A R C H I T E C T U R E C I V I L E N G I N E E R I N G

The Silesian University of Technology



d o i : 10.21307/ACEE-2018-063

FNVIRONMENT

RESEARCH ON THERMAL CONDITIONS IN VENTILATED LARGE SPACE BUILDING

Agnieszka PALMOWSKA a*, Gabriel MICZKA b

^a PhD; Faculty of Energy and Environmental Engineering, The Silesian University of Technology, Konarskiego 18a, 44-100 Gliwice, Poland

*E-mail address: agnieszka.m.palmowska@gmail.com

^b MSc Eng.; Gabriel Miczka Przedsiębiorstwo, Jasna 31B, 44-112 Gliwice, Poland

Received: 24.05.2018; Revised: 24.07.2018; Accepted: 8.10.2018;

Abstract

The aim of the paper was to verify whether it is possible to predict thermal indoor environment in an industrial facility, characterized by large cubature and significant heat sources using a computational fluid dynamics (CFD) technique. For this purpose, thermal imaging measurements were carried out to obtain boundary conditions, followed by numerical simulations of the airflow using the Ansys CFX software. Direct results of temperature and velocity distribution of air were analyzed for two different cases. The possibility of reducing the supply airflow rate without affecting either technology safety or thermal comfort was examined. The results of calculations can be helpful for improving the energy efficiency of the tested object by indicating the optimal operating modes for given operating conditions of the production hall.

Keywords: CFD; Large space building; Thermal conditions; Thermovision; Ventilation.

1. INTRODUCTION

Recently, large space buildings have become very popular in the design of modern buildings. They include not only civil buildings, such as atria, shopping malls, concert halls, sports centers, but also industrial buildings, such as various manufacturers, factory workshops, warehouses and production halls [1]. These are objects with a capacity of several to tens of thousands of cubic meters and with unusual height compared to standard buildings.

According to Annex 26 in the International Energy Agency (IEA) project, a large space building is defined as an enclosed ventilated airspace that is partly occupied and containing various contaminant and heat sources [2]. It is considered to be "large" due to the dominant effect thermal buoyancy has on air motion, the occupied zone is small compared to the total volume and airflow is dominated by temperature [3].

Building ventilation for large enclosures is a great

challenge in heating, ventilation and air conditioning (HVAC) system design [4]. High energy consumption is a difficult issue. In large space buildings, most of it is dedicated to the HVAC system [1]. We have to deal with the high value of change rate of ventilation air, reaching even a few changes per hour and resulting from (1) product requirements (technological reasons), (2) working environment requirements and (3) building requirements [5]. Over-sizing of these amounts, usually caused by the lack of knowledge and guidance at the design stage, results in significant energy consumption. Thermal conditions in the occupied zone are also important. Many parameters determine comfort conditions, the most important of which are: air temperature, vertical temperature gradient, air speed, humidity and air purity.

Ventilation of large enclosures deserves careful attention by design engineers [3]. The designer should take into account the thermal needs of the building, and also should anticipate organized and accidental air movements. The difficulties in design of a ventilation system include: heat gains that accumulate upwards (at the ceiling level) of the hall, causing so-called air (heat) cushion, supply and deposition of cold air layers in the lower part of the hall and the amount, type and location of so-called point heat sources [6]. Moreover, usually only a small portion of the entire volume is occupied, therefore energy efficiency may be achieved by special strategies aimed at directing ventilation air and thermal conditioning to the occupied zone. Vertical air streams driven by temperature differences gather large momentum in tall buildings. Resultant cold down-drafts from vertical surfaces may have severe comfort implications. Large enclosures are often found in unique buildings where no previous experience exists. Therefore careful analysis of the ventilation design is advisable [3].

Large space building is characterized by complex geometry and complex phenomena of air, heat and moisture flow. Nowadays conducting research for such a facility is possible by the CFD technique, which is particularly applicable where measurements in the facility are difficult or impossible, and where traditional engineering methods do not work [7]. Annex 26 in the IEA project recommends CFD when studying large complex facilities [3].

However, the CFD technique has its limitations. Numerical simulation of the flow field in large enclosures presents many difficulties due to the large size of the enclosure and the complicated flow field. Furthermore, a high number of control volumes or flow cells is needed to represent the flow. This adds to the complexity of simulating phenomena such as turbulence and increases the risk of numerical errors [3]. In the case of large-scale objects, a compromise between the real size of the model in relation to the available computing capabilities of the servers and for the possibility of obtaining reliable results plays on important role. Thus, the geometry of the tested object has to be simplified.

Moreover, the CFD technique requires a good definition of the tested object and therefore acquiring boundary conditions, most often through measurement or calculation. In practice, it can be difficult to obtain such data, in particular in the case of industrial buildings where there are various types of technical equipment or complex technologies and the measurements cannot interfere with the manufacturing process. In this case, the use of thermovision is worth of attention.

Thermographic measurements are an effective and non-destructive diagnostic method. The main advan-

tage of this technique is the fact that measurements are made during normal operation of devices, without the need to disturb the technological process. Using a thermovision camera, an image of the temperature field of the tested object is obtained, with a resolution of up to 0.1°C. Thermographic diagnosis is widely used in facilities where the source of the problem manifests itself with the change of the temperature distribution on its surface, and therefore also in industrial or production plants [8].

So far, including thermovision in CFD research has been done for a large scale sports facility and it was shown in [9], while examples of general studies of large objects, experimental or numerical, in [4, 5, 10].

Finally, the use of CFD to study complex physical processes in the built environment requires model validation. Typical methods for validating CFD results are the following:

- the comparison of simulation results with reducedscale or full-scale experimental data: the most popular method, which generally involves graphical comparison of computational results and experimental data. If the CFD results mostly agree with the experimental data, the computational results are declared validate;
- benchmark solutions;
- analytical solutions.

In a given application, the results of computer calculations do not have to be verified by methods used in scientific works, because the purpose of calculations is not to achieve precise numerical results, but only to check whether the results indicate incorrect thermal conditions of ventilation systems operation and, consequently, reduce the likelihood of such conditions occurring in fact. Verification of the results can take place each time in the real object.

The presented analysis is a great importance for the current state of knowledge and can be also widely used in practice, for example, to reduce energy consumption in industrial facilities and other large buildings by the use of the CFD method. Only a few examples of similar research can be indicated in the literature as e.g. [11, 12].

In this paper, the basic numerical simulations of air and heat flow in a ventilated industrial large-scale facility were undertaken. Thermal imaging measurements were made to identify heat sources. Calculations were carried out for two cases. The possibility of reducing the supply airflow rate without affecting either technology safety or thermal comfort was examined.

2. OVERVIEW OF THE TESTED FACILI-TY AND ITS NUMERICAL MODEL

The test object, located in the Mazowieckie Voivodeship in Poland, was a processing hall (with a non-rectangular shape) with a floor area about 1550 m^2 and a maximum dimensions $74 \times 25 \text{ m}$ (length \times width). The average and maximum hall height are 6.5 m and 9 m, respectively. Its total volume is over 10000 m³. This hall is located inside the facility of the entire production plant, therefore part of the partitions are internal partitions. The sources of heat are various types of machines, devices, and technologies for food production.

The ventilation system consists of displacement diffusers placed under the ceiling at the height of about 4 m. It consists of 2 supply and exhaust systems (SYS1 and SYS2), each with a capacity of 30000 m³/h, equipped with water heaters, and coolers, heat recovery, dampers, filters, fans. It is a continuous air ventilation system i.e. system without the possibility of reducing the flow (no regulation of the fan rotation speed). The performance regulation is done through the use of regulating dampers. Air handling units are located on the roof of the building.

The object's geometry was prepared on the basis of as-built drawings and inspection of the object. The geometric model was prepared in the SpaceClaim software. It includes real dimensions of the facility and technical equipment such as machines, devices, technologies. It also contains a lighting system, people, walls, gates, and a supply and exhaust ventilation systems. Fig. 1 shows the numerical model of the tested facility with significant equipment elements, whose parts were enlarged in Fig. 2, detail C.



Due to the complicated geometry, diffusers were simplified to solids consisting of rectangular flat surfaces (Fig. 2, detail A), whose active field was equal to the maximum effective area of supply openings, in fact round, in order to maintain the air flow velocity of 1.5 m/s, recommended in this type of diffusers.

Exhausts were modeled as flat, round surfaces with real dimensions along with a drip tray underneath (Fig. 2, detail B). All technical equipment was simplified to rectangular blocks divided into smaller areas (Fig. 2, detail C), in order to be able to separate areas with different temperatures in boundary conditions. Lighting and people were modeled in a simplified manner as rectangular heat sources.



3. CALCULATION CASES AND THEIR BOUNDARY CONDITIONS

In this study, it was examined how the ventilation system works in design conditions and in conditions where amount of supply air was reduced. Therefore numerical simulations were carried out for two basic cases:

- a) case 1 variant with the design amount of supply air;
- b) case 2 variant with amount of supply air reduced by 40%.

For each case, the ventilation air mass flow rate and its temperature were taken into account in boundary conditions in air supply openings, and static pressure in exhaust openings. Temperature and emissivity were set on the surfaces of the technical devices. Heat gains from lighting and 4 people were related to the surface of their model. The surface temperature was set for the walls and gates, only for the roof, the heat transfer coefficient and the temperature of the outdoor air were included. Information on the temperature distribution on the surfaces of devices and walls surrounding the hall was taken from thermovision measurements with the use of FLIR E50 camera (Fig. 3). Outdoor, indoor and supply air temperature was measured. The emissivity of surfaces and heat

Table 1.						
Boundary	conditions	for th	ie heat	transfer	common	for case
1 2						

,	
Boundary condition	Value/range of values
Heat transfer coefficient of roof	0.6 W/m ² K
Outdoor air temperature	8°C
Surface temperature of technical equipment surfaces	28–90°C
Surface emissivity of technical equipment	0.95
Surface temperature of partitions and gates	26–32°C
Surface temperature of technological elements	35–145°C
Surface emissivity of technological elements	0.95
Lighting power	18 kW
People	1 kW

transfer coefficient of the roof were determined. Surfaces of technical equipment or technological elements for which the temperature was set at boundary Table 2.

Boundary conditions for ventilation openings for case 1, 2

Boundary condition	Value
Mass flow rate of supply air of SYS 1 (case 1)	9.83 kg/s
Mass flow rate of supply air of SYS 1 (case 2)	5.90 kg/s
Mass flow rate of supply air of SYS 2 (case 1)	9.09 kg/s
Mass flow rate of supply air of SYS 2 (case 2)	5.40 kg/s
Temperature of supply air (SYS 1,2) (case 1, 2)	23.5°C

conditions were shown in the Fig. 4. The boundary conditions were summarized in Table 1, 2.

4. NUMERICAL PROCEDURE

The numerical calculations were carried out using ANSYS CFX 18.2 code. The simulations have been performed for steady-state, three-dimensional and non-isothermal conditions. The CFD model used the Shear Stress Transport turbulence model. Thermal



Figure 3.

Example of a thermogram of a heat source in the tested facility (picture-in-picture)



Figure 4.

Model surfaces for which different temperature was set in boundary conditions and marked global XYZ coordinate system



A fragment of the cross-section through the discretization grid

radiation between walls and technical objects located inside the hall was fulfilled by Discrete Transfer Model. Discretization of model equations was solved by Finite Volume Method. The applied unstructured discretization grid (Fig. 5) was composed of 15827591 cells, mostly of tetrahedral elements, and 3288588 nodes. It included also boundary layers and local refinement around the outlet, inlet openings, and technological elements. The basic mesh element size was 38 cm. The total number of model surfaces was 1028543. The calculations were carried out using iteration method as long as the convergent solutions were obtained with the use of workstation PC. Its basic configuration was as follows:

- 1 CPU, 6-cores, Intel® Xeon® E5 family;
- NVIDIA Quadro graphics card;
- 128 GB of memory.

The calculation time was 3–4 days just for one numerical case.

5. RESULTS AND DISCUSSION

As a result of the numerical calculations, the temperature and air velocity distributions in the facility were obtained. The results of calculations were presented in graphical form (Fig. $6 \div 13$) and as averaged values of air parameters. For each case, the results were presented for different planes. The ZX horizontal planes were placed at 2 different heights above the floor (Fig. $6 \div 7$, $9 \div 10$); the vertical plane YZ, X = 10.3 m passed through the plane of diffusers (Fig. 12); the vertical plane YZ, X = 40.0 m passed approximately through the center of the hall along its short side (Fig. 13); the vertical plane XY, Z = 5.0 m, passed approximately through the center of the hall along the long side of the hall (Fig. 8, 11). In the occupied zone the height of 1.5 m was selected as a representative. An average height of the most important devices emitting heat was on this level.

Comparing the air velocity distribution at a height of 1.5 m, no significant differences were observed between cases 1 and 2 (Fig. 6), while at a height of 6.0 m, clear velocity profile around the diffusers can be seen in case 1 (Fig. 7). For the vertical plane (Fig. 8), the similarity of the velocity distribution for both cases as well as many "stillness zones" – areas where the air speed is extremely low (or actually zero), appeared.

Considering air temperature in the facility, its average value in the entire volume was 25.9°C and 26.9°C, respectively for case 1, 2. Clear differences in the distribution of this parameter can be seen in the upper parts of the object, i.e. at a height of 6.0 m (Fig. 10), where accumulated, warm air occurred. Air stratification also appeared. This phenomenon was shown in Fig. 11–13 too, to a greater extent for case 2.

The stratification of air did not affect the air temperature at a height of 1.5 m above the floor, i.e. in the occupied zone, where its value at most was in the NVIBONMEN



Figure 6.

The comparison of air velocity isosurfaces maps in the tested facility in the horizontal plane ZX, Y = 1.5 m a) case 1 b) case 2



Figure 7.

The comparison of air velocity isosurfaces in the tested facility in the horizontal plane ZX, Y = 6.0 m a) case 1 b) case 2



Figure 8.

The comparison of air velocity isosurfaces in the tested facility in the vertical plane XY, Z = 5.0 m a) case 1 b) case 2



Figure 9.

The comparison of air temperature isosurfaces maps in the tested facility in the horizontal plane ZX, Y = 1.5 m a) case 1 b) case 2

e

ENVIRONMENT



Figure 10.

The comparison of air temperature isosurfaces in the tested facility in the horizontal plane ZX, Y = 6.0 m a) case 1 b) case 2



Figure 11.

The comparison of air temperature isosurfaces in the tested facility in the vertical plane XY, Z = 5.0 m a) case 1 b) case 2



Figure 12. The comparison of air temperature isosurfaces in the tested facility in the vertical plane YZ, X = 10.3 m a) case 1 b) case 2



The comparison of air temperature isosurfaces in the tested facility in the vertical plane YZ, X = 40.0 m a) case 1 b) case 2

e

ENVIRONMEN.

range of $24.4^{\circ}C \div 25.8^{\circ}C$ (Fig. 9) and on average was $25.3^{\circ}C$ and $25.9^{\circ}C$ for cases 1 and 2, respectively. Air temperature value for case 1 has been confirmed by measuring this parameter during thermovision measurements.

6. CONCLUSIONS

- 1. By means of CFD, it was possible to take into account most of the phenomena associated with the flow of air and heat, in a ventilated industrial facility, despite its large size. The performed numerical calculations allowed for mapping of thermal conditions identified during thermovision measurements and flow phenomena related to the operation of the ventilation system.
- 2. Thermovision measurements have proven to be a good method for obtaining the boundary conditions without disturbing the technology process.
- 3. In both cases, a clear stratification of air temperature, characteristic for displacement ventilation, was observed.
- 4. There was no too high air velocity in the facility due to the laminar airflow from diffusers.
- 5. Reducing the amount of supply air by 40% resulted in an increase of air temperature in the entire facility on average by less than 1°C.
- 6. The highest value of air temperature, exceeding 30°C, was observed at a height above 6 m, in particular in the case 2, with a reduced amount of supply air.
- 7. In the occupied zone, at the height of 1.5 m, no significant differences in air temperature distribution for both cases were observed.
- 8. The thermal power in the supply air for both cases was different by 50 kW. This result indicates a significant potential for improving energy efficiency in the regulation of the ventilation system and thus cost savings.
- 9. In the research, it was shown that a reduced airflow could be a possible way to reduce the energy use at this facility without influence on technology safety or thermal comfort.

REFERENCES

- [1] Liang, C., Shao, X., & Li, X. (2017). Energy saving potential of heat removal using natural cooling water in the top zone of buildings with large interior spaces. *Building and Environment, 124*, 323–335.
- [2] Heiselberg, P., Murakami, S., & Roulet, C.-A. (1998). Ventilation of Large Spaces in Buildings: Analysis and Prediction Techniques. Energy Conservation in Buildings and Community Systems, IEA Annex 26: Energy Efficient Ventilation of Large Enclosures, Denmark.
- [3] Moser, A. (1998). Technical Synthesis Report, IEA Annex 26: Energy Efficient Ventilation of Large Enclosures.
- [4] Wang, H., Huang, C., Cui, Y., & Zhang, Y. (2018). Experimental study on the characteristics of secondary airflow device in a large enclosed space building. *Energy & Buildings*, 166, 347–357.
- [5] Rohdin, P., & Moshfegh, B. (2007). Numerical predictions of indoor climate in large industrial premises. A comparison between different k- models supported by field measurements. *Building and Environment*, 42, 3872–3882.
- [6] Tomczak, I. (2013). Wentylacja i ogrzewanie hal magazynowych i produkcyjnych (Ventilation and heating of storage and production halls). *Chłodnictwo i Klimatyzacja, 11*, 52–55.
- [7] Lipska, B., Palmowska, A., Ciuman, P., & Koper, P. (2015). Modelowanie numeryczne CFD w badaniach i projektowaniu rozdziału powietrza w pomieszczeniach wentylowanych (Numerical modelling CFD in the research and design of air distribution in ventilated rooms). *INSTAL*, *3*, 33–43.
- [8] Miczka, G. (2003). Diagnostyka termograficzna. Przegląd zastosowań. (Thermographic diagnosis. Application overview) Wydanie 1/03, Gabriel Miczka Przedsiębiorstwo.
- [9] Palmowska, A., & Lipska, B., (2016). Experimental study and numerical prediction of thermal and humidity conditions in the ventilated ice rink arena. *Building and Environment, 108*, 171–182.
- [10] Nishioka, T., Ohtaka, K., Hashimoto, N., & Onojima, H. (2000). Measurement and evaluation of the indoor thermal environment in a large domed stadium. *Energy & Buildings*, 32, 217–223.
- [11] Wang, H., Chen, H., Cui, Y., Jia, X. (2015). Research on a Secondary Airflow-Relay System to Improve Ventilation Performance of Nozzle Supply in Large Space Buildings. *Procedia Engineering*, 121, 816–823.
- [12] Song, J., Meng, X. (2015). The Improvement of Ventilation Design in School Buildings Using CFD Simulation. *Procedia Engineering*, 121, 1475–1481.