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

Ing. Petr Zelenský

Supervisors: prof. dr. ir. Jan L. M. Hensen, Ing. Martin Barták, Ph.D.

## Introduction

The main aim of the current research is to target issues of improper air distribution risk in large indoor spaces with a high number of heat sources and/or variable occupancy patterns, such as various entertainment facilities, industrial premises, lecture halls, atriums, etc. The achievement of high indoor environment quality with simultaneous consideration of energy demands in such places can be challenging. One of the tools that can help to analyze and understand the complex interactions in the indoor environment is computational fluid dynamics (CFD) modelling and simulation. However, there are currently some limits of CFD, arising especially from the limited capacity of available computers and making simulation of large spaces with large number of heat sources difficult.

A new modelling method to represent models of heat sources for indoor air flow studies was developed in order to reduce computational burden of CFD simulations and ensure high reliability of the obtained results.

The method is targeting simulations of air flow in the space, evaluation of temperature conditions and ventilation efficiency in an indoor environment. It reflects the issues which are not addressed by the basic ways of simplification currently used for numerical modelling of indoor heat sources (simplification of the shape and geometrical details, boundary conditions and/or operational characteristics).

## New modelling method to represent heat sources

The proposed modelling method is based on the replacement of a heat source by one or more appropriately positioned simplified boundary conditions (*SBCs*) inducing a thermal plume potentially identical to the one rising above the real heat source, see for example Fig. 1. Each of the *SBSs* substituting the heat source prescribes velocity, temperature and turbulence of the induced flow. These characteristics are determined in advance on the basis of initial reference CFD simulation with detailed representation of the heat source. The work-flow of the method is described in the table below.

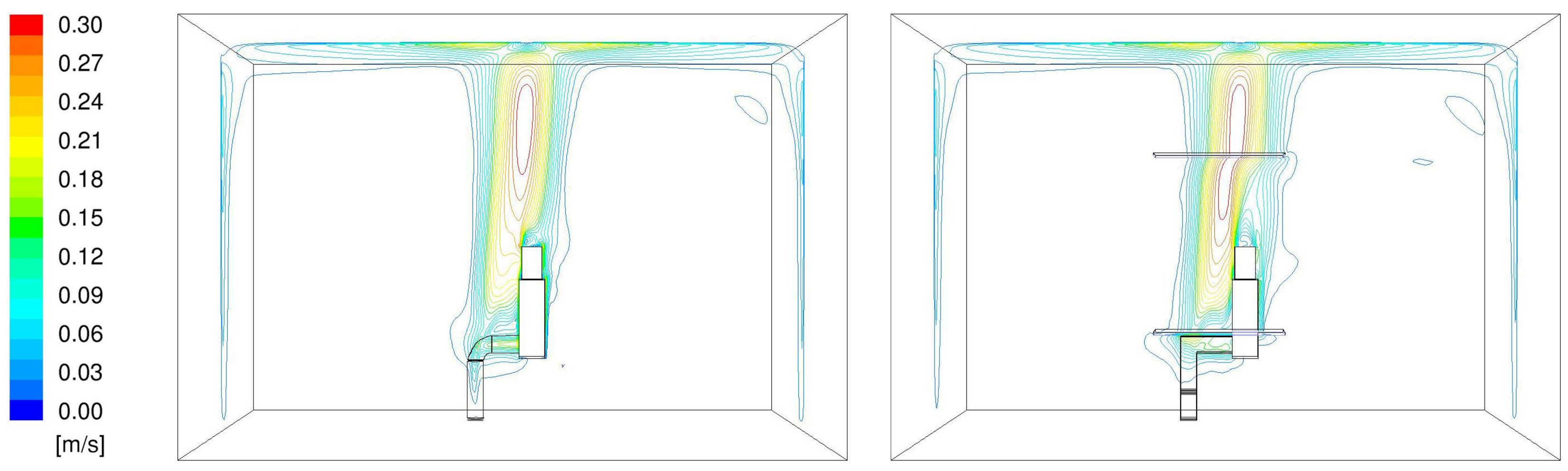


Fig. 1 - Thermal plume around a detailed and substitute model of heat source

| STEP | WORK-FLOW OF THE DEVELOPED MODELLING METHOD |
|---|---|
| 1 | **Preparation of the numerical model for reference simulation**<br>- detailed representation of a heat source |
| 2 | **Reference CFD simulation with a detailed model of the heat source:**<br>- the best practice of CFD simulations should be followed; heat and momentum transfer near the heated surface is modelled with integration of governing equations in the viscous sublayer<br>- 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):**<br>- 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 |
| 4 | **Creation of a numerical model with simplified representation of heat source(s):**<br>- the heat source body is substituted by a simplified geometrical object with adiabatic surface, acting solely as an obstacle to the ambient flow (no heat transfer from the object)<br>- 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):**<br>- one simulation with multiple simplified heat sources (i.e. large number of models created in the Step 4)<br>- several simulations with repeatedly used simplified model (i.e. variant numerical studies using the model created in the Step 4) |

## Experimental validation of the CFD simulation results

The proposition of the new modelling method was done in two steps. At first, a numerical simulation with detailed model of a sample heat source and a very fine numerical mesh was conducted. The results were empirically validated by comparison with experiment, using a physical model of a heat source identical to the one in the reference CFD simulation (i.e. thermal manikin).

A sheet of plain paper was stretched vertically above the centre of the manikin's head, see Fig. 2, and the temperature field was continuously transferred from the thermal plume to the paper by convection. 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 time-averaged horizontal temperature profiles were evaluated at 7 different heights above the manikin's head in two planes (front view, side view).

Fig. 2 – Experiment with thermal manikin

## Testing of the modelling method

Following the initial experimental studies, the new method to represent heat sources was tested in order to uncover its limitations. The testing was based on the comparison 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 comparing it with the experimental results. Several numerical studies were performed with following targets:

- to assess sensitivity of computations on the **selection of a turbulence model** and to 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 of heat source;
- to assess the **optimal distance of the *SBC* 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 the **influence of the simplification on the merging of multiple thermal plumes**.

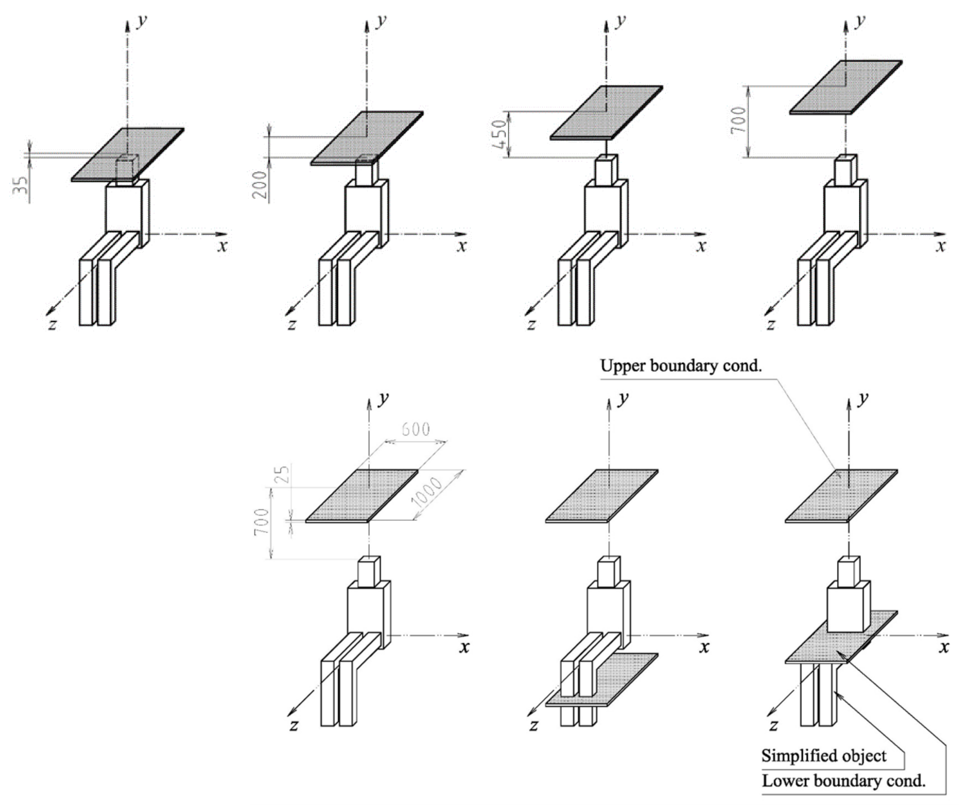


Fig. 3 – Various arrangements of the SBC(s)

## Reduction of 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 (see the following table). The thermal manikin was modelled in detail in the first case and simplified according to the proposed method in the other cases.

| 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 % |

## Case study

The proposed method 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. It was desecrated at the end of the 18th century, recently completely reconstructed and it now serves as a community centre and universal space suitable for events such as concerts, conferences, etc., with a total capacity of up to 350 visitors staying for different time periods during the day. A study based on the results of CFD simulations with simplified models of visitors acting as heat sources under two different occupancy scenarios (65 and 304 visitors) was elaborated in order to demonstrate the usability of the method for practical applications.

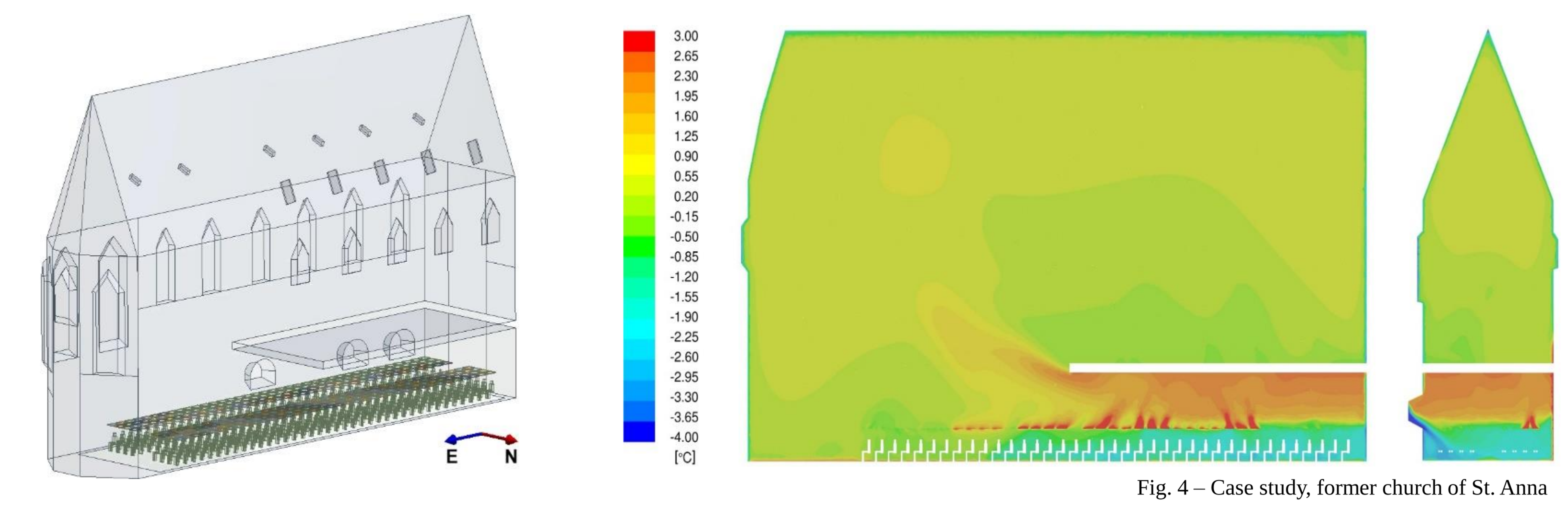

Fig. 4 – Case study, former church of St. Anna

## Conclusion

The key outcome is the new modelling method to represent heat sources for CFD studies of indoor air flow. 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 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. It can be used especially for CFD simulations of environments with high quantity of indoor heat sources and/or simulations of environments with high variability of heat sources' distribution, where it is problematic to model each heat source in detail. The possibility of numerical variant studies of such places is enhanced as well.

The proposed method was validated, tested and described in detail for further use or adaptation. In order to make the approach universal with respect to different ambient thermal conditions, the boundary condition inducing the thermal plume was derived using a non-dimensional temperature profile. A lower demand for computing time has been demonstrated. A user defined function (UDF) for ANSYS Fluent inducing the artificial thermal plume above a simplified model of heat source was proposed to facilitate the use of the developed method for practical applications.

**The presented method enables CFD simulations to be a more effective part of practical design processes.**

petr.zelensky@fs.cvut.cz | utp.fs.cvut.cz

FACULTY OF MECHANICAL ENGINEERING CTU IN PRAGUE

DOCTORAL THESIS – PETR ZELENSKÝ – 2018