ARCHITECTURE CIVIL ENGINEERING ENVIRONMENT
The Silesian University of Technology
No. 4/2019

doi: 10.21307/ACEE-2019-059

NUMERICAL STUDIES ON THE NATURAL SMOKE VENTING OF ATRIA

Karolina ŻYDEK [a]*, Małgorzata KRÓL [b]

[a] MSc.; Department of Heating, Ventilation and Dust Removal Technology, Faculty of Energy and Environmental Engineering, Silesian University of Technology, Konarskiego 20, 44-100 Gliwice, Poland
*E-mail address: *karolina.zydek@polsl.pl*

[b] PhD Eng.; Department of Heating, Ventilation and Dust Removal Technology, Faculty of Energy and Environmental Engineering, Silesian University of Technology, Konarskiego 20, 44-100 Gliwice, Poland

Received: 15.10.2019; Revised: 18.11.2019; Accepted: 18.11.2019

Abstract

The use of numerical fluid mechanics to model smoke flow in buildings where a fire develops is common. It allows to check the effectiveness of ventilation systems at the design stage. It also gives the opportunity to determine the conditions that will be on escape routes. Numerical analyzes of smoke flow in buildings are most often performed using Fire Dynamics Simulator (FDS).

The paper presents numerical analyzes performed for the atrium building. The purpose of the calculations was to build a numerical model that corresponds to the real object located in the laboratory in Murcia, Spain. The analyzes consisted of fitting a numerical model based on the temperature distribution at selected points of the atrium. The model mapped the geometry of the real building and assumed the same fire power. Calculations showed high temperature compliance throughout the atrium, except for the vicinity of the fire source itself.

Keywords: **FDS; Natural venting; Atrium.**

1. INTRODUCTION

Buildings with a large open space are popularly called atriums. Atria are part of modern architecture. The architects design them as separate objects or they form a part of a building complex. It is a big challenge for engineers to create appropriate conditions for people to stay in such facilities. This applies to maintaining the right temperature and air quality. However, the biggest challenge is to ensure the safety of people in the event of a fire. When a fire breaks out there is a high probability that the entire building will be filled with smoke. This will cause difficulties in the evacuation of people. People in smoke lose their orientation and their speed of movement definitely decreases [1]. Smoke can move in an uncontrolled manner as a result of the impact of external factors such as wind and temperature [2]. Especially the wind intensively affects the smoke flow in the building by creating windward and leeward zones on the outer walls [3]. The influence of wind is particularly important in the case of a natural smoke-exhausted building. The natural smoke exhaust system involves installation of the smoke exhaust vents on the roof of the building (or in the neighborhood, on the outer wall). The make-up air inlets for compensation air are located on the ground floor. The driving force of this process is the thermal buoyancy. The efficiency of such natural smoke venting depends on many factors such as the wind speed, the wind direction, the air inlets layout and the shape or the height of the building. In this situation, the designed system should be checked at an early stage of the project. Many designers do this using CFD (Computational Fluid Dynamics) programs. The

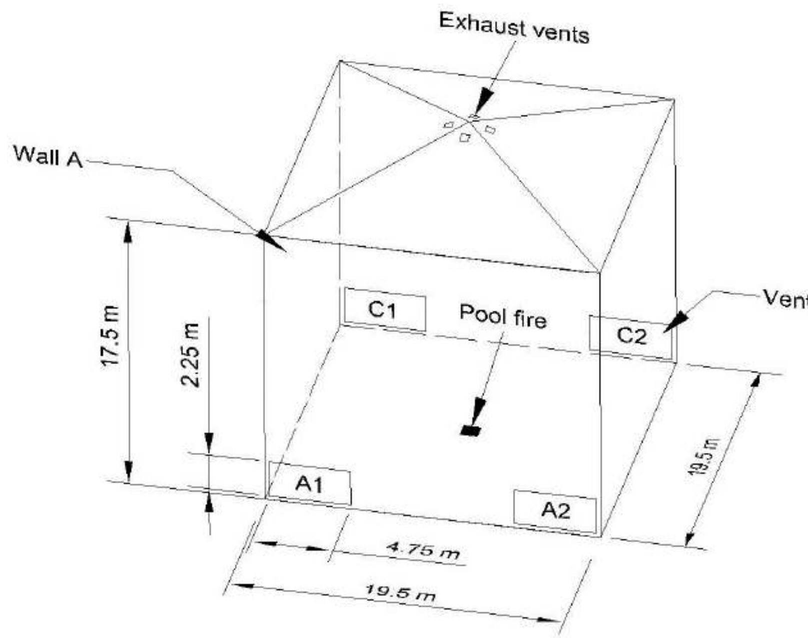

Figure 1.
Layout and main dimensions of the model

program, particularly commonly used in these circumstances is FDS (Fire Dynamics Simulator) [4]. The use of CFD programs always raises doubts about the reliability of the results obtained. When making calculations using the CFD program, many choices must be made. Among other things, it is necessary to decide on the choice of boundary conditions, the choice of turbulence model or the choice of the size of the calculation grid. All these parameters have an impact on the quality of the results obtained.

To reduce or even eliminate the uncertainty of the results obtained, there are many works aimed at the validation and verification of the FDS program [5–13]. The numerical analysis of a smoke removal system involves testing the influence of the factors which disturb the system's efficiency. The wind influence is considered above all [14–16]. Although FDS has undergone many validation analyzes, after building a new building model it is beneficial to subject it to validation based on real measurements.

The paper presents numerical analyzes made using the FDS program. The aim of the analysis was to match the numerical model to the results obtained in the real atrium building. For the validation of the numerical model, the results of the temperature distribution in the atrium located in Murcia, Spain were used. The atrium is in the laboratory of Technological Metal Centre. Many studies were carried out in the atrium and all of them were widely described [17–21]. High compliance of the temperature distribution was obtained at selected points in the numerical model compared to tests in the real object. The least compliance occurs in the vicinity of the fire source.

2. DESCRIPTION OF THE NUMERICAL TESTS

2.1. Numerical model of the atrium

To build the appropriate numerical model of the atrium, research presented by Gutierrez-Montes was used [21]. Fire test simulations were carried out using a program Fire Dynamics Simulator (FDS). FDS numerically solves equations describing the phenomena of heat flow and transport in space where fire occurs. Using the FDS program the model of the atrium was built. The model of the atrium consists of a prismatic structure of 19.5×19.5×17.5 m and a pyramidal roof raised 4.5 m at the centre. Total dimensions are 19.5×19.5×22 m. The numerical model of the atrium was the same as the real atrium described by Gutierrez-Montes [21]. In the model, the natural ventilation was designed. There are four square exhaust vents on the roof, each of the dimensions of 0.5 m. There are also four open vents at the lower parts of the walls. Two on the wall A and two on the wall C. Each vent has dimensions of 4.75×2.25 m. The model with its dimensions is shown in Fig. 1.

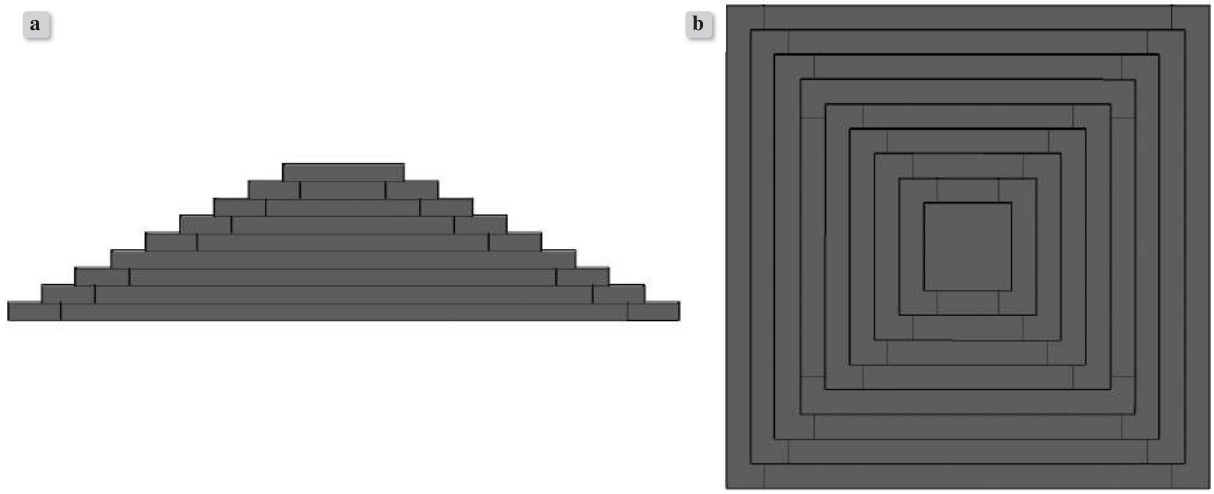

Figure 2.
The construction of the atrium roof: a) front view, b) view from above

Table 1.
Summary of the three considered cases.

Case	Heat release rate	Burning time	Dimensions of the computational domain	Dimensions of the mesh cells	Number of cells in each direction
1	2.34 MW	900 s	25×25×25 m	0.25×0.25×0.25 m	X,Y,Z: 100
2			22×22×24 m	0.2×0.2×0.2 m	X,Y: 110 Z: 120
3			25×25×25 m	5 m around plume region: 0.125×0.125×0.125 m Remaining part of the model: 0.25×0.25×0.25 m	Near the plume region: X,Y: 40 Z: 200 Remaining part of the model: X,Y,Z: 100

The pyramidal roof of the atrium was designed using nine steps. Each step was 0.5 m high. The construction of the roof is shown in Fig. 2.

The roof and walls were modelled as 6 mm thick steel with a density of 7800 kg/m^3, specific heat of 0.46 kJ/kg K and conductivity of 45 W/K m. The floor was modelled as concrete with density of 1860 kg/m^3, specific heat of 0.78 kJ/kg K and conductivity of 0.72 W/K m. In the model of the atrium, square (1×1 m) pool fire was designed. The fire source was located in the centre of the atrium floor. The burning fuel in combustion process was heptane. Three different cases of the fire test simulation have been conducted. Each of 2.3 MW heat release rate fire. In all three cases, the same ambient conditions of the weather were adopted. The ambient air temperature of 16°C, pressure of 997 mbar and humidity of 49%. The default division of solid angles has been used. Other parameters have been left as the default values. In the simulations, the ongoing 900 seconds fire was investigated.

2.2. Computational domain

The computational domain includes the atrium space, the roof, the walls and additional space around the model of the atrium. In this paper, three cases of fire simulation are shown. Each case consists of different mesh sizes and different positions of the atrium model in the computational domain. In each case, different dimensions of the mesh cells were used. To achieve optimal simulation accuracy, mesh cells that are approximately the same size in all three directions were used. Table 1. shows the meshes used.

In case 1 and 2 regular meshes were used. In these two cases, the main differences are dimensions of the mesh cells and the location of the model in the computational domain. In the first case, mesh with main dimensions of 25×25×25 m was used, while in the second case the main dimension of the mesh was 22×22×24 m. In the first case, the mesh had a dimension of the cell 0.25 m in each direction and in the second case that dimension was 0.2 m. Fig. 3 shows the position of the model in case 1. The model was situated closer to one of the computational domain boundaries. The wall A was only 1m from this boundary, which gives 4.5 m additional space between wall C and the opposite boundary. Walls without vents were located respectively 2.5 m and 3 m from the remaining boundaries. Above the roof, at the highest point, there were 3 meters to the upper boundary.

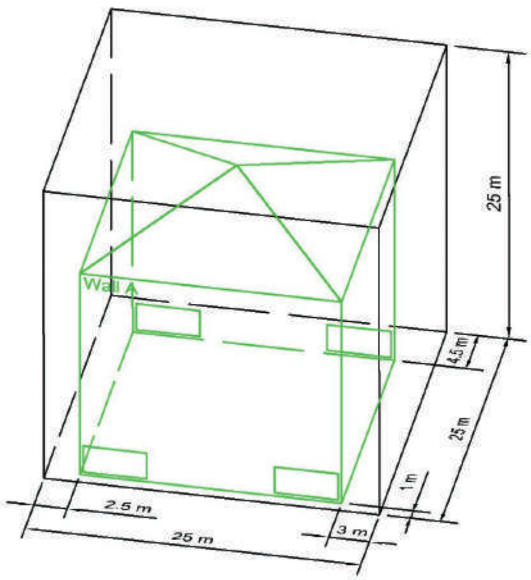

Figure 3.
Model situation in the computational domain in case 1

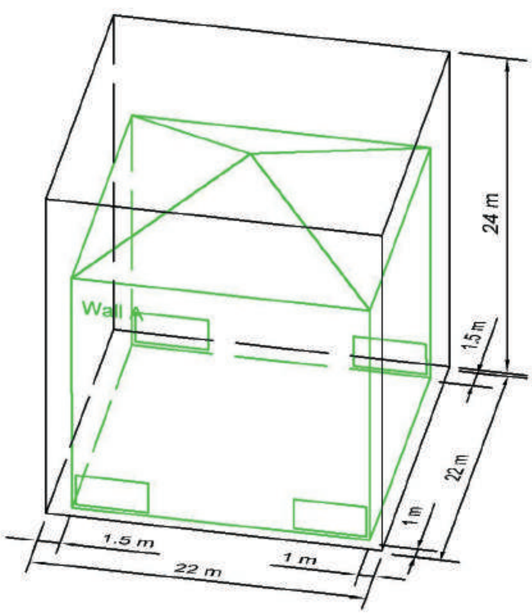

Figure 4.
Model situation in the computational domain in case 2

In the second case, the size of the computational domain has been reduced which allowed locating the model more evenly. Wall A was situated 1m from the computational domain boundary and wall C 1.5m from the opposite boundary. The remaining walls were located 1.5 m and 1m from the boundaries. In this case, the dimension from the highest point of the roof to the upper boundary was 2 meters.

In case 3, the computational domain was divided into two meshes. In the region above the pool fire and 5 meters around this region, the mesh with a cell size of 0.125 m in each direction was used. The remaining part of the computational domain was composed of a mesh with a cell size of 0.25 m in each direction. This combination was used to verify if thicker mesh around the plume region influences the results. The computational domain had dimensions of 25×25×25 m. The model was situated evenly from the boundaries. Wall A and wall C were located 2.75 m from the boundaries, and the walls without vents respectively 2.5 m and 3 m (Fig. 5). Similarly to the case 1, above the highest point of the roof were 3 meters to the upper boundary of the computational domain.

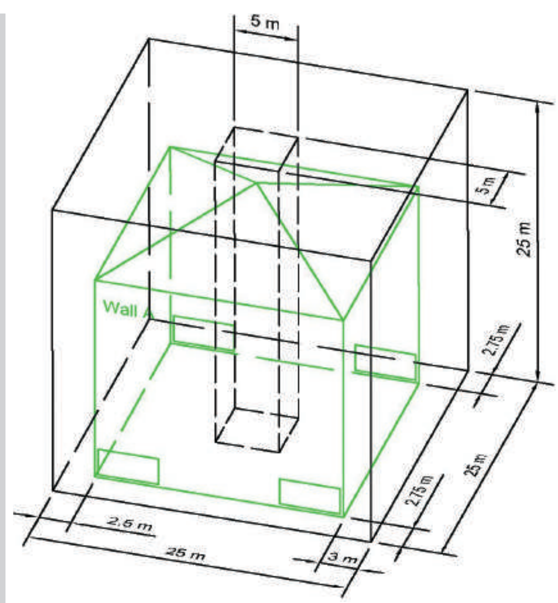

Figure 5.
Model situation in the computational domain in case 3

The most important numerical parameter in FDS is the mesh cell size. To verify how well the flow field is resolved, the non-dimensional expression $D^*/\delta x$ was calculated. According to McGrattan, the quantity $D^*/\delta x$ represents the number of computational cells spanning the characteristic diameter of the fire [4]. D^* is a characteristic fire diameter defined through

Table 2.
Relationship between D* and δx

Case	Cell size, m δx	Heat Release Rate, kW	Characteristic fire diameter, m D*	D*/δx
1	0.25	2600	1.35	5.40
2	0.20			6.75
3	0.125			10.80
	0.25			5.40

the HRR of a fire and the thermal properties of ambient conditions and δx is the nominal size of a mesh cell. In general, the smaller the characteristic fire diameter, the smaller the cell size should be in order to adequately resolve the fluid flow and fire dynamics. It is suggested that the D*/δx value has to range from 4 to 16.

$$D^* = \left(\frac{\dot{Q}}{\rho \cdot c_p \cdot T \cdot \sqrt{g}}\right)^{\frac{2}{5}} \quad (1)$$

where:

$\dot{Q}$ – Heat Release Rate, kW,

ρ – Air density (≈1.2), kg/m^3,

c_p – Air thermal capacity (≈1), kJ/kg K,

T – Ambient air temperature, K,

g – Gravitational acceleration, m/s^2

Table 2 shows the values of the D*/δx ratio for the three cases analyzed. As can be seen in all cases the values of the D*/δx ratio are within the recommended range.

2.3. Devices

To study the thermal fields induced by the fire, several sensors were installed in the model. The arrangement of sensors was the same as in the real atrium [21]. The gas phase, walls and roof temperatures were measured. Measurements of plume temperatures were taken with 3 mm K type thermocouples. The layout of the sensors is shown in Fig. 6. For air temperature measurements near the walls, 6 mm diameter type K thermocouples were used. The devices were installed 30 cm from the walls. Fig. 7. shows the layout of sensors near the wall A and wall C. To measure wall temperature, solid-phase devices were used. Gas-phase devices were used to measure temperature in exhaust vents. The roof temperature was measured using 1 mm diameter thermocouples.

The layout of the thermocouples measuring air temperature near the walls is symmetrical. As can be seen, both near the wall A and near the wall C, the sensors were installed at the same heights and the same distances.

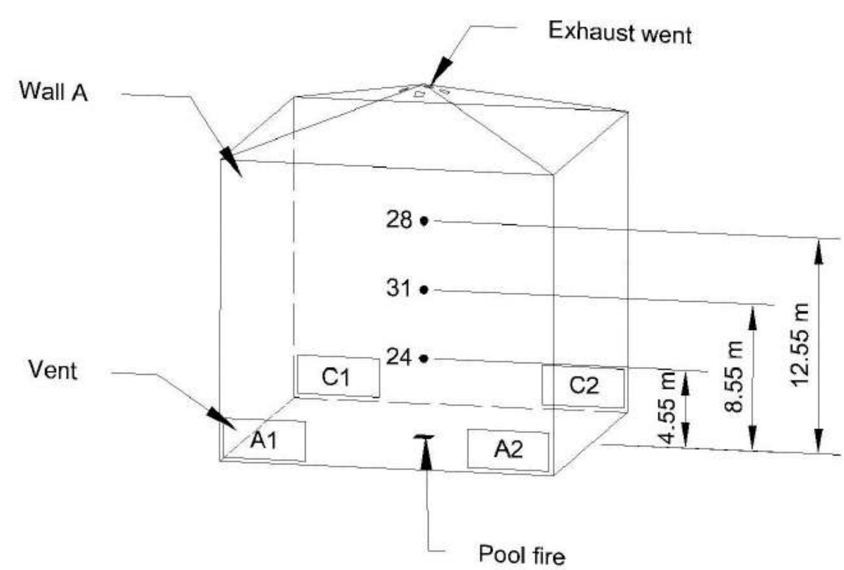

Figure 6.
Layout of the sensors in the plume region

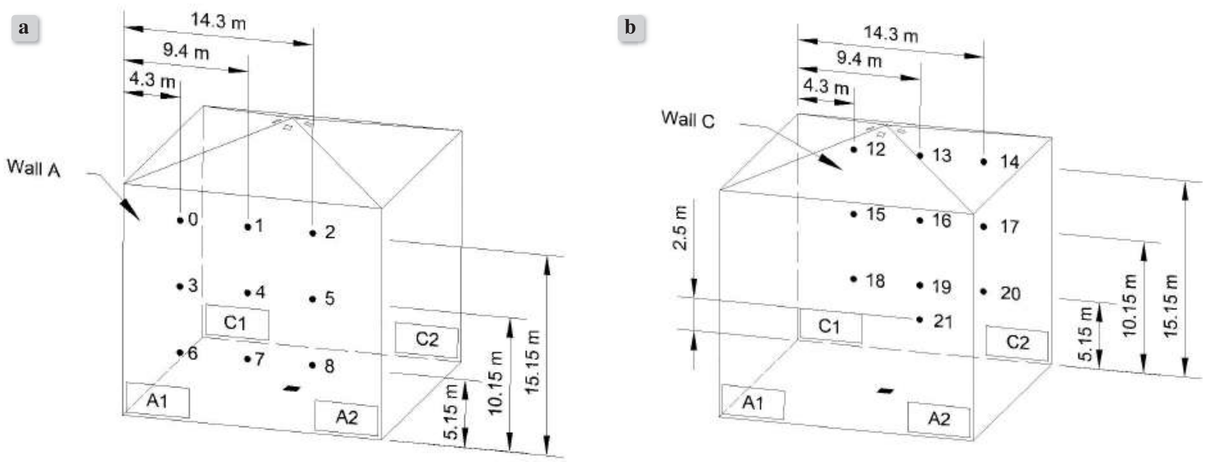

Figure 7.
Layout of the sensors near the wall: a) wall A, b) wall C

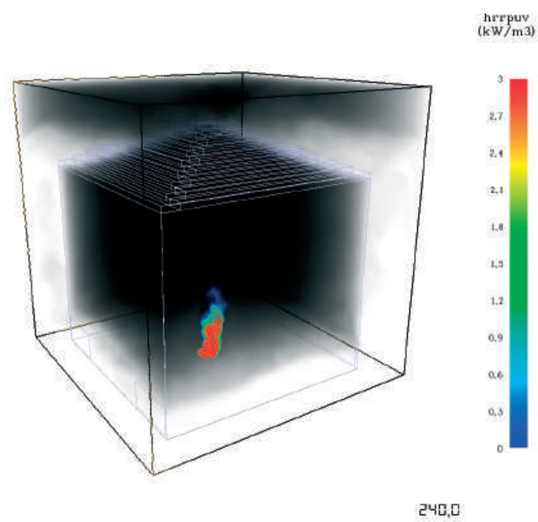

Figure 8.
FDS results. Case 3: soot density at fire time 240s

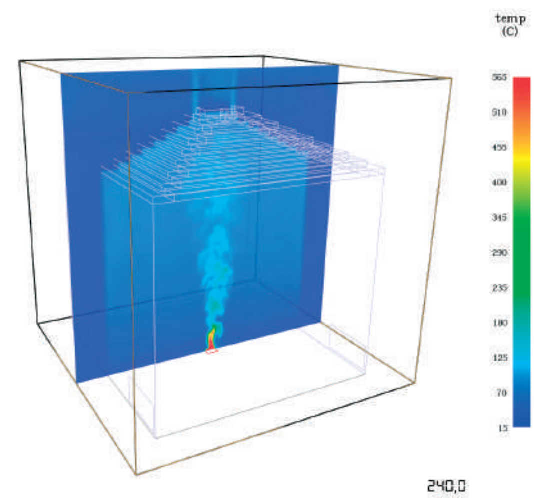

Figure 9.
FDS results. Case 3: temperature at time 240s in the section plane of a pool fire

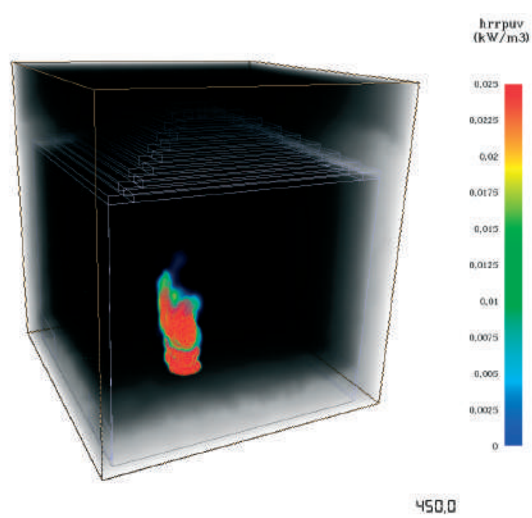

Figure 10.
FDS results. Case 2: soot density at fire time 450s

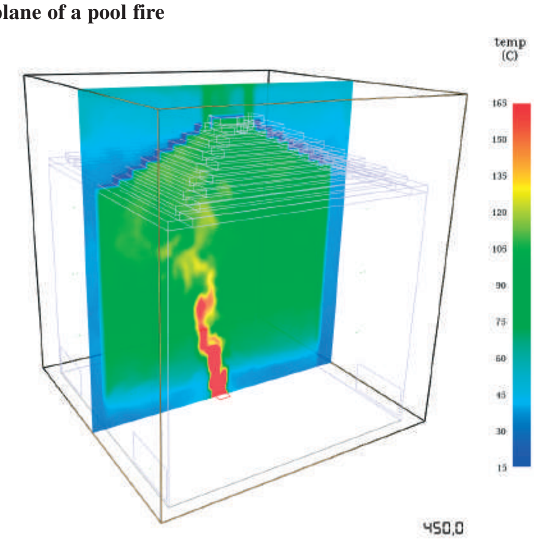

Figure 11.
FDS results. Case 2: temperature at time 450s in the section plane of a pool fire

4. RESULTS

During the numerical analyzes, attempts were made to adjust the model so that the results obtained during the simulation coincide with those obtained in studies on the real model. The calculations used temperature as a parameter whose compliance was tested. Figs. 8–11 show exemplary FDS results.

The FDS results in three key regions are reported: the exhaust smoke temperature, the plume temperature and the temperature of the air close to the wall. The results of the three cases were compared to the results of the real measurements conducted in Murcia.

The results of the air temperature analysis in the exhaust vent are shown in Fig. 12. The numerical simulation in all cases over predicts the air temperature in the exhaust vent in the early stage of fire. After 300s, the best FDS results adjustment to the real measurements displays case 2. After 300s fire ignition, the results in case 1 are significantly lower than the real measurements.

The Fig. 13 shows the comparison of the temperature in the plume in three cases. In each case, the temperature was measured at three heights with thermocouples marked “24”, “31” and “28” (Fig. 6). The diagrams below show the plume temperature at h = 4.55 m (Fig. 13a), at h = 8.55 m (Fig. 13b) and at h = 12.55 m (Fig. 13c). In comparison to the measurements in real atrium, FDS predicts much lower plume temperatures on the height 4.55 m. At this height there are the greatest differences between the measurements and the results of the calculations. At other heights (8.55 m and 12.55 m), the fit is greater. The calculation results show the highest agreement with the measurements in the real atrium at the height 12.55 m for case 2. As regards the simulation at 12.55 m, the best general correlation to the real measurements shows case 2 (Fig. 13c).

The Fig. 14 shows the results of the temperature close to wall A, which is similar to the temperature close to wall C. As previously, the temperature at three heights has been shown. Presented temperatures were measured by three thermocouples marked “1”, “4”, and “7” (Fig. 7a).

As can be seen the temperature rises at the height of 15.15 m first due to the smoke accumulation under the roof. (Fig. 14a). At this height, temperature achieves the highest values. The lowest values of the temperature close to wall A were observed at the height of 5.15 m.

Results of the analysis at the height of 5.15 m indicate the worst correlation with measurements in the real atrium. FDS over predicts the temperatures near the wall (Fig. 14c). This can be caused by the sensitivity to flow perturbations at the lowest parts. It is observed that the FDS slightly overestimate the final smoke layer height. The best adjustment between FDS results and real measurements for all cases is observed at height 15.15 m. Regard for the grid study, all of the cases display similar results of air temperature near the wall at all heights.

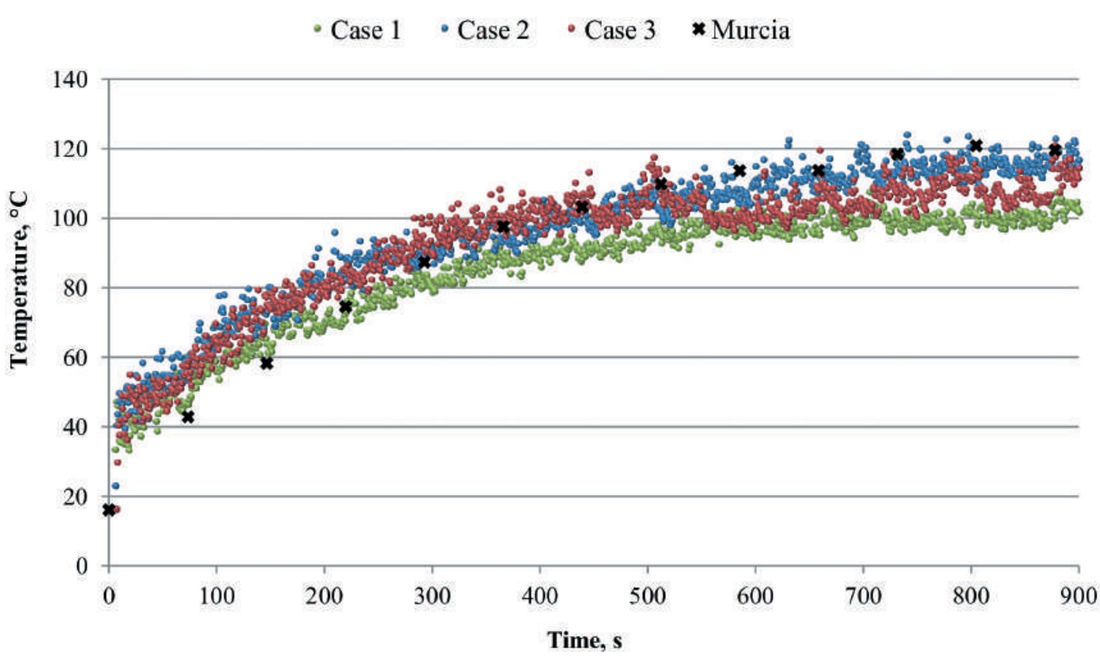

Figure 12.
Temperature predictions in the exhaust vent

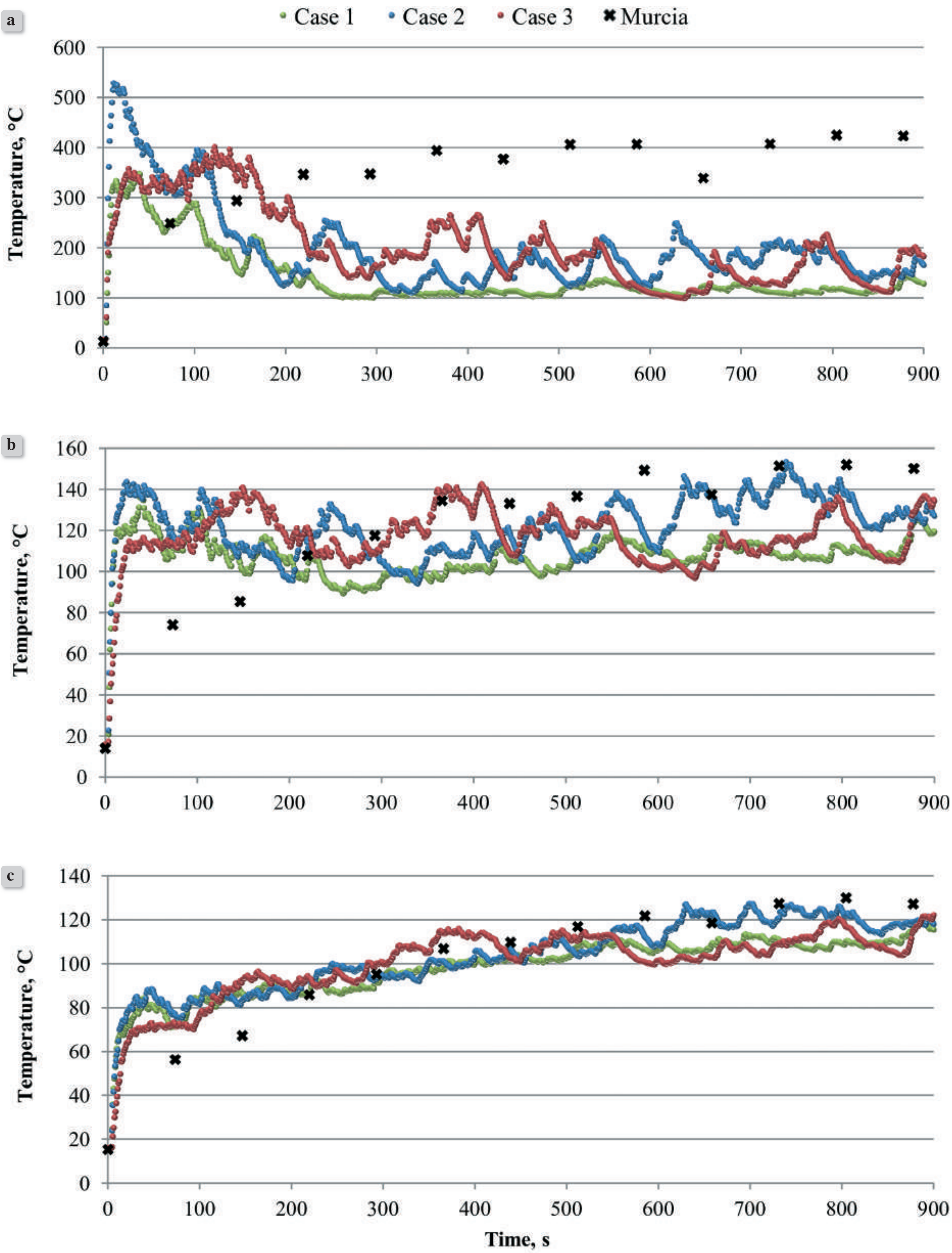

Figure 13.
Temperature predictions in the plume: a) 4.55 m above the pool fire, b) 8.55 m above pool fire, c)12.55 m above pool fire

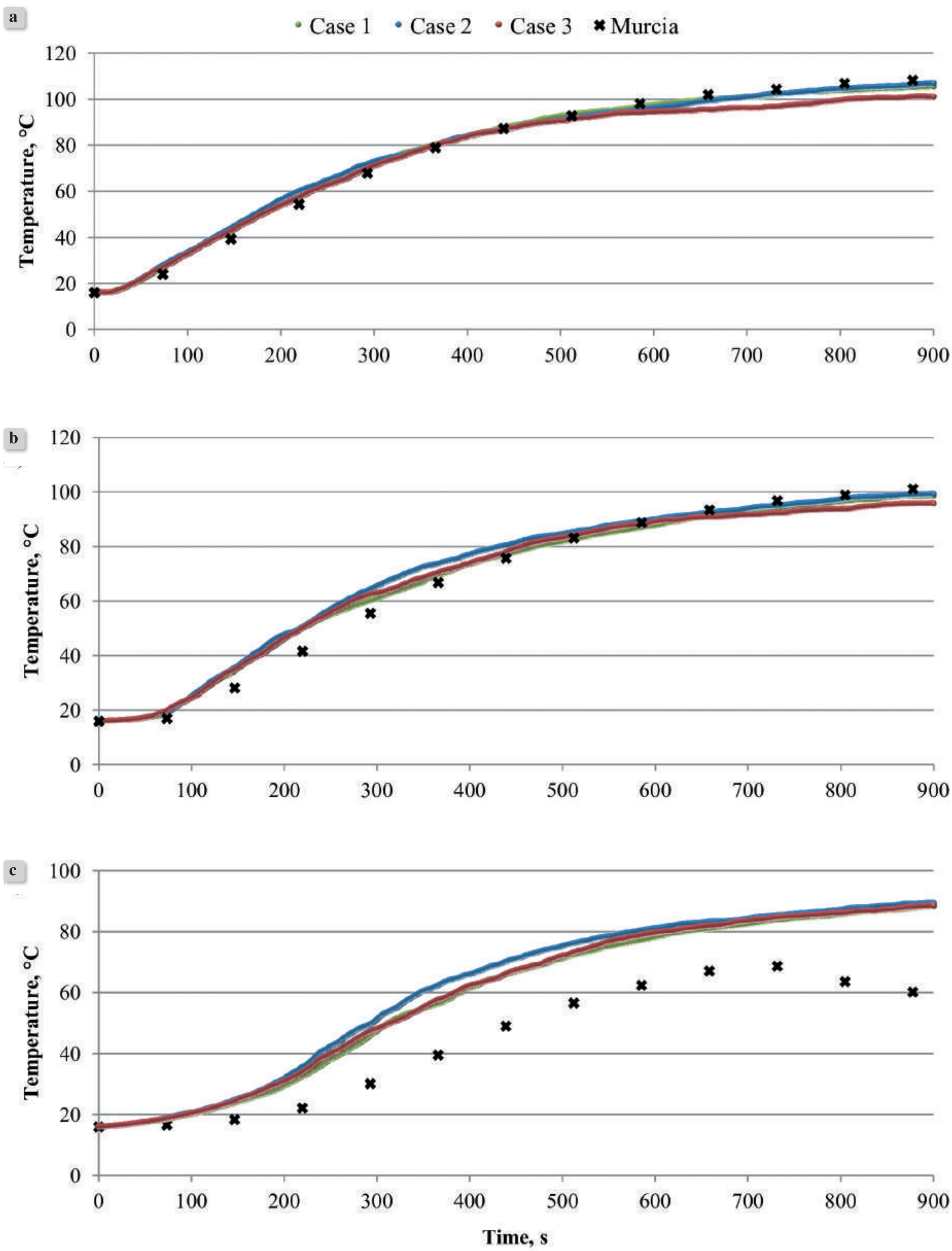

Figure 14.
Air temperature predictions close to wall A: a) h = 15.15 m, b) h = 10.15 m, c) h = 5.15 m

4. CONCLUSION

The paper presents the results of three numerical simulations carried out in FDS. Simulations have been conducted to check whether FDS can correctly determine the temperature distribution in the atrium model. The calculation results have been compared with the results obtained from measurements in the real atrium. Three cases of the simulation were conducted to check the influence of the grid size on the results.

Calculations show good agreement with the results of the measurements in the atrium, except the plume region near to pool fire. The space above the source of fire is a place of very intense turbulent flows. The best compatibility of the results of measurements and numerical calculation for the temperature was obtained for a uniform mesh of 0.2 m. Creating a denser grid around the fire source did not improve the numerical results. Incompatibility of the results of measurements and numerical calculations appeared above the fire source regardless of the density of the mesh.

REFERENCES

[1] Klote, J.H., Milke, J.A., Turnbull, P.G., Kashef, A., Ferreira, M.J. (2012). Handbook of Smoke Control Engineering. ASHRAE, Atlanta, GA.

[2] Mowrer, F.W. (2009). Driving forces for smoke movement and management. *Fire Technology* 45, 147–162.

[3] Król, M., Król, A. (2017). Wind influence on a building with the natural smoke removal system. *Architecture Civil Engineering Environment 3*(10), 119–126.

[4] McGrattan, K. Hostikka, S. McDermott, R. Floyd, J. Weinschenk C., Overholt, K. (2015). NIST Special Publication 1019 Fire Dynamics Simulator User's Guide Version 6, National Institute of Standards and Technology.

[5] Ayala, P., Cantizano, A., Gutierrez-Montes, C., Rein, G. (2013). Influence of atrium roof geometries on the numerical predictions of fire tests under natural ventilation conditions. *Energy and Buildings* 65, 382–390.

[6] Capote, J.A., Alvear, D., Abreu, O.V., Lazaro, M., Espina, P. (2009). Scale tests of smoke filling in large atria. *Fire Technology* 45, 201–220.

[7] Ray, S.D., Gong, N.-W., Glicksman, L.R., Paradiso, J.A. (2014). Experimental characterization of full-scale naturally ventilated atrium and validation of CFD simulations. *Energy and Buildings* 69, 285–291. doi:10.1016/j.enbuild.2013.11.018

[8] Rundle, C.A., Lightstone, M.F., Oosthuizen, P., Mouriki, E., (2011). Validation of computational fluid dynamics simulations for atria geometries. *Building and Environment* 46, 1343–1353.

[9] Tilley, N., Merci, B. (2009). Application of FDS to adhered spill plumes in atria. *Fire Technology* 45, 179–188.

[10] Tilley, N., Merci, B. (2013). Numerical study of smoke extraction for adhered spill plumes in atria: impact of extraction rate and geometrical parameters. *Fire Safety Journal* 55, 106–115.

[11] Tilley, N., Rauwoens, P., Merci, B. (2011). Verification of the accuracy of CFD simulation in small-scale tunnel and atrium fire configurations. *Fire Safety Journal* 46, 186–193.

[12] Yang, P., Tan, X., Xin, W. (2011). Experimental study and numerical simulation for a storehouse fire accident. *Building and Environment* 46, 1445–1459.

[13] Xiao, B. (2012). Comparison of numerical and experimental results of fire induced doorway flows. *Fire Technology* 48, 595–614.

[14] Król M. (2016). Numerical studies on the wind effects on natural smoke venting of atria. *International Journal of Ventilation* 15, 67–78.

[15] Król M., Król A. (2017). Multi-criteria numerical analysis of factors influencing the efficiency of natural smoke venting of atria. *Journal of Wind Engineering & Industrial Aerodynamics* 170, 149–161.

[16] Węgrzyński W., Krajewski G. (2017). Combined Wind Engineering, Smoke Flow and Evacuation Analysis for a Design of a Natural Smoke and Heat Ventilation System. *Procedia Engineering* 172, 1243–1251.

[17] Alaya P., Cantizano A., Rien G., Vigne G., Gutierrez-Montes C. (2016). Fire Experiments and Simulations in a Full-Scale Atrium Under Transient and Asymmetric Venting Conditions. *Fire Technology* 52, 51–78.

[18] Gutierrez-Montes C., Sanmiguel-Rojas E., Burgos M.A., Viedma A. (2012). On the fluid dynamics of the make-up inlet air and the prediction of anomalous fire dynamics in a large-scale facility. *Fire Safety Journal* 51, 27–41.

[19] Gutierrez-Montes C., Sanmiguel-Rojas E., Kaiser A.S., Viedma A. (2008). Numerical model and validation experiments of atrium enclosure fire in a new fire test facility. *Building and Environment* 43, 1912–1928.

[20] Gutierrez-Montes C., Sanmiguel-Rojas E., Viedma A. (2010). Influence of different makeup air configurations on the fire-induced conditions in an atrium. *Building and Environment* 45, 2458–2472.

[21] Gutierrez-Montes C., Sanmiguel-Rojas E., Viedma A., Rien G. (2009). Experimental data and numerical modeling of 1.3 and 2.3 MW fires in a 20 cubic atrium. *Building and Environment* 44, 1827–1839.