CZECH TECHNICAL UNIVERSITY IN PRAGUE

FACULTY OF MECHANICAL ENGINEERING

DOCTORAL THESIS STATEMENT

CZECH TECHNICAL UNIVERSITY IN PRAGUE
FACULTY OF MECHANICAL ENGINEERING
DEPARTMENT OF ENVIRONMENTAL ENGINEERING

DOCTORAL THESIS STATEMENT

OPTIMUM REPRESENTATION OF HEAT SOURCES IN SIMULATIONS OF AIR FLOW IN INDOOR ENVIRONMENT

Ing. Petr Zelenský

Doctoral Study Programme: Mechanical Engineering

Study Field: Environmental Engineering

Supervisor: prof. dr. ir. Jan L. M. Hensen

Supervisor Specialist: Ing. Martin Barták, Ph.D.

Prague, September 2018

The doctoral thesis was produced in full-time manner Ph.D. study at the Department of Environmental Engineering of the Faculty of Mechanical Engineering of the CTU in Prague.

Candidate: Ing. Petr Zelenský
[1] Department of Environmental Engineering,
Faculty of Mechanical Engineering, CTU in Prague
Technická 4, 160 00 Prague 6, Czech Republic

Supervisor: prof. dr. ir. Jan L. M. Hensen
[1] Department of Environmental Engineering,
Faculty of Mechanical Engineering, CTU in Prague
Technická 4, 160 00 Prague 6, Czech Republic
[2] Department of the Built Environment,
Eindhoven University of Technology (TU/e)
De Zaale, 5600 Eindhoven, Netherlands

Supervisor specialist: Ing. Martin Barták, Ph.D.
[1] Department of Environmental Engineering,
Faculty of Mechanical Engineering, CTU in Prague
Technická 4, 160 00 Prague 6, Czech Republic

Opponents:

The doctoral thesis statement was distributed on:

The defence of the doctoral thesis will be held on:

at the meeting room nr. B1-819 of the Department of Environmental Engineering of the Faculty of Mechanical Engineering of the CTU in Prague, Technická 4, Praha 6, before the Board for the Defence of the Doctoral Thesis in the branch of study Environmental Engineering.

Those interested may get acquainted with the doctoral thesis concerned at the Science and Research Department of the Faculty of Mechanical Engineering of the CTU in Prague, Technická 4, Praha 6.

Assoc. Prof. Ing. Jiří Hemerka, CSc.
Garantor of the field of study Environmental Engineering
Faculty of Mechanical Engineering of the CTU in Prague

SHRNUTÍ

Výzkum byl motivován problematikou správné distribuce vzduchu ve velkých prostorech s velkým počtem zdrojů tepla a/nebo variabilním obsazeností návštěvníky, jako jsou např. přednáškové síně, divadla, kina, atria apod. Projektanti systémů techniky prostředí pro tyto prostory často čelí nejistotě, zda navržená řešení splní požadavky na kvalitu vnitřního prostředí.

Dosažení vysoké kvality vnitřního prostředí velkých prostor při současném zohlednění energetických nároků může být náročné. Jedním z nástrojů, které mohou pomoci analyzovat a pochopit složité interakce ve vnitřním prostředí, je modelování a simulace pomocí počítačové mechaniky tekutin (CFD). Nicméně v současné době existují omezení CFD, vyplývající zejména z omezené kapacity dostupné výpočetní techniky. Numerické modely simulovaných prostor je tak vždy nutné do určité míry zjednodušovat.

Byla vyvinuta nová metoda modelování zdrojů tepla pro studium proudění vzduchu ve vnitřním prostředí s cílem snížit výpočetní nároky CFD simulací se zdroji tepla a zajistit spolehlivost získaných výsledků. Výzkum byl založen na numerickém modelování a simulacích v CFD softwaru ANSYS Fluent. Bylo provedeno několik numerických a experimentálních studií zaměřených na problematiku přirozené konvekce ve vnitřním prostředí a její simulaci:

- byly stanoveny optimální parametry výpočetní sítě v blízkosti ohřívané stěny modelovaného zdroje tepla;
- byl stanoven optimální model turbulence pro CFD simulace s převažujícím účinkem přirozené konvekce na proudění vzduchu;
- byla zkoumána citlivost konvekčních proudů nad zdroji tepla na okolní teplotu;
- byla vyhodnocena vzájemná interakce konvekčních proudů nad více zdroji tepla, které jsou umístěny blízko u sebe.

Byla zpracována řada simulací trojrozměrného konvekčního proudění ve vnitřním prostředí. K validaci navržené metody modelování zdrojů tepla byly použity výsledky vlastního měření v experimentální místnosti společně s porovnáním modelů s různým stupněm zjednodušení. Navržená metoda byla adaptována tak, aby odrážela různé tepelné podmínky okolního prostředí a byla vytvořena uživatelsky definovaná funkce (UDF) pro software ANSYS Fluent. Metoda byla následně aplikována ve vybrané případové studii a byla demonstrována její použitelnosti v praktických aplikacích.

SUMMARY

The research was motivated by the issue of proper air distribution in large indoor spaces with a high number of heat sources and/or variable occupancy patterns, such as e.g. lecture halls, theatres, cinemas, atriums etc. Designers of heating, ventilation and air conditioning (HVAC) systems for these spaces frequently cope with the uncertainty whether the proposed design will meet the requirements for adequate indoor environment quality.

Achievement of high indoor environment quality with simultaneous consideration of energy demands in large indoor environments can be challenging. Modelling and simulation tools, such as computational fluid dynamics (CFD) can help to analyse and understand the complex interactions in the indoor environment. However, there are currently some limits of CFD arising especially from the limited capacity of available computers and resulting in the necessity to simplify numerical models of the simulated cases.

A new modelling method to represent models of heat sources for indoor air flow studies was developed in order to reduce the computational burden of CFD simulations with heat sources and ensure high reliability of the obtained results. The research was based on numerical modelling and simulation in the CFD software ANSYS Fluent. Several numerical and experimental studies were conducted, targeting issues related to natural convection indoors and its simulations:

- optimal characteristics of the near-wall region mesh around the modelled heat source was determined;
- an optimal model of turbulence for CFD simulations with prevailing effect of natural convection on the air flow was proposed;
- the sensitivity of the thermal plumes above heat sources on the ambient air temperature conditions was studied;
- the mutual effect of thermal plumes rising above multiple heat sources positioned close to each other was assessed.

Various cases of three-dimensional convective flow in the indoor environment were simulated. The results of the experiment performed in the experimental room were used for validation of the proposed modelling method to represent heat sources, together with comparative testing of models at different levels of simplification. The proposed method was adapted to reflect different thermal conditions of the ambient environment and a user defined function (UDF) for ANSYS Fluent was developed. The method was applied in a selected case study in order to demonstrate its usability in practice.

TABLE OF CONTENTS

NOMENCLATURE

k	[m²/s²]	turbulence kinetic energy
t	[°C]	temperature
u	[m/s]	velocity
u_x, u_y, u_z	[m/s]	velocity components
u^+	[-]	dimensionless velocity
y^+	[-]	dimensionless wall distance
A	[m²]	area
$\dot{I}$	[kg·m/s²]	momentum transfer rate
T	[K]	temperature
$\dot{V}$	[m³/s]	volume flow rate
ε	[m²/s³]	turbulent dissipation rate
μ	[Pa·s]	dynamic viscosity
ρ	[kg/m³]	density
τ_w	[Pa]	wall shear stress
ω	[1/s]	specific dissipation rate
x	[m]	position coordinate (Cartesian axis)
y	[m]	position coordinate (Cartesian axis)
z	[m]	position coordinate (Cartesian axis)

List of indexes

x, y, z	Cartesian axis
max	maximum
min	minimum
amb	ambient

List of abbreviations

BES	Building energy simulation
CFD	Computational fluid dynamics
HVAC	Heating ventilation and air conditioning
IT	Information technology
SBC	Subsidiary zone with simple boundary condition (see Chapter 6)
UDF	User defined function

1 INTRODUCTION

The main aim of the current research is to target issues of improper air distribution risk and discomfort risk in large indoor spaces with large number of heat sources and/or variable occupancy patterns, such as atriums, lecture halls, theatres and other entertainment facilities, etc. Designers of HVAC systems for these spaces frequently cope with the uncertainty of the proposed design meeting requirements for adequate thermal comfort and proper air distribution under different conditions of occupancy.

One of the techniques that can help to address the design issues of such places is computational fluid dynamics (CFD) modelling and simulation. It is increasingly applied in building design practice, as it allows the user to study the indoor environment at several levels of resolution at the same time, providing both overall insight and detailed information. Thus, it helps to test whether the specified requirements can be met before the design realization. Although the availability of CFD simulation is rising, there are still some limits, especially in the capacity of available computers. It is very often necessary to simplify the reality to a certain level. However, as the quality of the results is determined by the accuracy of the input, the simplification must be done in a proper way, considering the aim of the simulation.

A new modelling method to represent heat sources in CFD simulations of the air flow in the indoor environment has been proposed. It aims to reduce computational demands of CFD simulations with indoor heat sources and ensure high reliability of the obtained results. Several numerical and experimental studies related to the simulation of thermal plumes around heat sources have been carried out during the development of the method and its adaptation for practical use. The method was tested and applied in a selected case study in order to show its usability for practical applications.

2 RESEARCH BACKGROUND

The design methods and tools in HVAC engineering are changing with growing demand and increasing capacity of IT. Standard (manual) methods are gradually complemented by computer-based alternatives. Simulations generally have become one of the most important engineering tools in design of buildings and HVAC systems. This is caused by the increasing number of non-standard solutions in modern building construction, which can be hardly dealt with standard design methods.

The appropriate method for detailed modelling of the indoor environment aiming to uncover for example air flow patterns in rooms, temperature conditions, indoor air quality and contaminant distribution is CFD (Djunaedy et al. 2003; Hensen 2004). It can provide very detailed flow information, which can not be obtained by the other available methods. Although it generally has very high demands on computational power, there are various application, which benefit from the high complexity of the obtained results.

2.1 Heat sources in indoor environment

The indoor heat sources can be classified into three basic groups: occupants, equipment and strong heat sources. Heat sources providing thermal comfort in the indoor environment can be identified as a separate category. The following text is focused on heat sources without mechanically induced air flow.

Natural air flow formed above a heat source is called convective stream or thermal plume. It directly influences the air flow pattern in the surrounding environment. In rooms without significant air mixing, even the weak convective plumes have very strong influence on the overall flow pattern, temperature gradient, contaminant distribution and thermal stratification. The momentum of convective flow formed around strong or multiple heat sources can be comparable, or even higher, than the momentum of the air flow supplied to the room by air-conditioning units. In such cases, the air flow from air distribution elements can be influenced, possibly in a negative way (Zukowska 2011; Awbi 2003) which can affect comfort of occupants. On the other hand, in the rooms with displacement ventilation, thermal plumes have special importance, as they provide driving force for natural ventilation and thus positively influence the quality of inhaled air (Skistad et al. 2002).

The upward rising convective flows are influenced by the shape and intensity of a heat source, its position and conditions of surrounding environment: room size and geometry (particularly the ceiling height), temperature, temperature stratification, arrangement of other heat sources and sources of forced flows, etc. (Skistad et al. 2002; Awbi 2003). The stability of thermal plumes is an important issue as well. It has been previously noticed that the convective flows do not reach steady state even when the conditions in the plume surroundings seem to be constant (Zukowska et al. 2010a; Deevy et al. 2008; Hyldgaard 1998). This phenomenon is called plume axis wandering and it is characterized as a periodical deviation of the plume axis from its mean vertical position.

For a limited number of rather simplified configurations, it is possible to solve convective flows analytically. However, if we deal with complex real world scenarios the analytical methods are insufficient. Very suitable tool for solving these cases can be experiments or CFD simulations, where simulations are nowadays generally cheaper and can be more efficient.

2.2 CFD simulations and simplified modelling

Effective use of CFD simulations depends on the capabilities of IT, which has undergone a very rapid development in the recent decades, resulting in a significant rise of accessible computer power. However, there are still considerable limitations. The more complex the problem is that we are solving, the longer its computational time takes and the more difficult it is to prepare the computational models. One of the ways to reach the target of reasonable computational time is simplification of numerical models used for the simulation, which, aside of the computational time, also reduces the time necessary for computational model preparation and meshing.

2.2.1 Models of indoor heat sources for CFD simulations and their simplification

There are various methods for modelling and simulation of indoor heat sources. The most straightforward is a detailed modelling. However, heat sources are often relatively small compared to the room size, with fine geometrical details requiring small cells of the surrounding numerical mesh to capture them and it is challenging to model them in detail. Moreover, there are big differences in properties of flow around the heat source and in the surrounding environment (differences in turbulent quantities, velocity magnitudes, velocity gradients, etc.). It can result in high number of control volumes in the domain and consequently slow calculation of the simulation. This may be addressed by simplified modelling.

When approaching the model simplification, we always have to take into consideration the question on hand and anticipate the possible influence of the simplification on the result of simulation (Nielsen et al. 2007). The method of simplification must be therefore carefully chosen. The common ways of heat sources simplification are:

- simplification of the shape and geometrical details;
- simplification of boundary conditions;
- simplification of operational characteristics.

However, these basic methods are not effective in all situations. It can be problematic to simulate, for example, the environment with high number of heat sources, as has been noticed at the Department of Environmental Engineering at the Faculty of Mechanical Engineering, CTU in Prague.

2.2.2 Turbulence modelling in indoor air flow simulations

Selection of the method to approximate turbulent processes in fluid flow is a very important part of CFD simulations; there are several approaches to deal with turbulence; they differ in complexity and accuracy (Zhai et al. 2007). The Reynolds decomposition and averaging of Navier-Stokes equations (RANS method) is usually used for CFD simulations for HVAC engineering applications (Zhai et al. 2007). In this approach, the mean flow quantities are solved directly and the effect of turbulent fluctuations is approximated on the basis of turbulence models.

A number of turbulence models have been developed for the RANS method. In HVAC engineering, the most frequently used turbulence models are the two-equation models such as k-ε and k-ω. It is for example model proposed by Launder and Spalding (1974), model according to Yakhot and Orszag (1986), model by Shih et al. (1995) or model developed by Wilcox (1988). Although there is currently a wide variety of different turbulence models to choose from, there is not yet a single, practical turbulence model that can reliably predict all turbulent flows with sufficient accuracy (Zhai et al. 2007). It is always necessary to consider problem on hand and physical phenomena in the solved simulation. The turbulence model must be carefully chosen according to it.

2.3 Modelling of air supply diffusers as an inspiration for the new modelling method

Appropriate description of the forced flow from air supply diffusers in CFD simulations is important for reaching reliable and trustful results (Srebric & Chen 2002; Nielsen et al. 2007). Similarly to heat sources, most of the diffusers are small compared to the size of the common rooms and they have complex geometry with a lot of details influencing the flow. There are also considerable differences of velocities inside the diffuser and in the room, which makes their joined simulation challenging.

This has been previously addressed by several methods of simplification describing how to more or less sufficiently deal with issues such as big differences in velocities or length scales in the CFD simulations (Srebric & Chen 2002). They can be generally classified into six categories – Direct description, Simplified Geometry, Momentum Method, Prescribed velocity method, Box method and Method of random blocking of CFD cells (Zhang et al. 2009).

The Box method is discussed in detail, considering the further steps of the research work. It determines flow and thermal conditions in certain distance from the diffuser. The quantities of the flow are determined in advance by a measurement and defined in a simulation as a boundary condition at an imaginary box surrounding the diffuser, see Fig. 2.1. The flow field inside the box is ignored and thus the diffuser itself does not have to be modelled (Nielsen 1997).

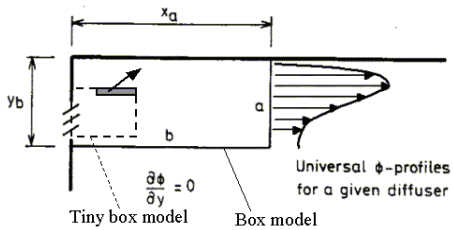

Fig. 2.1 – Box method of air supply diffusers modelling (Srebric & Chen 2002)

3 RESEARCH GOALS – NEW MODELLING METHOD TO REPRESENT HEAT SOURCES

The main goal of the presented research was to develop a new modelling method to represent heat sources in numerical models of indoor environment in order to enhance the use of CFD simulation in practice. It is targeting especially assessment of proper air distribution in large indoor spaces with a large number of heat sources and/or variable occupancy patterns (such as atriums, large meeting rooms, lecture halls, theatres, cinemas and other entertainment facilities, etc.). The method reflects the issues, which are not addressed by the basic ways of simplification currently used for numerical modelling of indoor heat sources, as discussed in the previous chapter. It should reduce computational demands of CFD simulations with heat sources and ensure high reliability of the obtained results.

As the issue of heat sources modelling is similar to modelling of air supply diffusers, the development of the new method was inspired by the currently used methods to model air supply diffusers for CFD simulations, particularly box method proposed by Nielsen (1997). Similarly to the jet streams from the air supply diffusers, the flow and thermal conditions of the thermal plume rising above a heat

source can be defined by previously determined boundary condition set at a certain distance above the heat source. Thus, the geometry of the heat source can be simplified and there is no need to create a fine mesh in the region around the heat source, which is otherwise necessary for correct heat transfer calculation. This results in an easier preparation of numerical models, significantly easier meshing and faster CFD simulation

The main challenges of the research, following the proposition of the new modelling method to represent heat sources in the CFD simulations of air flow indoors, were summarized into the following tasks:

- to assess sensitivity of computations on the selection of turbulence model and propose an optimal model of turbulence for cases with prevailing effect of natural convection on the air flow
- to assess physical correctness of the simulated results when using simplified model;
- to assess the optimal distance of the flow and thermal boundary condition from the heat source;
- to assess sensitivity of the method on the temperature of the ambient air;
- to assess usability of the method for CFD simulation with multiple heat sources (i.e. especially influence of the simplification on the merging of multiple thermal plumes);
- to summarize assets and drawbacks of the proposed method;
- to sum up limitations of the method, considering its possible applications for real world scenarios;
- to provide guideline for the future use of the developed method.

4 RESEARCH MATERIALS, TOOLS AND METHODS

The main part of the research was based on CFD simulations solved in a commercial software ANSYS Fluent. The numerical solution of governing equations in the ANSYS Fluent software is based on the finite volume method (Patankar 1980). All simulations were solved as non-isothermal flow of incompressible air. The flow was considered as unsteady. Two-equation turbulence models were used to close the RANS and thermal energy equations. A combined numerical-experimental study was done to assess the performance of four most common two-equation k-ε and k-ω turbulence models for simulations of thermal plumes, in order to determine the most appropriate model of turbulence.

Integration of governing equations in the entire boundary layer was used in most of the simulations. Therefore, the emphasis was placed on the numerical mesh near the heated surface. In these regions, $y^+ \leq 2.5$ was required for at least first seven cells near the wall, while the $y^+ \leq 1$ was required for the first cell by the heated surface (Wilcox 2006). The estimation of the y^+ value was based on the initial test simulation of a heat source with the same surface area and heat output as the detailed heat source, but with a simplified geometry. The required distance of the cell centre is defined by the formula (4.1), and it was calculated using the values τ_w and ρ estimated on the basis of this initial simulation.

$$y = \frac{y^+ \cdot \mu}{\sqrt{\rho \cdot \tau_w}} \tag{4.1}$$

4.1 Development steps towards proposition of the new modelling method

The proposition of the new modelling method to represent heat sources in CFD simulations of indoor air flow was done in two steps. First, a numerical simulation with detailed model of a sample heat source was conducted. The computational case was meshed by a fine mesh in order to ensure high quality of the simulation. The obtained results were empirically validated by comparison with experiment described in the following chapter. Some issues, as for example choice of an appropriate turbulence model for the simulations and the effect of the ambient air temperature on the thermal plume above the heat source were studied by comparison with the results from experiment as well.

Following the initial studies, the new method to represent heat sources was proposed. It was thoroughly tested and validated to uncover any possible errors and examine the ways of its application. The validation of the method was based on comparative testing of numerical models with different levels of simplification. The reference case for the inter-model comparison was the simulation with the detailed model of the sample heat source, which was previously empirically validated by comparison with the experimental results.

Thermal manikin resembling a sitting occupant was selected as the sample heat source to test and validate the method, see Fig. 4.1. Metal thermal manikin previously assembled by Koiš (2009) was used for the experiments.

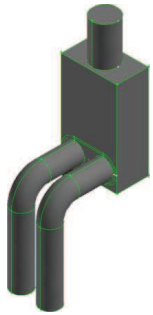

Fig. 4.1 – Thermal manikin

4.1.1 Empirical validation

Experiments in thermally stabile chamber were carried out in order to use relevant experimental data to ensure physical correctness of the developed method. The experiment was performed according to the recommendations found in literature (Jančík & Bašta 2011). A sheet of plain paper was stretched vertically above the centre of thermal manikin's head, see Fig. 4.2. The temperature field was continuously transferred from the thermal plume to the paper by convection.

Fig. 4.2 – Experimental chamber with thermal manikin, 2 measured arrangements

Thermal images of the paper surface were recorded using the infrared camera Flir T620 at 1 Hz sampling rate for the total period of 120 s. The horizontal temperature profiles evaluated at seven different heights above the manikin's head in two perpendicular planes (front view, side view) were time-averaged in order to take into account the fluctuation of the thermal plume (Zukowska 2011).

4.1.2 Validation by inter-model comparison

In order to test the proposed method, simulations with heat sources (sitting thermal manikins) represented by simplified numerical models were compared to the simulations with detailed models of the manikin, which were considered as reference cases (as they were previously validated by comparison with the experimental results). The comparative testing was based on the assessment of velocity magnitude isolines in two vertical planes intersecting the centre of the thermal manikins' head (front view and side view) and profiles of velocity and temperature, which were determined for every computational case in several heights above the models of the manikins.

Two types of numerical studies were performed during the inter-model comparison. The first type was a set of simple studies with one manikin placed in the middle of an enclosed room with floor dimensions 4.5 m x 4.5 m, ceiling height of 3 m and no obstacles. The second type was a study with four sitting thermal manikins placed close to each other in the middle of a room. The room had floor dimensions 5 m x 5 m and ceiling height of 5.6 m.

The inter-model comparison, aside of validation purposes, was expected to show especially the effect of the variant positioning of the subsidiary boundary condition and drawbacks of the developed modelling method to represent heat sources.

4.1.3 Usability demonstration

Following the validation and initial testing of the developed modelling method, a case study was performed in order to investigate the usability of the method for real-life situations. The main target was to show applicability of the method in practice and uncover possible drawbacks of the theoretical propositions. The previously developed modelling approach to represent heat sources in air flow modelling simulations was applied in a real scenario of a recently refurbished former church built in the 14th century, now used as a concert and conference hall with up to 350 visitors staying for different periods during each day. Two cases of two different occupancy scenarios were simulated.

5 DEVELOPMENT OF THE MODELLING METHOD

5.1 Numerical mesh for simulations of heat transfer by natural convection

Development and validation of the proposed method was based on CFD simulations of thermal manikin, placed in the middle of an experimental chamber. Discretisation of the whole domain was influenced mainly by appropriate meshing of near-walls regions around the heat sources, which are crucial for correct simulation of heat transfer by natural convection (Wilcox 2006). The recommendations found in literature were followed (see Chapter 4). The height of the first cell, growth rate and total number of the layers of the boundary layer mesh around the heated surfaces and walls of the room are shown in Tab. 5.1.

Maximal dimension of the cells in the computational domain was 50 mm. The numerical mesh was refined around the thermal manikin, which was surrounded by cells with edge dimensions 12.5 mm (12.5mm cells), followed by 25mm cells. The region of the raising thermal plume (above the heat source) was also meshed with smaller, 25mm cells, see Fig. 5.1 and Fig. 5.2. A simple grid dependence study was performed.

Tab. 5.1 – Boundary layer mesh

Surface	Number of layers	Height of the 1st layer	Growth rate	y^+ of the first cell	Nr. of cells in viscous subl.
Manikin	20	0.21 mm	1.16	0.2	13
Room walls	7	3.50 mm	1.20	0.9	3

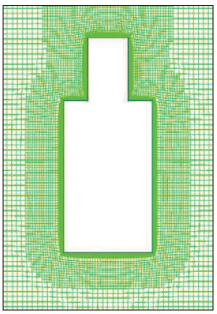

Fig. 5.1 – Cross section x-z (front view)

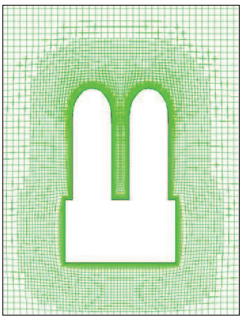

Fig. 5.2 – Cross section x-y (top view)

5.2 Influence of turbulence model on simulations of natural convection

The performance of four two-equation models (*k-ε Standard*, *k-ε RNG, k-ε Realizable* and *k-ω Standard)* in the thermal plume simulation was assessed. The results from several simulations with different models of turbulence were compared mutually, and also with experimentally obtained temperature profiles, see Chapter 5.4.

The sitting thermal manikin placed in the middle of a room was used as an example of heat source generating thermal plume in indoor environment. Profiles of velocity, temperature and turbulence kinetic energy of the thermal plume above the manikin, simulated with different turbulence models are compared in Fig. 5.3.

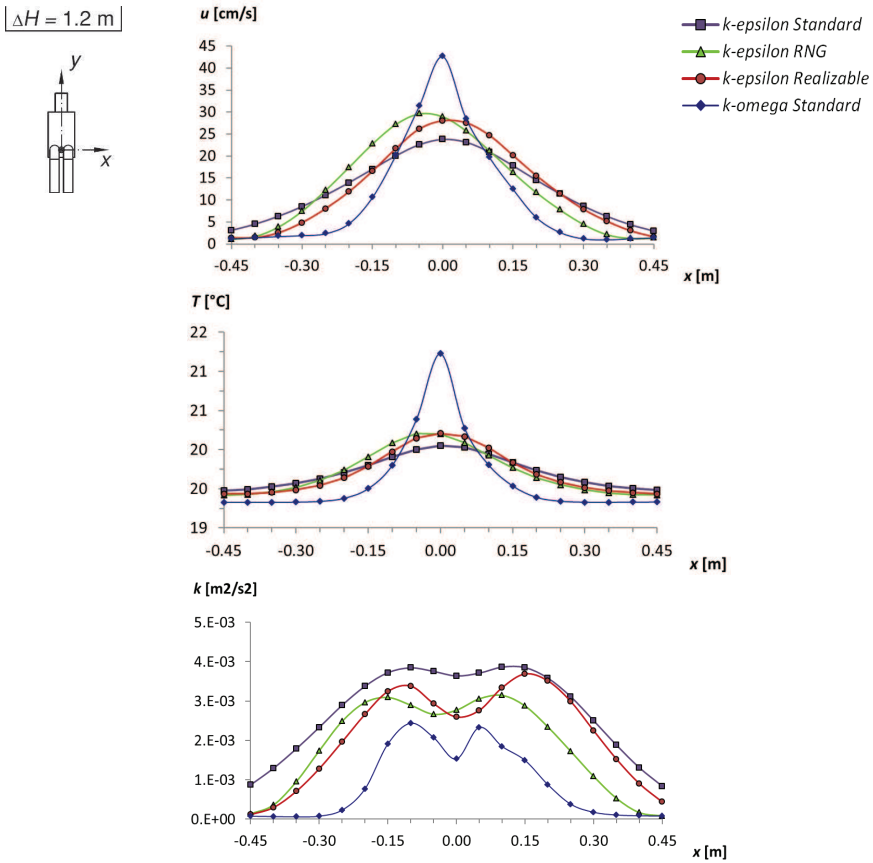

Fig. 5.3 – Profiles of velocity, temperature and turbulence kinetic energy in vertical plane x-y, 1.2 m above the manikin

The velocity and temperature profiles obtained from the simulations with all three tested *k-ε* turbulence models are very close to each other. On the other hand, they are very different from the outputs of simulation with the *k-ω Standard* model, which predicts much higher velocities and temperatures in the thermal plume axis; their magnitudes also decrease with the increasing height slower than in the case of the *k-ε* turbulence models.

The profiles of turbulence kinetic energy k indicate possible reasons for different behaviour of the thermal plume when using different turbulence models. The higher the turbulence kinetic energy k is, the higher the intensity of turbulent mixing in air flow and the higher the spreading rate of the thermal plume. It is obvious that the *k-ε Standard* model produces thermal plume with the highest turbulence intensity, and thus with the highest spreading rate. On the other hand, the *k-ω Standard* model shows the lowest values of k and the thermal plume in this case is therefore much narrower and spreads slower than in the simulations with *k-ε* models.

5.3 Influence of ambient temperature conditions on thermal plume

The effect of the ambient air temperature on thermal plume development is one of the important factors for modelling and simulation of heat sources. The influence of the ambient air temperature on the thermal plume was initially studied on the basis of analytical solutions describing convective flows (Awbi 2003; Popiolek 1987). It was expected that the velocity profile of the thermal plume does not depend on the ambient air temperature, providing that the heat output of a heat source is constant. Nevertheless, the ambient air temperature may influence the temperature profile. To prove this, the velocity and temperature profiles from two CFD simulations of one thermal manikin exposed to different air temperatures 19 °C and 24 °C were compared, see Fig. 5.4.

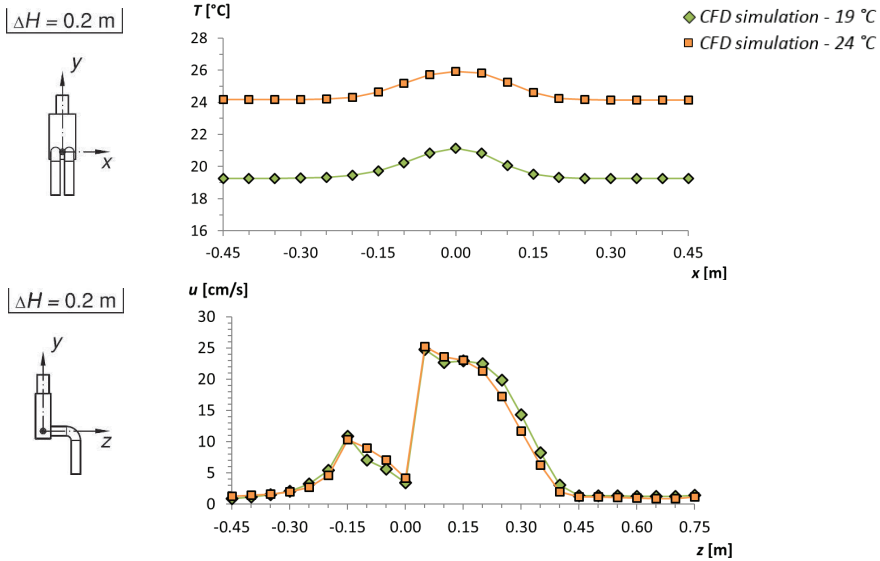

Fig. 5.4 – Temperature and velocity profiles 0.2 m above the heat source

It is obvious that the air temperature profiles, reflect the difference in the ambient temperature. The higher is the ambient temperature, the higher is the air temperature in the thermal plume. On the other hand, the velocity profiles in the simulations with different ambient air temperature are almost identical.

To make the new modelling approach universal with respect to different ambient thermal conditions, the temperature conditions of the thermal plume can be defined independently of different thermal environments by using a non-dimensional form defined using the following formula:

$$\frac{\Delta T}{\Delta T_{max}} = \frac{T(x) - T_{amb}}{T_{max} - T_{amb}} \tag{5.1}$$

Fig. 5.5 shows the non-dimensional profiles of air temperature evaluated at the height of 0.2 m above the heat source according to formula (5.1). It was concluded that the non-dimensional temperature profiles are not significantly influenced by the ambient air temperature, as the temperature profiles obtained from the simulation with two temperatures of the ambient air are close to each other.

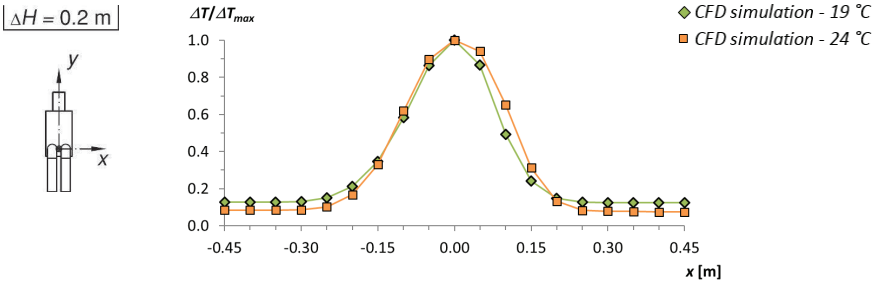

Fig. 5.5 – Non-dimensional temperature profiles in vertical plane x-y,

5.4 Comparison of simulations with detailed model with measurement

Temperature profiles obtained from the simulations described in Chapter 5.2 and Chapter 5.3 were compared with an experimental measurement by a thermal imaging camera. The non-dimensional temperature profiles were used for the comparison, as defined by the formula (5.1). Results of this study were used both to choose the most suitable model of turbulence for further simulations and to prove the correctness of the simulated results.

Fig. 5.6 presents the non-dimensional profiles of air temperature evaluated 1.2 m above the heat source in the experiment and in the simulations with different turbulence models. The temperature profile closest to the experimental data is the one simulated with the *k-ε Standard* model of turbulence, which was used for all the following simulations. On the other hand, the temperature profile simulated with the *k-ω Standard* turbulence model was the most different from the measured one, being significantly narrower.

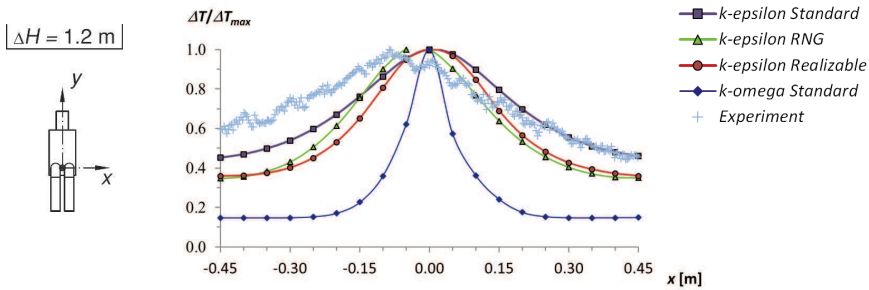

Fig. 5.6 – Non-dimensional temperature profiles in vertical plane x-y, 1.2 m above the manikin

Fig. 5.7 shows the comparison of the non-dimensional profiles of air temperature of the thermal plume from the experiment with the simulations at the ambient air temperatures 19 °C and 24 °C, respectively. It is possible to see that all the profiles show a good correspondence. The thermal plume in the experiment is slightly deflected to the left, which could have been caused by an instability of the rising convective flow that was affected by non-uniformities of boundary conditions in the real environment.

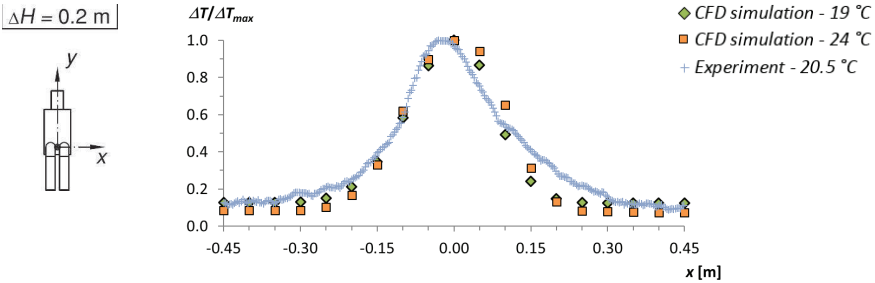

Fig. 5.7 – Non-dimensional temperature profiles in vertical plane x-y, 0.2 m above the manikin

6 RESULTS

The key outcome of the research was the proposition of a new modelling method to represent heat sources in indoor air flow simulations. It was inspired by the box method used for modelling of air supply diffusers (Nielsen 1997). It is based on the replacement of a heat source by one or more appropriately positioned simplified boundary conditions inducing a thermal plume potentially identical to that rising above the real heat source, see for example Fig. 6.1. Each of the boundary conditions substituting the heat source prescribes velocity, temperature and turbulence of the induced flow. These characteristics are determined in advance on the basis of reference CFD simulation with detailed representation of the heat source. The work-flow of the method is described in Tab. 6.1.

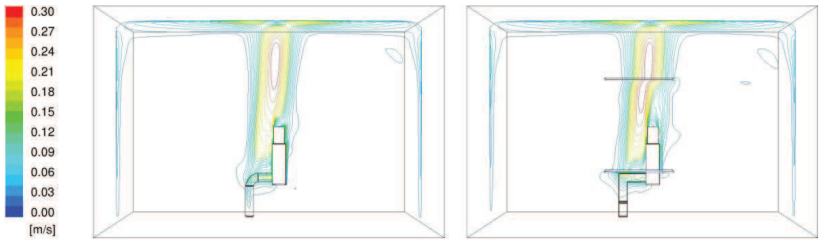

Fig. 6.1 – Thermal plume above a detailed and substitute model of heat source (velocity isolines) (Zelenský et al. 2012)

The way to simplify a model of a heat source using the proposed method was demonstrated on the example of a sitting thermal manikin. The rising thermal plume is, in the simplified case, induced by the velocity and temperature profiles set as the boundary conditions at subsidiary zones created at chosen heights in the place of the original thermal manikin model, see Fig. 6.2. The subsidiary zone with the prescribed simple boundary condition is, for better clarity, hereinafter referred to as *SBC*.

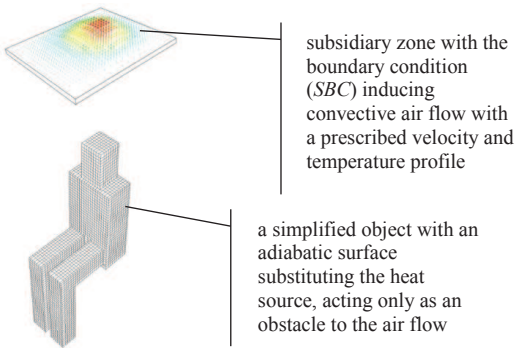

Fig. 6.2 – Simplified model of the sitting occupant

The original model of the heat source is substituted by a significantly simplified object with an adiabatic surface, which acts only as an obstacle to the indoor air flow and it does not release heat to the surroundings. This brings lots of advantages, especially in the process of meshing. It is not necessary to generate fine mesh in the boundary layer region surrounding the object's surface and the computational cells in the vicinity of the object can be bigger.

The velocity and temperature profiles imposed on the *SBCs* are determined in advance from the reference simulation with detailed model of the heat source. Each of the velocity components u_x, u_y, u_z, temperature T and turbulence quantities k and ε are obtained as an average of 120 values acquired during 120 s of computational time at the grid of measuring point set at the corresponding height. The averaged profiles of velocity components, temperature, k and ε are imported into the simplified computational cases and set as boundary conditions on each *SBC*. The induced thermal plume leaves the *SBC* from its top; equivalent flow rate is drawn through its bottom from the domain.

Tab. 6.1 – Work-flow of the new modelling method to represent heat sources

Step	Description
1	**Preparation of the numerical model for reference simulation** **→ detailed representation of a heat source:** - current best practice methods of numerical modelling for CFD simulations should be followed; - the numerical model of a heat source should be modelled in detail, without any significant simplification.
2	**Reference CFD simulation with a detailed model of the heat source:** - the best practice of CFD simulations should be followed; - heat and momentum transfer near the heat source surface is modelled using a fine boundary layer mesh (i.e. with integration of governing equations in the viscous sublayer); - turbulence can be modelled using RANS-based turbulence models – it can be advised to used the *k-ω Standard* model, see Chapter 5.4; - CFD simulation is performed to obtain velocity and temperature fields around the heat source, with focus on the generated thermal plume rising around the heat source.
3	**Determination of simplified boundary condition(s):** - air flow velocity, temperature and turbulent quantities are recorded from the reference CFD simulation in the rising thermal plume, using a grid of measuring points; - the obtained values should be time-averaged, in order to reflect the oscillation of the thermal plume.
4	**Creation of a numerical model with simplified representation of heat source(s):** - the heat source body is substituted by a simplified geometrical object with adiabatic surface, acting solely as an obstacle to the ambient flow (there is no heat transfer from the surface of the object); - the thermal plume is induced artificially, by boundary condition(s) prescribed in the place of each heat source.
5	**Real scenario simulation with simplified model(s) of heat source(s):** - one simulation with multiple simplified heat sources (i.e. large number of models created in the Step 4); - several simulations with repeated use of the simplified model (i.e. variant numerical studies using the model created in the Step 4).

6.1 Vertical position of substituting boundary condition

The optimal position of the *SBC* above the heat source was studied by Zelenský et al. (2017) in the case of thermal manikin situated in the middle of a room. Four computational cases with different vertical positions of the *SBC* were solved and compared, as shown in Fig. 6.3.

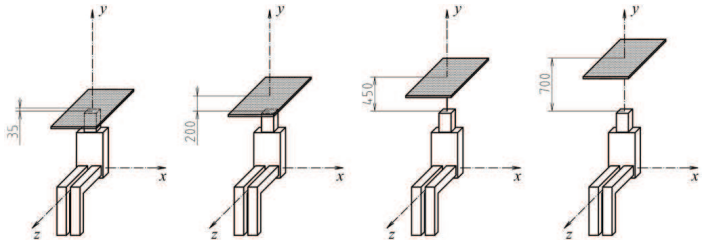

Fig. 6.3 – Vertical positions of the SBC above the manikin's head

The examples of isolines of velocity magnitude in the vertical plane *x-y* (front view) and *z-y* (side view) are displayed in Fig. 6.4 (case A is the reference simulation). The thermal plume patterns were close to each other in all the computational cases at the region above the *SBCs*. The thermal plumes adhere to the ceiling of the room and spread further towards the vertical walls in a very similar way in all the simulations. There is no convective flow formed around the manikins in the simplified simulations. The lower the vertical position of the *SBC*, the closer the simulated velocity field is to the reference simulation, as the thermal plume is induced nearer to the manikin's head.

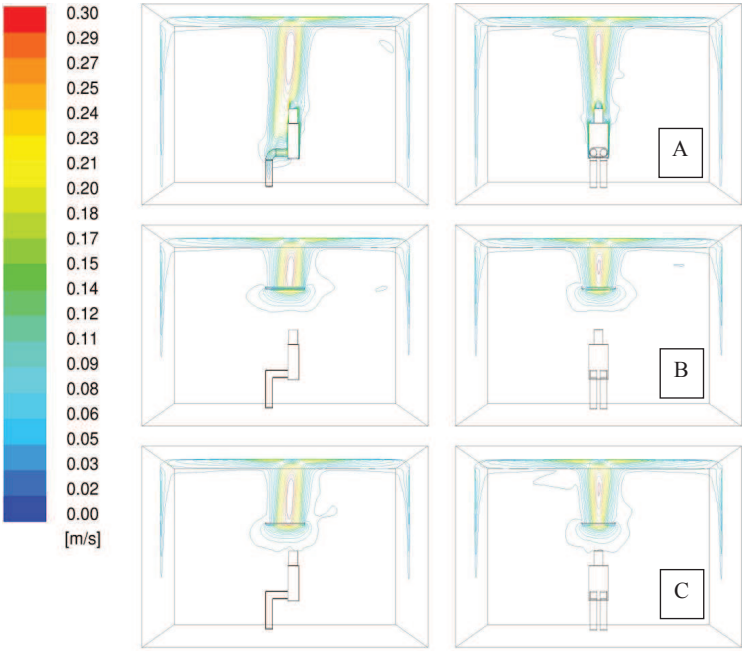

Fig. 6.4 – Velocity magnitude isolines in thermal plume above detailed and substitute model of heat source: A – detailed model; B and C – models with SBC 700 and 450 mm above the manikin

Velocity and temperature profiles

Velocity and temperature profiles were compared for more exact evaluation of the thermal plumes. The results at the chosen heights above the thermal manikin in the vertical plane *x-y* (front view) intersecting the centre of the manikin's head are displayed in Fig. 6.5 and Fig. 6.6.

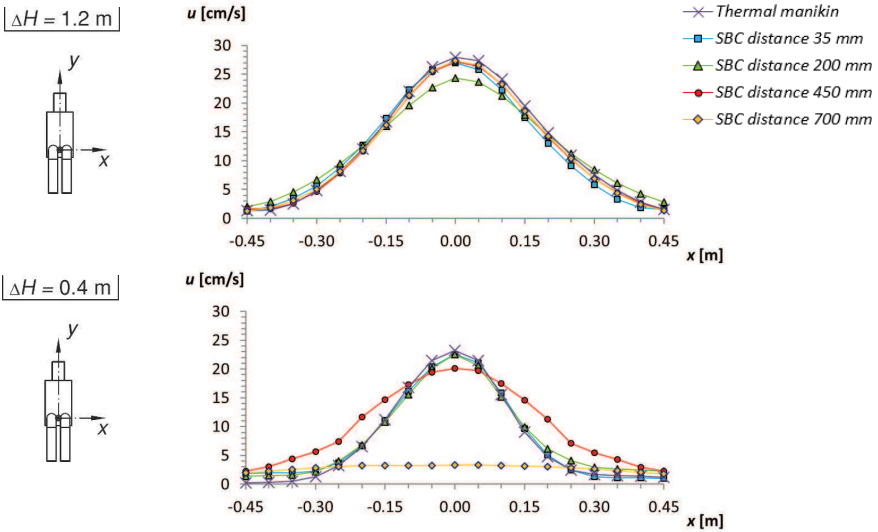

Fig. 6.5 – Velocity profiles in vertical plane x-y, at two different heights ΔH above the manikin

The velocity profiles determined for the simulations with simplified models are close to the reference case (*Thermal manikin*) especially at higher regions of the room, i.e. above the *SBCs,* see Fig. 6.5 (ΔH = 1.2 m). The exception is the case with the *SBC* positioned 200 mm above the manikin's head, where the comparison shows a lower velocity in the centre of the convective flow than in the reference simulation. The deficiency was caused by an instability of the thermal plume in the reference case, at the height where the *SBC* for the simplified simulation was determined.

There are bigger differences in the locations closer to the manikin, see Fig. 6.5 (ΔH = 0.4 m). It is possible to see that in the region under the *SBC* the results of the simplified simulation do not correspond to the reference case.

The temperature profiles in the simulations with simplified models show good correspondence to the reference case at the heights above the *SBCs,* see Fig. 6.6. The exception is the simulation with the *SBC* placed 200 mm above the head of the manikin. In this case, the thermal plume has slightly lower maximal temperature than in the reference case. The difference is approximately 0.2 K.

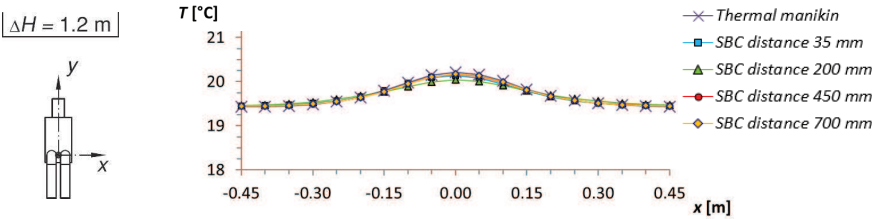

Fig. 6.6 – Temperature profiles in vertical plane x-y, 1.2 m above the manikin

It can be advised to position the *SBC* at a reasonable height 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). 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 on the basis of the reference detailed simulation.

In the regions below the *SBC*, the results with the simplified models did not correspond to the reference case. This issue was addressed by adding another *SBC* in a lower height, as described in the following chapter.

6.2 Multiple SBCs

The influence of the combination of two *SBCs* inducing the thermal flow was studied by Zelenský et al. (2012). Three cases with different configurations of the *SBCs* were compared, see Fig. 6.7.

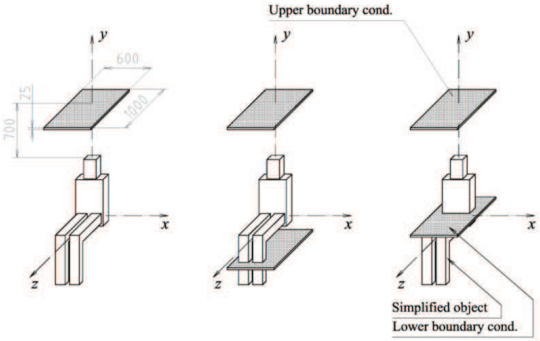

Fig. 6.7 – Substitution of the heat source by two SBCs (Zelenský et al. 2012)

The isolines of velocity magnitude in the vertical plane *z-y* (side view) intersecting the centre of the thermal manikin's head are displayed in Fig. 6.8. The simulations with the simplified heat source were compared to the simulation with the detailed model of thermal manikin, i.e. with the reference case (denoted as side view A).

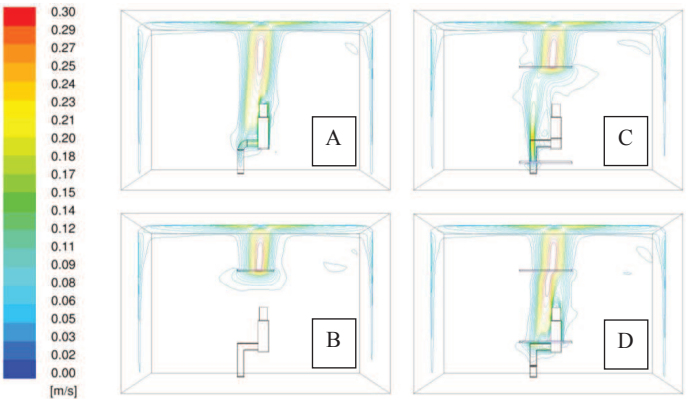

Fig. 6.8 – Velocity magnitude isolines of thermal plumes (Zelenský et al. 2012): A – detailed model; B to D – simplified models with various combinations of SBCs

The thermal plume patterns are close to each other in all the computational cases at the height above the upper *SBC*. The 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 the thermal plumes under the upper *SBCs* differs more significantly in the individual simulations. There is no convective flow formed around the manikin in the simulation with just one *SBC* (Fig. 6.8 – B). This deficiency was addressed by placing another *SBC* to the lower height above the floor. The convective current generated by the lower *SBC* flows around the object substituting the thermal manikin and enters the upper *SBC*, which generates the plume rising towards the ceiling.

The flow in the case with the second *SBC* positioned at the level of waist (Fig. 6.8 – D) is the closest to the reference case. The generated thermal plume rises above the thighs and appropriately adheres to the manikin. The flow above thighs significantly influences the rising thermal plume. The imperfection in this case is caused by the insufficient flow along the back of the manikin, which would, after merging with the flow above the legs, bend the resulting thermal plume more behind the manikin.

Velocity and temperature profiles

Velocity and temperature profiles were determined for every computational case at several heights above the manikin, see for example Fig. 6.9 (velocity profiles). The velocity profiles are corresponding to each other especially at the higher regions of the room, see ΔH = 1.2 m. There are apparent differences between individual simulations at lower heights (under the upper *SBC*). The simulation with two *SBCs*, where the lower *SBC* is positioned at the waist level, is the one 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.

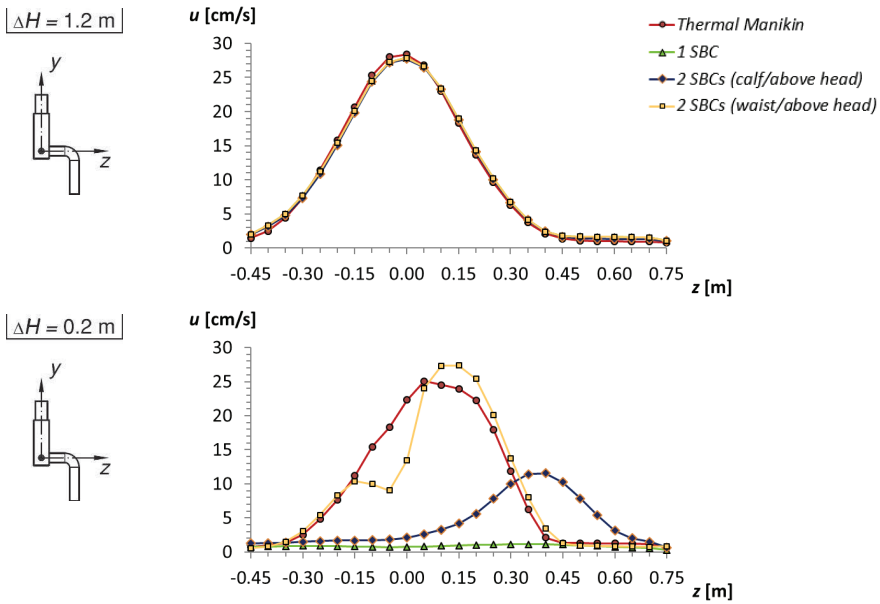

Fig. 6.9 – Velocity profiles in vertical plane y-z, at two different heights ΔH above the manikin

6.3 Merging of thermal plumes

Simulations with multiple heat sources should reflect the merging of individual thermal plumes in higher regions. There was a concern about how the developed method influences this physical process. A numerical study was elaborated upon with the focus on this issue (Zelenský et al. 2016). It compares a set of simulations with numerical models of four sitting thermal manikins placed inside an enclosed room, see Fig. 6.10.

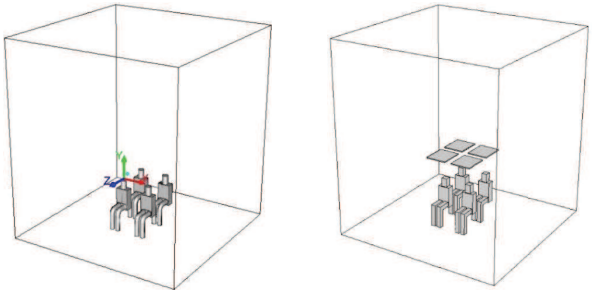

Fig. 6.10 – Simulation setup with detailed (left) and simplified (right) models

Fig. 6.11 presents the isolines of velocity magnitude in the room in the vertical planes *x-y* (front view) and *z-y* (side view). The thermal plumes patterns are resembling each other in both computational cases at the region above the SBCs, especially close to the ceiling. Merging of the four thermal plumes is obvious in both cases, although it is more noticeable above the detailed models due to the lower merging height in this reference simulation.

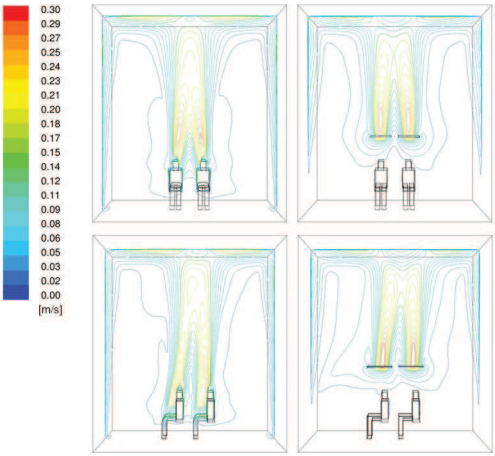

Fig. 6.11 – Velocity magnitude isolines, detailed (left) and simplified (right) models (top – front view; bottom – side view)

In the side view, it is possible to see that the thermal plume above the front seated manikin rises in a slightly different way when comparing the simulations with detailed and simplified models. In the case with the detailed models, it is deflected more from the vertical direction and rises more towards the rear manikin than in the case with the simplified models. However, in the higher region, the thermal plumes merge completely in both simulations, adhere to the ceiling and spread further to the vertical walls in a very similar way.

The formation of thermal plumes under the *SBC* differs significantly in the two cases. There is no convective flow formed around the manikins in the simulation with the simplified models. This deficiency could be solved by placing another *SBC* to the lower height above the floor of the room. However, the deficiency was neglected in this study as it was targeting the merging of thermal plumes above the heat sources.

Velocity and temperature profiles

The velocity and non-dimensional temperature profiles of the rising thermal plume for the heights 1.0 m and 3.6 m above the thermal manikins are displayed in Fig. 6.12 and Fig. 6.13, respectively.

There are obvious differences in the velocity profiles determined for each simulation at the height 1.0 m, see Fig. 6.12 (ΔH = 1.0 m). The reference case with the detailed models of thermal manikins provides narrower flow with lower maximal velocity at the corresponding height and individual thermal plumes adhering more to each other. It is possible to see that the thermal plume above the front manikin is deflected from the vertical direction towards the rear manikin. In the case of simplified models, the thermal plumes remain less influenced by each other.

The higher above the manikins, the more the thermal plumes resemble each other in the two simulated cases, see Fig. 6.12 (ΔH = 3.6 m). Although the plumes in the simulation with the simplified models are not fully merged yet, they are very close to the plumes in the simulation with the reference models. The absolute velocities, in both cases, are comparable and it can be expected that the full merging in the simplified case would be achieved if the space was higher. This deficiency could be solved by placing the prescribed *SBC* lower above the thermal manikins' heads, so the merging would be achieved sooner in the case with the simplified models too. However, depending on the problem on hand, this may not be necessary (for example, when the primary interest is to simulate air flow in the higher regions above the floor).

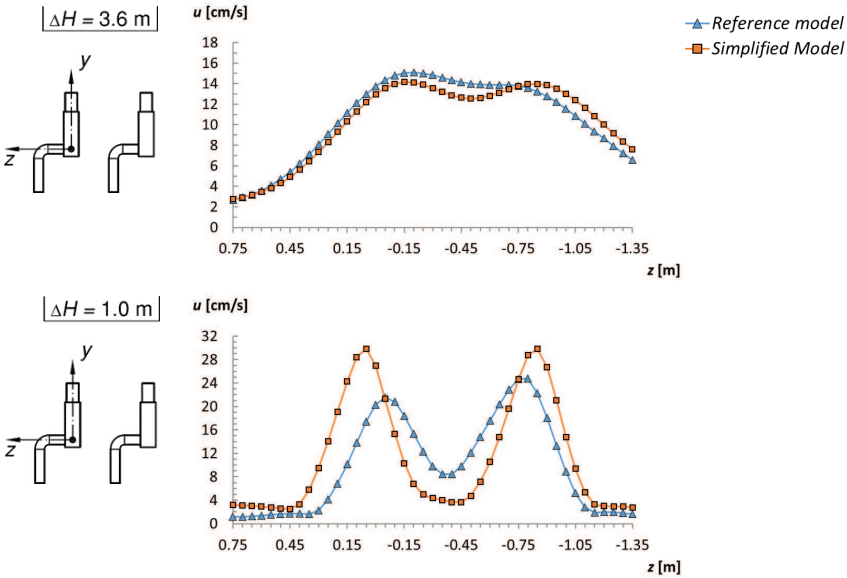

Fig. 6.12 – Velocity profiles in the vertical plane y-z, at two different heights ΔH above the manikins

The non-dimensional temperature profiles are influenced by the simplification of the heat sources in a way similar to velocity profiles, see Fig. 6.13. Slower merging of the individual plumes above the heat sources in the simulation with the simplified models causes deficiency of the results. Although in the higher regions the thermal plumes from reference and simplified simulation resemble each other closely, it is possible to see that the merging of the plumes in the simulation with the simplified models is not completed yet. The maximum temperature difference between the two profiles at the corresponding height was 0.3 K. The temperature in the case with the simplified models was lower in almost all the points and it was more varying than in the reference simulation.

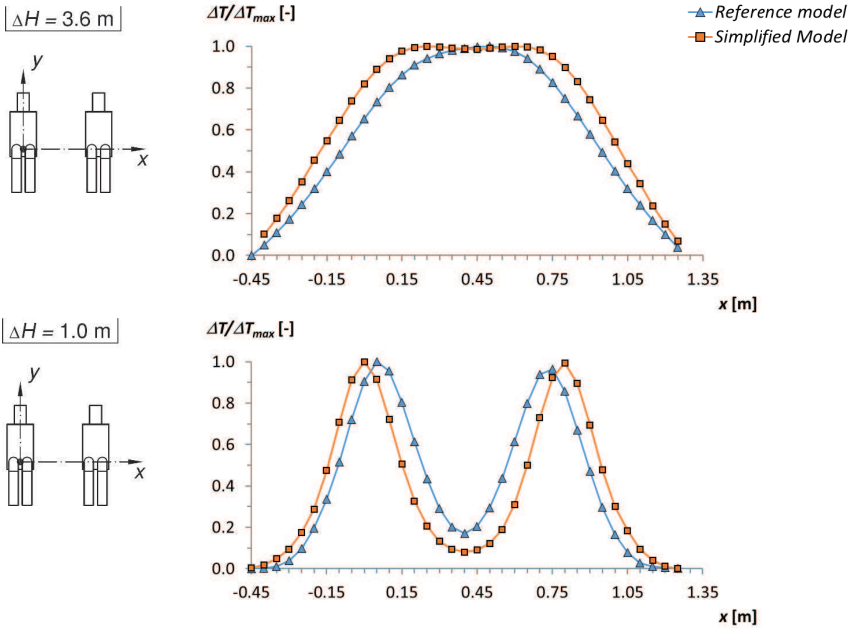

Fig. 6.13 – Non-dimensional air temperature profiles in vertical plane x-y, at two different heights ΔH above the manikin

Momentum of the thermal plumes

Momentum transfer rate of the raising thermal plumes was evaluated and compared for both simulations in order to assess the influence of the simplification. The momentum flux was calculated as an integral over a rectangular region with dimensions 2.2 x 2.5 m, intersecting the raising thermal plumes at the height 2.4 m above the heads of the thermal manikins. Custom field function programmed in ANSYS Fluent was used to enable the calculation following the formula (6.1), where $\dot{I}$ is the momentum transfer rate, u_y is vertical velocity of the air flow and ρ is air density.

$$\dot{I} = \int_A u_y^2 \rho \cdot dA \tag{6.1}$$

The momentum transfer rate of the raising thermal plume was 0.057 kg·m/s^2 in the simulation with the reference models and 0.052 kg·m/s^2 in the case with the simplified models, which is lower by 9 %. This indicates that the proposed simplification of heat sources can slightly underestimate the momentum of the rising thermal plumes.

6.4 Computational demands

Four computational cases with sitting thermal manikin placed in the middle of an enclosed room were simulated in order to show the effect of the simplification on the speed of simulations. The thermal manikin was modelled in detail in the first case and in the other three cases, it was simplified according to the proposed method. The simplification setups were identical to the ones in Chapter 6.2.

Tab. 6.2. summarizes the total number of computational cells in all the compared cases and computational time necessary for 10 iterations. All the simulations were calculated at the same computer, 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. From the summarized values, it is possible to see the advantage of substituting the 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 the detailed model of the thermal manikin.

Tab. 6.2 – Comparison of mesh cell number and computing time

Computational case	Mesh cells	Computational time	
Thermal manikin	3 569 984	130 s	100 %
1 *SBC*	1 469 837	45 s	35 %
2 *SBCs* (calf/above head)	1 548 459	51 s	40 %
2 *SBCs* (waist/above head)	1 538 362	51 s	40 %

6.5 Development of UDF for practical use

The usability of the proposed method for more complex cases with higher number of heat sources can be enhanced by using the User Defined Function (UDF) tool which is available in ANSYS Fluent. The velocity and turbulent quantities of the induced flow can be prescribed in absolute values. The temperature of the convective flow may be prescribed as relative to the reference ambient temperature. The workflow of the programmed UDF which was used by Zelensky et al. (2015) in the performed study targeting air flow simulation in a large cultural space is described below.

1. Set the local coordinate system with the origin in the geometrical centre of the *SBC*;
2. get the reference temperature in the vicinity of the *SBC*;
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 conditions for the turbulent quantities k and ε.

7 CASE STUDY – CONCERT HALL HOUSED IN A FORMER CHURCH

The previously developed modelling approach to represent heat sources in CFD simulations was applied in a real scenario of the former church of St. Anna, a 14th century gothic building located in the Old Town of Prague (hereinafter referred to as *the church)*. The building was desecrated, reconstructed and it now serves as a universal cultural space with the maximum capacity 350 visitors.

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 of *the church*. The biggest concerns were indoor air flow velocities and air temperature distribution in the vicinity of the internal wall surfaces (Barták et al. 2001). The design team had to assess the possible negative effects of the changes in the indoor environment of *the church* on the original stucco decorations on its walls and the original wooden roof trusses.

A CFD study with simplified models of visitors acting as heat sources under two different occupancy scenarios (65 and 304 visitors) was elaborated (Zelenský et al. 2015). The models of the visitors were represented according to the previously developed method. The results of the simulations with 65 and 304 visitors were compared mutually and also with the simulation without the explicitly modelled visitors (Barták et al. 2001). The effect of the heat sources on the indoor air flow, temperature stratification and ventilation rates were studied. The study also demonstrates the usability of the previously developed modelling method to represent indoor heat sources in complex real situations.

7.1 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 western half of the nave is divided by a gallery located 6 m above the ground floor, see Fig. 7.1. *The church* has three large window openings on the street level and eleven small window openings in the roof. Only natural ventilation through these openings is possible to use. The model include a 1.5 m wide region of external space surrounding *the church*, in order to simulate the process of natural ventilation. Two numerical models of *the church* with a different number of seated visitors were created (65 and 304 visitors). The building interior and the external space surrounding the building was divided by an unstructured tetrahedral grid. The minimum size of a cell was 25 mm and the maximum was 250 mm. A fine mesh was created in the space of the auditorium and near the walls.

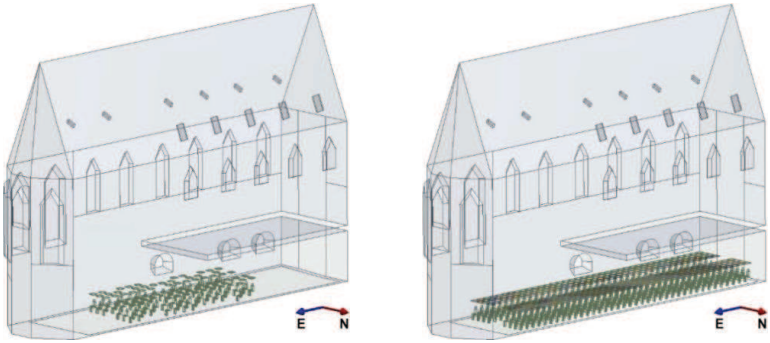

Fig. 7.1 – Models of the church, 65 visitors (left) and 304 seated visitors (right)

The boundary conditions of the surfaces facing the surrounding environment were specified as a free boundary with zero pressure gradient. The composition of the building constructions was prescribed in the model and the temperature of the internal surfaces were calculated by the solver, i.e. the thermal conduction through walls was taken into account. The external air temperature was set to –7 °C, as winter conditions were considered. From two sides, *the church* is partially surrounded by other occupied buildings, with the indoor temperature 20 °C. The temperature of the ground under the floor was considered 5 °C.

The CFD simulations were solved as a non-isothermal flow of incompressible ideal gas (air). The flow in the proximity of the walls was solved using wall functions. The two-equation turbulence model by Launder and Spalding (1974) – so called *k-ε Standard* – was used, extended with respect to the influence of temperature and buoyancy on the turbulence. The Body Force Weighted scheme was chosen for the discretisation of the pressure term. The convective terms were solved using a second order upwind scheme. A coupled and steady-state solver was used.

7.2 Results analysis and discussion

The velocity vectors in the vertical plane *y-z* intersecting the centre of *the church* (side view) for the case with no explicitly modelled heat sources from the simulation performed during the study of Barták et al. (2001) are displayed in Fig. 7.2; Fig. 7.3 shows the velocity vectors and isolines of the velocity magnitude of two simulated cases with the models of visitors represented using the previously developed method.

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 (Fig. 7.2) there are two large vortices above the raised gallery in the western part of *the church* and one large vortex in the eastern part of the building. However, in both cases with the models of visitors (Fig. 7.3), the vortices 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 visitors and rises from the space under the gallery. Also, it is possible to see that in the case with 304 visitors there is a stronger air circulation in the space of *the church* than in the case with 65 visitors, although the flow trajectories resemble each other.

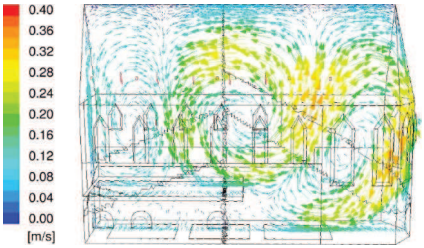

Fig. 7.2 – Velocity field in the case without explicitly modelled heat sources (velocity vectors) (Barták et al. 2001)

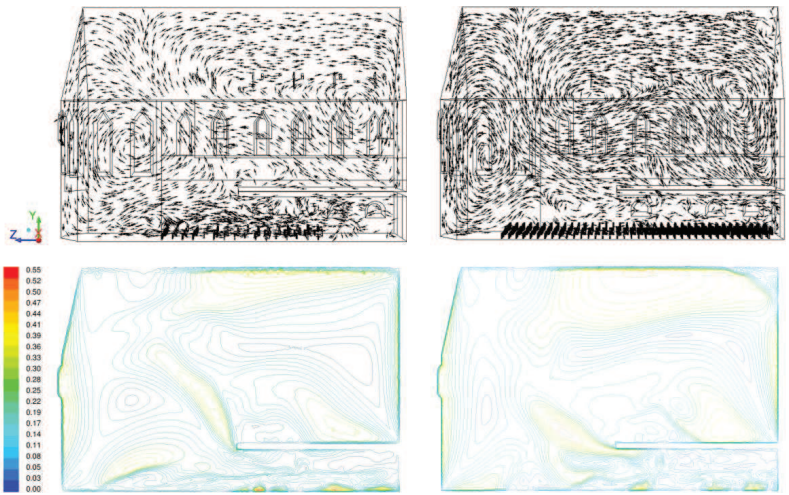

Fig. 7.3 – Velocity fields in the case with 65 visitors (left) and 304 visitors (right) upper: velocity vectors; lower: velocity magnitude isolines

The temperature distribution in *the church* was evaluated on the basis of the temperature contours. The selected cross-sections for the cases with 65 and 304 visitors are displayed in Fig. 7.4.

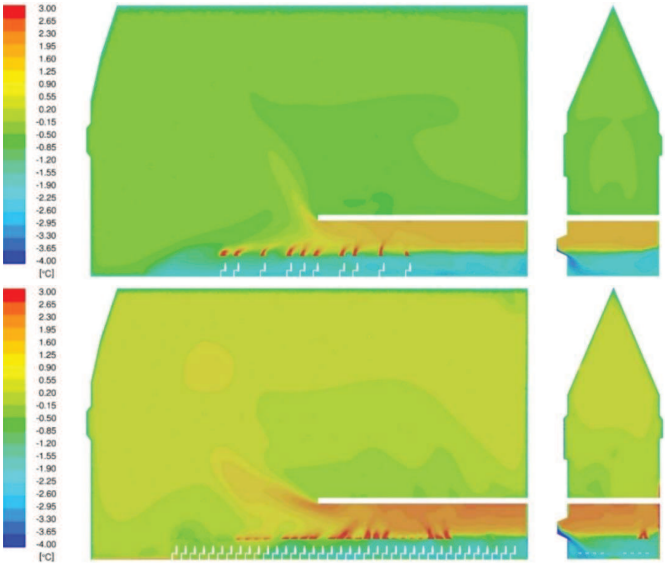

Fig. 7.4 – Temperature fields (isosurfaces of temperature); 65 (top) and 304 (bottom) visitors

The influence of the visitors acting as heat sources is obvious. In the case with 65 visitors the space below the gallery in the western part of the church is colder with more gradual stratification. In the case with more visitors the supplied air is heated up faster and the influence of the cold air flow 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 increase in the average temperature in the space was only 1.1 °C. Also, 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.

Ventilation flow rates

In both simulated cases, where the visitors were modelled using the developed method to represent heat sources, the ventilation flow rates have been evaluated, see Tab 7.1.

Tab. 7.1 – 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 per visitor is more than sufficient for this type of space in both simulated cases. The recommended value of the volume flow rates for the auditorium seating area is 2.7 L/s per person (ASHRAE 2003). However, according to Czech standards, the minimal recommended volume flow rate for occupied spaces is 5 L/s per person (ÚNMZ ČR 2010), so the case with 304 visitors would not comply with the recommendation. Although this standard is indicated for forced ventilation systems only, it should be taken into account.

7.3 Case study outcomes

It was shown that the presence of the visitors influences velocity patterns in the whole space of *the church* and the temperature fields, especially in the regions close to the floor. In the case with more visitors, the supplied air is heated up faster and the influence of the cold air flow from the low window openings is decreased. The higher regions of *the church* show a good thermal stability. The increase in the number of visitors from 65 to 304 caused an increase in the average temperature in the space of only 1.1 °C. This relatively small temperature change should not have a negative effect on the preserved internal constructions of the building. The main concern may be very low temperatures in the space of auditorium, caused by the cold air supply from the low window openings. A heating system should be used during winter days.

It has been shown that natural ventilation of the building in the winter period is a feasible option, as it provides enough fresh air. It can be 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. A partial blocking of the windows could reduce this fast cold air flow.

In addition, it has been shown that the previously developed modelling method to represent indoor heat sources in CFD simulations is suitable for this type of study. The proposed method enabled the simplification of both heat sources' geometry and computational mesh around them, but it preserved the rising thermal plumes patterns. Thus, the possibility of the variant CFD simulation study was enhanced and the obtained results were more realistic in comparison with the simulation without the explicitly modelled heat sources.

8 CONCLUSION

The main goal of the dissertation work was to identify the current methods of indoor heat sources modelling, ascertain their limitations and propose a new modelling method to represent indoor heat sources in numerical models for CFD simulations. It was targeting better design of large indoor spaces with high number of heat sources and/or variable occupancy patterns. The aim was to enhance the effective use of CFD simulation for practical applications in HVAC Engineering.

8.1 Theoretical contributions

The main theoretical contribution of the current research is the new modelling method to represent heat sources for numerical studies of indoor air flow by CFD simulations. It is based on the replacement of a heat source by one or more appropriately positioned simplified boundary conditions (in this study referred to as *SBCs*) inducing a thermal plume potentially identical to the one rising above the real heat source. It enables simplification of the heat source model geometry and reduces the requirements on the computational mesh around it, while it preserves the thermal plume pattern above the heat source in the simulation. Although in the current research the method was applied to models of occupants, it can be in general used to represent any other indoor heat source.

The method was validated, tested and described in detail for further use or adaptation. In order to make the new modelling approach universal with respect to different ambient thermal conditions, the boundary condition inducing the thermal plume was derived using a non-dimensional temperature profile.

Aside of the new modelling method to represent heat sources, the following two contributions for theory can be mentioned.

Thermal plumes development in different ambient air temperatures

The current research brings a better understanding of the behaviour of thermal plumes rising above heat sources under different ambient air temperature conditions. The issue was initially studied on

the basis of a semi-empirical formulae available in the literature. However, the influence of the ambient air temperature is not fully described in the available publications. Therefore, an additional numerical study targeting thermal plume development around a thermal manikin under different temperature conditions was conducted.

It was demonstrated that the velocity profile of the thermal plume does not depend on the ambient air temperature, providing that the heat output of the heat source is constant. On the other hand, the ambient air temperature influences the absolute values of air temperature in the thermal plume, while non-dimensional temperature profile of the thermal plume is not significantly influenced.

Modelling of turbulence in CFD simulations of thermal plumes generated by people indoors

A numerical study was conducted targeting the performance of the k-ε and k-ω two-equation turbulence models that are the most frequently used for CFD simulations in the field of indoor air flow. It was shown that the choice of a particular turbulence model affects the turbulent mixing and consequently the spreading rate of the thermal plume and its velocity and temperature profiles.

The turbulence models of k-ε type have been found appropriate for the simulation of thermal plumes caused by natural convection. The k-ε *Standard* turbulence model, proposed by Launder and Spalding (1974), produced results which showed the best agreement with the experiment. The use of this turbulent model was recommended for CFD simulations of thermal plumes above heat sources.

8.2 Practical contributions

The proposed method to represent heat sources in CFD simulations of indoor air flow enables faster preparation of numerical models, easier and faster meshing, it decreases the number of computational cells in the domain and consequently reduces the computational time. It facilitates simulations of large spaces with large numbers of heat sources and/or variable occupancy patterns, such as various entertainment facilities, industrial premises, lecture halls, etc., which would otherwise be very challenging. The possibility of numerical variant studies of such places is enhanced as well. It is possible to easier prepare a numerical model, solve the simulation faster and reach relevant results more rapidly. Thus, the presented method enables CFD simulations to be more effective part of practical design process.

A user defined function (UDF) for ANSYS Fluent inducing the artificial thermal plume above a simplified model of heat source was proposed in order to facilitate the use of the developed method for practical applications.

8.3 Suggestion for future work

The goals of the dissertation thesis have been met; nevertheless, there are issues that could be further investigated.

- The presented method could be adapted in order to simulate emission of contaminants and water vapour by the source (as heat sources often release also other substances). The *SBC* can be extended by a boundary condition of contaminant and/or water vapour emission.
- In order to make simulations of large groups of heat sources more accurate, the interaction of thermal plumes rising above individual heat sources could be taken into account already during the preparation of the simplified models, following the proposed method. A set of heat source models with different *SBCs* could be created, considering the heat source position in the group (front row, back row, left row, right row and middle of the group).
- To enhance the use of the developed method itself, a database of simplified models of heat sources that are the most commonly present in indoor environments could be created.

REFERENCES

Author's references directly related to the current research

Zelenský, P., Barták, M., Hensen, J. L. M. (2012). Model sedící osoby jako zdroje tepla ve vnitřním prostředí. *Vytápění Větrání Instalace*, *5*, 22–26. [in Czech]

Zelenský, P., Barták, M., Hensen, J. L. M. (2013). Simplified Representation of Indoor Heat Sources in CFD Simulations. In *Proceeding of the 13th International Conference of the International Building Performance Simulation Association* (pp. 3-29). Chambéry, France.

Zelenský, P., Barták, M., Hensen, J. L. M. (2013). Faktory ovlivňující CFD simulaci konvekčního proudu nad zdrojem tepla ve vnitřním prostředí. *Vytápění, Větrání, Instalace*, *22*(5), 214–220. [in Czech]

Zelenský, P., Barták, M., Hensen, J. L. M., Vavřička, R. (2013). Influence of Turbulence Model on Thermal Plume in Indoor Air Flow Simulation. In *Proceeding of the 11th REHVA World Congress „Energy Efficient, Smart and Healthy Buildings" – Clima 2013* (pp. 6755-6764). Prague, Czech Rep.

Zelenský, P., Barták, M., Hensen, J. L. M. (2015). Simulation of Indoor Environment in the Concert Hall housed in a Converted Former Church. In *Proceeding of the 14th International Conference of the International Building Performance Simulation Association* (pp. 1716-1721). Hyderabad, India.

Zelenský, P., Barták, M., Hensen, J. L. M. (2016). Influence of Thermal Plumes Interaction on CFD Simulation of Indoor Heat Sources. In *Sborník 9. národní konference s mezinárodní účastí Simulace budov a techniky prostředí* (pp. 127–132). Brno, Czech Republic.

Zelenský, P., Barták, M., Hensen, J. L. M. (2017). Simulation of Indoor Environment in the Concert Hall Housed in a Former Church. *Vytápění, Větrání, Instalace*, *26*(6), 342–348.

The other significant references

ANSYS Inc. (2013). *ANSYS Fluent User's Guide*. USA: ANSYS Inc.

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. 563 p.

Barták, M., Drkal, F., Lain, M., Schwarzer, J. (2001). *Obrazy proudění v kostele Sv. Anny v Praze 1* [Research report no. 01002]. CTU in Prague, Department of Environmental Engineering. [in Czech]

Bradshaw, P., Huang, G. P. (1995). The Law of the Wall in Turbulent Flow. *In Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 451 (1941), 165–188.

Deevy, M., Sinai, Y., Everitt, P., Voigt, L., Gobeau, N. (2008). Modelling the effect of an occupant on displacement ventilation with computational fluid dynamics. *Energy and Buildings*, *40*(3), 255–264.

Djunaedy, E., Hensen, J. L. M., Loomans, M. G. L. C. (2003). Towards External Coupling of Building Energy and Airflow Modeling Programs. *ASHRAE Transactions*, *109*(2), 771–787.

Hensen, J. L. M. (2004). Integrated building airflow simulation. *Advanced building simulation*, 87–118.

Hyldgaard, C. E. (1998). Thermal plume above a person. In *Proceeding of the 6th International Conference on Air Distribution in Roooms – Roomvent 1998* (pp. 407–413). Stockholm, Sweden.

Jančík, L. & Bašta, J. (2011). Termovizní vizualizace teplotního pole neizotermního vzdušného proudu. *TZB Haustechnik, 1,* 11–14. [in Czech]

Koiš, G. (2009). *Analýza proudů v klimatizovaném prostoru* [MSc. Thesis]. CTU in Prague, Department of Environmental Engineering. Prague. [in Czech]

Launder, B. E. & Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Energy*, (3), 269–289.

Matuška, T. (2005). *Experimentální metody v technice prostředí* (1st ed.). Prague: Czech Technical University in Prague. [in Czech]

Nielsen, P. V. (1997). *The Box Method: a practical procedure for introduction of an air terminal device in CFD calculation*. Aalborg University, Department of Building Technology and Structural Engineering, Aalborg, Denmark.

Nielsen, P. V., Allard, F., Awbi, H. B., Davidson, L., Schälin, A. (2007). *Computational Fluid Dynamics in Ventilation Design - REHVA Guidebook No. 10*. Finland: Forssa. 104 p.

Patankar, S. V. (1980). *Numerical Heat Transfer and Fluid Flow*. Washington: Hemisphere Publishing Corp.

Popiolek, Z. (1987). *Testing and modelling of buoyant convective flows in consideration of the formation of ventilation process* [D.Sc. Thesis]. Silesian University of Technology. Gliwice, Poland.

Shih, T., Liou, W., Shabbir, A., Yang, Z., Zhu, J. (1995). A new eddy viscosity model for high reynolds number turbulent flows. *Computers & Fluids*, *24*, 227–238.

Skistad, H., Mundt, E., Nielsen, P. V., Hagstrom, K., Railio, J. (2002). *Displacement ventilation in non-industrial premises. REHVA Guidebook No. 1.* Trondhaim: Tapir. 95 p.

Srebric, J. & Chen, Q. (2002). Simplified Numerical Models for Complex Air Supply Diffusers. *HVAC&R Research*, *8*(3), 277–294.

ÚNMZ ČR. (2010). *Větrání nebytových budov - Základní požadavky na větrací a klimatizační systémy - Standard ČSN EN 15 665/Z1*. [in Czech]

Wilcox, D.C., 1988. Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, 26, 1299–1310.

Yakhot, V. & Orszag, S. A. (1986). Renormalization group analysis of turbulence. *Journal of Scientific Computing*, *1*, 3–51.

Zbořil, V., Melikov, A., Yordanova, B., Bozhkov, L., Kosonen, R. (2007). Airflow Distribution in Rooms with Active Chilled Beams. In *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007* (pp. 1–7). Helsinki, Finland.

Zhai, Z. J., Zhang, Z., Zhang, W., Chen, Q. Y. (2007). Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1 – Summary of Prevalent Turbulence Models. *HVAC&R Research*, *13*(6), 853–870.

Zhang, T., Lee, K., Chen, Q. (2009). A simplified approach to describe complex diffusers in displacement ventilation for CFD simulations. *Indoor Air*, *19*, 255–267.

Zukowska, D. (2011). *Airflow interactions in rooms - Convective plumes generated by occupants* [Ph.D. Thesis]. Technical University of Denmark, Department of Civil Engineering. Lyngby.

Zukowska, D., Melikov, A., Popiolek, Z. (2007). Thermal plume above a simulated sitting person with different complexity of body geometry. In *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007* (pp. 191–198). Helsinki, Finland.

Zukowska, D., Melikov, A., Popiolek, Z. (2008). Impact of Thermal Plumes Generated by Occupant Simulators with Different Complexity of Body Geometry on Airflow Pattern in Rooms. In *Proceeding of the 7th Internation Thermal Manikin and Modelling Meeting - University of Coimbra* (pp. 3–6). Coimbra, Portugal.

Zukowska, D., Melikov, A., Popiolek, Z. (2010a). 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.

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