

The manikin was placed in the middle of an enclosed room with the floor dimensions 4.5 m x 4.5 m and ceiling height of 3 m. The room dimensions and no obstacles around the manikin provided enough space, for the thermal plume to develop sufficiently. The same dimensions of an enclosure used Borges et al. (2007) in their experiments focused on thermal plume above a human body.

The computational domain was meshed with orthogonal grid, only the areas of more complex geometry (close to the manikin) were meshed by tetrahedral and prismatic cells. The model of thermal manikin was surrounded by cells with edge dimension of 12.5 mm, which were followed up by cells with edge dimension of 25 mm and in more distant areas by cells with edge dimension of 50 mm which filled the remaining volume of the modeled room. The computational mesh was refined in the close vicinity of the thermal manikin surface in order to simulate the heat transfer precisely. Dimensionless wall distance y^+ was less than 5 for the first five cell layers next to the manikin surface. The computational mesh was refined also in the close vicinity of the walls of the room.

The boundary conditions at the thermal manikin surface were considered as constant. The uniform heat flux from the surface was set to 57.3 W/m^2, which yielded the total sensible heat output of 90 W.

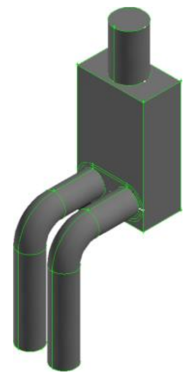

Figure 1 Model of thermal manikin

Substitution of the heat source by simplified boundary conditions

The rising thermal plume is in this case induced by velocity and temperature profiles set as boundary conditions at subsidiary zones created at chosen heights in the place of original model of thermal manikin. As the shape of the subsidiary zone resembles a box for pizza, it has been named *pizza-box* which is used in the following description.

Three computational cases with different configurations of subsidiary zones were solved. In the simplest case, only one zone was created at the height 700 mm above the head of the model, in the region where the thermal plume is considered to be fully developed (Zukowska, Melikov, & Popiolek, 2010b). In the other two cases, the velocity and temperature profiles were assigned to the surfaces of two *pizza-boxes,* see Figure 2. The upper one was created at the height of 700 mm above the model head, for the reasons mentioned above. The lower one was defined either at the height of calves of the thermal manikin or at the height of its waist in order to assist the upper *pizza-box* in creating the convective current around the manikin's body. The aim was to show the importance of the lower *pizza-box* position. Its optimum location, however, was not systematically studied.

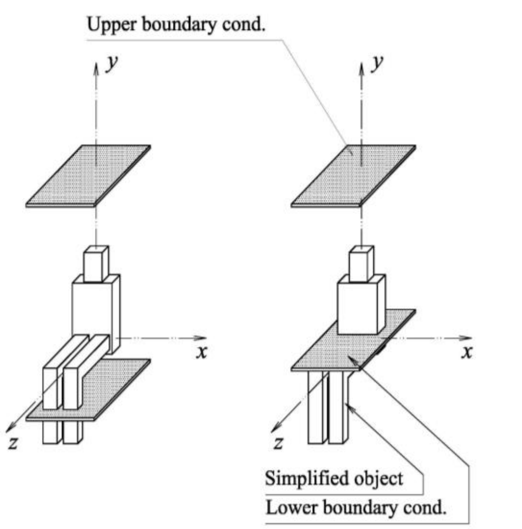

Figure 2 Simplification of the heat source (substitution by two boundary conditions)

The original model of heat source (i.e. model of thermal manikin, see Figure 1) was substituted by significantly simplified object, whose surface was adiabatic. Cylindrical legs and head were replaced by rectangular blocks. The object acts only as an obstacle to the indoor air flow and it does not release heat to the surroundings. This brings lot of advantages especially in the process of meshing – it is not necessary to generate dense boundary layer mesh surrounding the object's surface and the computational cells in the vicinity of the object can be bigger. Also neglecting the cylindrical parts of the model makes meshing easier.

The velocity and temperature profiles imposed on *pizza-boxes* were determined from the previous simulation with the detailed model of thermal manikin. Horizontal grids of 240 points covering the area 1000 x 600 mm were set at three heights – at the level of calves, waist and 700 mm above the head of thermal manikin. Each of the velocity components u_x, u_y, u_z, temperature T and turbulence parameters k and ε were obtained as an average of 120 values acquired during 120 s of computational time, see below. The averaged profiles of velocity, temperature, k and ε were imported into the simplified computational cases and set as boundary conditions on each *pizza-box*.

CFD SIMULATIONS AND METHOD OF RESULTS EVALUATION

CFD simulations were solved using the software Fluent 6.3 as non-isothermal flow of incompressible air with the influence of thermal expansion (so called Boussinesq approximation).

The boundary conditions were equal for all the simulations in order to enable mutual comparison of the results. The surface temperature of the room walls was 19 °C, their emissivity was 0.94 and the emissivity of the thermal manikin surface was 0.98.

The flow in the proximity of the walls was solved using dense boundary layer mesh (without wall functions) considering the influence of temperature and buoyancy on the turbulence. Two equation k-ε turbulence model by Shih et al. (1995) – so called *realizable* – was used for the simulations evaluated in this paper. Heat radiation was simulated using the surface-to-surface (*S2S*) model (Fluent, 2006).

The *Body Force Weighted* scheme was chosen for the discretization of pressure equation as it is recommended for solving buoyancy driven flows (Fluent, 2006). The convective terms of the equation were solved using 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 order of magnitude of 10^{-5} or lower.

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

Images of air flow in the room (velocity isolines), as presented in Figure 3, were recorded for every computational case at the time 600 s (i.e. termination of the simulation, when the flow can be considered as fully developed). The isolines were recorded in the planes *x-y* and *z-y* intersecting the center of the thermal manikin's head.

The evaluation of the results was aimed at the influence of heat source substitution on the formation of the thermal plume and also at the influence of subsidiary zone positioning on the simulation results (i.e. three cases with difference positioning of *pizza-boxes* were compared). Moreover, the influence of ambient temperature and the influence of turbulence model were assessed.

INFLUENCE OF THE HEAT SOURCE SUBSTITUTION

Images of simulated velocity field in the experimental chamber were recorded for each computational case. The isolines of velocity magnitude in the vertical plane *z-y* (side view) intersecting the center of the thermal manikin's head are displayed in Figure 3. The simulations with the heat source substituted by *pizza-boxes* were compared to the simulation with the model of thermal manikin, which is considered to be the reference case (as all the simplified boundary conditions were determined from this case).

The thermal plumes patterns are close to each other in all the computational cases at the level above the upper *pizza-boxes*. The thermal plumes adhere to the ceiling of the room and spread further to the vertical walls in a very similar way in all the simulations.

The formation of thermal plumes under the upper *pizza-boxes* differs more significantly in the individual simulations. There is no convective flow formed around the manikin in the simulation with just one *pizza-box* (Figure 3, case B). This inaccuracy was solved by placing another subsidiary zone to the lower level above the floor of the room. The convective current generated by the lower *pizza-box* flew around the object substituting the thermal manikin and entered the upper zone, which generated the following plume rising towards the room ceiling.

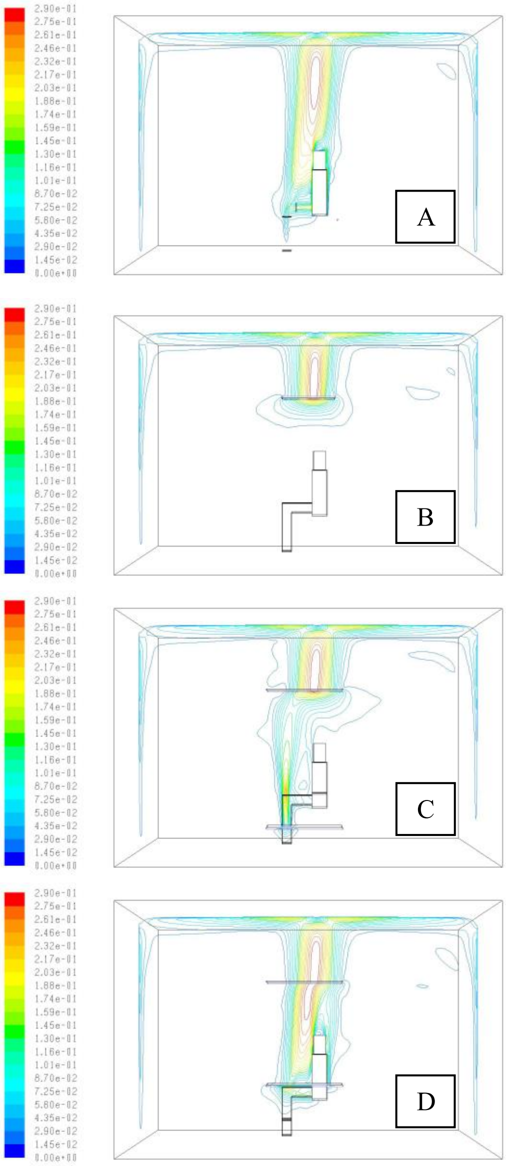

Figure 3 Thermal plume around detailed and substituted model of heat source (Velocity isolines [m/s])

The flow in the computational case with second *pizza-box* positioned at the height of calves (Figure 3, case C) is formed also in the area under the higher *pizza-box*. However, this flow is significantly different from the reference flow (as observed in the simulation with detailed model of thermal manikin – Figure 3, case A). The thermal plume induced by the simplified boundary condition rises only above the knees of the manikin and partly adheres to its front. There is a very weak flow above the head under the upper *pizza-box* and no flow along the back of the model.

Flow in the case with second *pizza-box* positioned at the level of waist (Figure 3, case D) is the closest to the reference flow. The generated thermal plume rises above the thighs and appropriately adheres to the manikin. The flow above thighs significantly influences the rising thermal plume, which is in accordance with findings of Zukowska et al. (2007). The imperfection in this case is caused by insufficient flow along the back of the manikin, which would, after merging with the flow above the legs, bend the resulting thermal plume more towards the manikin's back.

Velocity profiles

Images of velocity field give a good preview of the simulation results. However, for more exact evaluation of the thermal plumes it is better to compare velocity and temperature profiles. They were determined for every computational case in several heights above the manikin. The results at the heights of 0.225 m and 1.225 m above the thermal manikin in the vertical plane *y-z* (side view) intersecting the center of the manikin's head are displayed below.

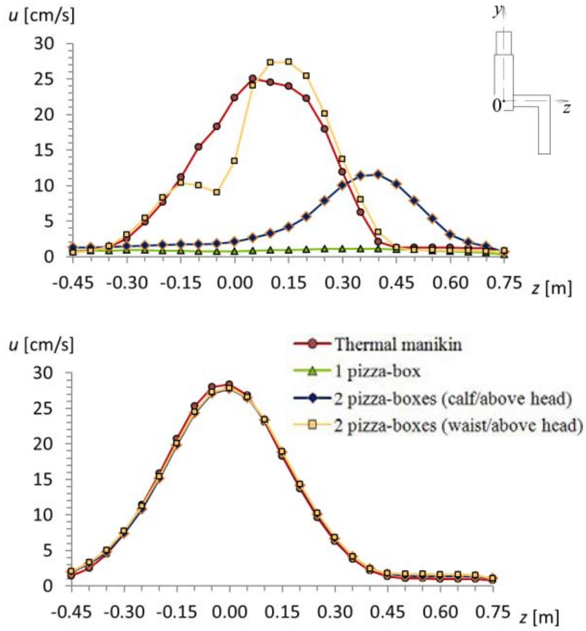

Figure 4 Velocity profiles in vertical plane y-z, height y = 0.975 m (upper) and y = 1.975 m (lower)

The velocity profiles determined for each simulation are close to each other especially at the higher levels of the room, see Figure 4. There are apparent bigger differences between individual simulations at the lower levels (around the thermal manikin and under the upper *pizza-box*). The simulation with two *pizza-boxes*, where the lower one is positioned at the waist level, is the most corresponding to the reference case. However, the thermal plume is still influenced by insufficient flow along the back of the manikin. The flow velocity is too high above the legs of the manikin and on the other hand too low above its head.

Temperature profiles

Temperature profiles of thermal plumes at the higher levels correspond to each other in all the simulations, see Figure 5. However, in the lower levels there is again apparent the inaccuracy of induced thermal flows. The thermal plume simulated with two *pizza-boxes* where the lower one is positioned to the height of waist, is the closest to the reference case.

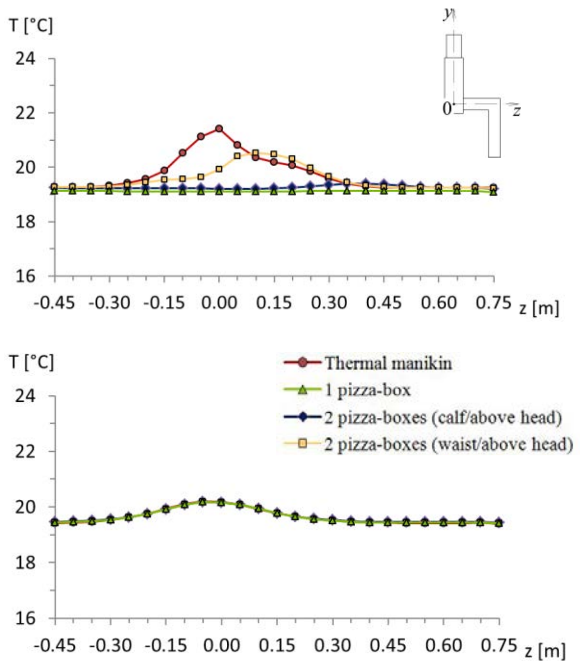

Figure 5 Temperature profiles in vertical plane y-z, height y = 0.975 m (upper) and y = 1.975 m (lower)

Computational demands

Table 1 summarizes the total numbers of computational cells in all the cases and computational times necessary for 10 iterations. All the simulations were calculated in the same way and with identical set-up during the testing calculations. The flow was considered as unsteady and 10 iterations were calculated for each time step of 0.1 s. This test was performed on the computer with 8 CPUs AMD Opteron 8378 (frequency 2.4 GHz) and 128 GB of operational memory. From the summarized values it is possible to see the advantage of substituting heat source by a simplified boundary condition. The size of the computational case as well as the computing time necessary for simulation are less than half of the case with detailed model of thermal manikin.

Table 1
Comparison of mesh cell number and computing time

COMP. CASE	MESH CELLS	COMP. TIME
Thermal manikin	3 569 984	130 s
1 *pizza-box*	1 469 837	45 s
2 *pizza-boxes* (calf/above head)	1 548 459	51 s
2 *pizza-boxes* (waist/above head)	1 538 362	51 s

FACTORS AFFECTING SPECIFICATION OF SIMPLIFIED BOUNDARY CONDITIONS

The detailed model of a heat source, which serves as the basis for the replacement boundary conditions to be afterwards used with the simplified model should be as realistic as possible. The factors affecting both physical behavior of the thermal plume as well as its simulation should be taken into account.

For a given heat source, the generated thermal plume is mainly influenced by the ambient air temperature. In CFD simulation, development of thermal plume is influenced by the choice of a turbulence model. These factors can affect the profiles of velocity, temperature and turbulence parameters, which are set as boundary conditions on the *pizza-boxes*.

Influence of turbulence model on simulated thermal plume

Two-equation turbulence models, such as k-ε and k-ω, are the most frequently used for CFD simulations in the field of indoor environmental engineering. The performance of four two-equation turbulence models in thermal plume simulation was assessed in the study published elsewhere (Zelenský et al., 2013). The simulations with different turbulence models were compared mutually and also with measurements by a thermal imaging camera, because solely inter-model comparison cannot indicate which turbulence model gives more realistic results.

The main outcome of the study was that turbulence models of k-ε type are much more appropriate for the simulation of thermal plumes, then k-ω models – see Figure 6 as an example of results.

The choice of a particular turbulence model affects especially the spreading rate of thermal plume and consequently the axial velocity magnitude and also the axial velocity decay with increasing height above the heat source.

The thermal plume simulated with k-ω turbulence model was significantly narrower than the measured one. Also the vertical decay of temperature was slower in simulations with k-ω model than in measurements.

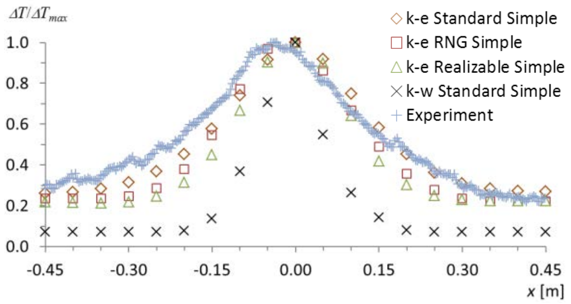

Figure 6 Non-dimensional temperature profiles in vertical plane x-y, height y = 1.375 m

Influence of ambient air temperature on simulated thermal plume

One can expect that the ambient temperature influences the values of air temperature in the thermal plume. Nevertheless, the boundary condition at *pizza-box* could be defined independently of different thermal environments when using non-dimensional temperature profiles. This could make our approach universal with respect to different ambient thermal conditions.

To prove this we compared velocity and temperature profiles from two simulated computational cases with different ambient temperatures 19 °C and 24 °C, see Figures 7 to 9. The simulations were performed and evaluated in the same manner as the other simulations described in this paper, using the 'realizable' k-ε model of turbulence.

It is apparent that the air temperature profiles, which are compared in the Figure 7, reflect the difference in the ambient temperature. The higher is the ambient temperature, the higher are air temperatures in the thermal plume. On the other hand, the velocity profiles are almost identical, see Figure 8.

The problem with air temperature shift can be solved by using non-dimensional temperature profiles. Figure 9 shows the non-dimensional profiles of air temperature defined at given height above the heat source as:

$$\frac{\Delta T}{\Delta T_{\max}} = \frac{T(x) - T_{amb}}{T_{\max} - T_{amb}} \qquad (1)$$

where:

$T(x)$ is the temperature at the distance x from the vertical axis y;

T_{max} is the maximum temperature in the plume at the given height;

T_{amb} is the ambient temperature.

The non-dimensional temperature profiles are not significantly influenced by the ambient temperature, as the temperature profiles obtained from the simulation with ambient temperature 19 °C are close to the temperature profiles from the simulation with ambient temperature 24 °C.

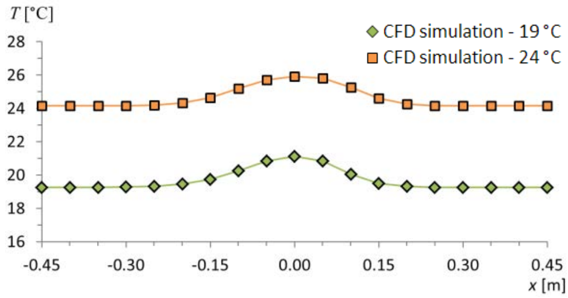

Figure 7 temperature profiles in vertical plane x-y, height y = 0.975

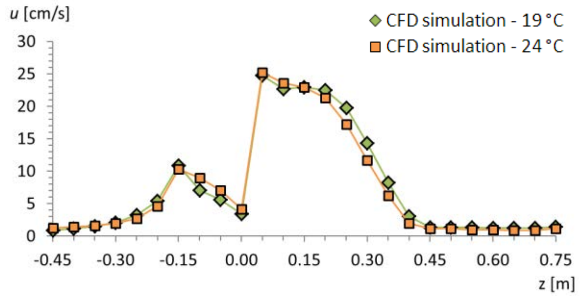

Figure 8 Velocity profiles in vertical plane y-z, height y = 0.775 m

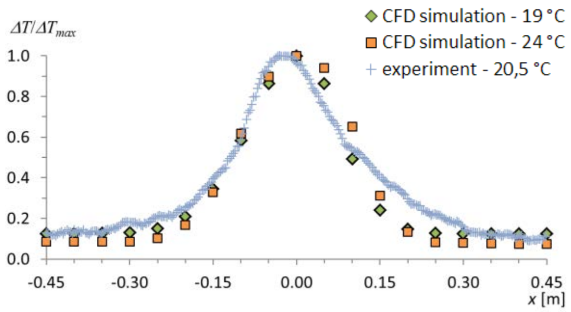

Figure 9 Non-dimensional temperature profiles in vertical plane x-y, height y = 0.975 m

CONCLUSION

Four different computational models of a sitting thermal manikin were compared in order to test the performance of proposed method to simplify heat sources by their substitution with simple boundary conditions generating artificial thermal plume. The models were created in three levels of simplification.

The proposed method to substitute heat sources by simplified boundary conditions which are set at the surfaces of subsidiary zones in the area of original thermal plume (substitution by *pizza-boxes*) could be utilized for simulating the effect of heat source on the surrounding environment as a whole. It would not be appropriate to use the methodology for simulating airflow in the close vicinity of heat sources, as the generated artificial thermal plume is showing big inaccuracy in this region.

The simplification in the layouts described in this paper could be suitable especially for the evaluation of air flow in the higher levels, where the thermal plumes in simplified cases correspond very well to the reference flow above the detailed model of the heat source. The simplification could be applied for example to study how indoor heat sources influence forced flow from the ventilation inlets placed below ceiling and how this affects the whole indoor environment.

From the captured velocity isolines it is apparent, that the positioning of subsidiary zones has a big influence on the simulation results. Thermal plumes from individual computational cases differ from each other especially at lower levels (under the upper subsidiary zone and around the object substituting the heat source). The thermal plume in the model with lower subsidiary zone placed in the level of waist is the most corresponding to the reference case, even though that it is still partly inaccurate. However, after further amendments this inaccuracy could be reduced and the proposed method could lead to more effective CFD simulations of indoor environments with heat sources. The proposed method enables simplification of the heat source geometry and also of the computational mesh around it, but it preserves the thermal plume pattern above the heat source.

Lower demand for computing time of the cases with simplified models of the heat source (cases with *pizza-boxes*) is apparent from the comparison of computational times necessary for 10 iterations. The proposed method could be therefore utilized for simulations of environments with high number of heat sources. Theatre halls, lecture rooms and stadiums can be mentioned as example applications. Detailed CFD simulations of these environments are difficult due to high computational demands – it is not viable to model each heat source in detail and calculate the convective flow around each of them separately. However, the thermal plumes formed above them should be simulated as precisely as possible, because thermal plumes rising above multiple heat sources can merge and form a single thermal plume with relatively high momentum. It can affect, possibly in a negative way, air flows from ventilation diffusers and consequently the indoor air flow patterns and indoor environment quality.

NOMENCLATURE

u_x, u_y, u_z	= velocity components	[m/s]
T	= temperature	[K]
k	= turbulence kinetic energy	[m^2/s^2]
ε	= turbulent dissipation rate	[m^2/s^3]
ω	= specific dissipation rate	[1/s]

ACKNOWLEDGEMENT

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

REFERENCES

Awbi, H. B. 2003. *Ventilation of Buildings.* 2nd ed. London: Spon Press.

Borges, C. M., Quintela, D. A., Brites, G. N., Gaspar, A. R., & Costa, J. J. 2007. Analysis of thermal plumes generated by a seated person, a thermal manikin and a dummy. *Proceedings of the 8th International Conference on Air Distribution in Rooms – Roomvent 2007,* Copenhagen, Denmark, 1, 253-256.

Fluent Inc. 2006. *Fluent 6.3 User's Guide.* Lebanon: Fluent Inc.

Koiš, G. 2009. *Analýza proudů v klimatizovaném prostoru* [MSc thesis, in Czech]. Czech Technical University in Prague, Department of Environmental Engineering.

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. 2012. Model sedící osoby jako zdroje tepla ve vnitřním prostředí [in Czech]. *Vytápění, větrání, instalace*, *5*, 22–26.

Zelenský, P., Barták, M., Hensen, J. L. M. & Vavřička, 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.

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

Zukowska, D., Melikov, A., & Popiolek, Z. 2007. Thermal plume above a simulated sitting person with different complexity of body geometry. *Proceedings of the 10th International Conference on Air Distribution in Roooms – Roomvent 2007,* Helsinky, Finland, 3, 191-198.

Zukowska, D., Melikov, A., & Popiolek, Z. 2010a. Impact of boundary conditions on the development of the thermal plume above a sitting human body. *Proceedings of the 10th REHVA World Congress „Sustainable Energy Use in Buildings" – Clima 2010,* Antalya, 2–7.

Zukowska, D., Melikov, A., & Popiolek, Z. 2010b. Determination of the integral characteristics of an asymmetrical thermal plume from air speed/velocity and temperature measurements. *Experimental Thermal and Fluid Science*, *34*(8), 1205–1216.