

THESIS ABSTRACT

Defended before

TECHNICAL UNIVERSITY OF CIVIL ENGINEERING BUCHAREST

To obtain the

PHD DEGREE

Specialization: Civil Engineering and Installations

By

LULEA MARIUS DORIN

RESEARCH ON THE INTERACTION OF ACTIVE FIRE PROTECTION SYSTEMS IN BUILDINGS

Defended on September 27, 2021

Member of the Jury

President	Prof. univ. dr. ing. Andrei Mugur GEORGESCU	Universitatea Tehnică de Construcții București
PhD Supervisors	1.Prof. univ. em. dr. ing. Vlad IORDACHE	Universitatea Tehnică de Construcții București
	2.Conf. univ. dr. ing. Ilinca NĂSTASE	Universitatea Tehnică de Construcții București
Official References	Conf. univ. dr. ing. ing. Florin BODE	Universitatea Tehnică Cluj-Napoca
	Conf. univ. dr. Ion ANGHEL	Academia de Poliție “Alexandru Ioan Cuza”
	Conf. univ. dr. Cătălin TEODOSIU	Universitatea Tehnică de Construcții București

TABLE OF CONTENTS

RESEARCH ON THE INTERACTION OF ACTIVE FIRE PROTECTION SYSTEMS IN BUILDINGS.....	1
TABLE OF CONTENTS.....	3
1. INTRODUCTION	5
2. EXPERIMENTAL STUDY.....	6
3. DETERMINATION OF THE NUMERIC MODEL.....	11
4. SIMULATION AND RESULTS.....	14
5. CONCLUSIONS.....	26

1. INTRODUCTION

This research thesis deals with two of the components that play a core role in ensuring the fundamental requirement of fire safety: the mechanical ventilation installations for the evacuation of smoke and hot air charged with pollutants and those used for the automatic extinguishing of fires with water, in this case the sprinkler systems.

In order to find out some of the answers to these questions, this research included two main components: the experimental study and the numerical study.

During a fire, the main attributes of the fire safety of buildings, in order of their importance, are the following: saving the users of buildings, preventing the collapse of buildings, protecting the intervention personnel (fire-fighters), preventing the fire from spreading inside the building as well as to other adjacent buildings, saving property and reducing material damage.

The main objective of this research is to establish how the mechanical ventilation system used to exhaust hot air charged with pollutants and smoke affects, through its operation, the triggering times of the automatic sprinkler water systems.

The automatic water sprinkler systems are an assembly that uses water to contain and extinguish a fire.

The ventilation of a space during the action of a fire is very important because it exhausts hot air, reduces the amount of noxious gases, ensures an adequate concentration of oxygen and eliminates smoke while providing good visibility.

In the technical and scientific literature, three types of models are currently most widely used: simplified mathematical models, zonal models, complex CFD (computational fluid dynamics) field models.

The studies in the field of fire safety and of the modelling of the fire action on a construction are focused on two main directions: experimental studies and numerical studies, or a combination of both. Experimental studies are carried out either at a real scale or at a reduced scale. Numerical studies shall be carried out at full scale. The conclusion of the bibliographic research is that in the vast majority of cases the study of fire is carried out by using mathematical models due to the higher degree of accessibility.

2. EXPERIMENTAL STUDY

The experimental study consisted in using a booth to carry out full scale fire simulation. The experimental booth is a prefabricated container-type construction. It is located in the courtyard of the Faculty of Installations Engineering, at 66 Pache Protopopescu Street, Bucharest.

The booth has interior dimensions of $B \times L \times H$: $3 \times 6 \times 2.7$ (m^3). The external enclosures are made of sandwich panels with a thickness of 0.10 (m), made of non-combustible materials, $A2s1d0$, respectively of mineral wool [6]. The panels are certified with fire performance, i.e. $EI180$ fire resistance (**Fig. 1**) [7].



Fig. 1. Experimental booth.

A fire resistant door, EI90-C [70], measuring 0.9×2.1 (m^2) was provided for access to the interior.

For the direct outside viewing and surveillance of the experiment, a fireproof glass, E30, with dimensions of 0.6×1.2 (m^2), was installed.

Two more voids were provided in the outer walls necessary for the operation of the mechanical ventilation system: one for the extraction of air by means of the fan, arranged on the upper side of the outer wall, with dimensions of 0.25×0.25 (m^2), and a second one for the

natural-organized supply of fresh compensation air arranged on the opposite side on the lower side, in the immediate vicinity of the burner, with the dimensions of $0.25 \times 0.10 \text{ (m}^2\text{)}$.

For the introduction of the water distribution system to the sprinkler, a void was provided at the upper side, with the size of $0.25 \times 0.25 \text{ (m}^2\text{)}$, but it was plugged during the experiments.

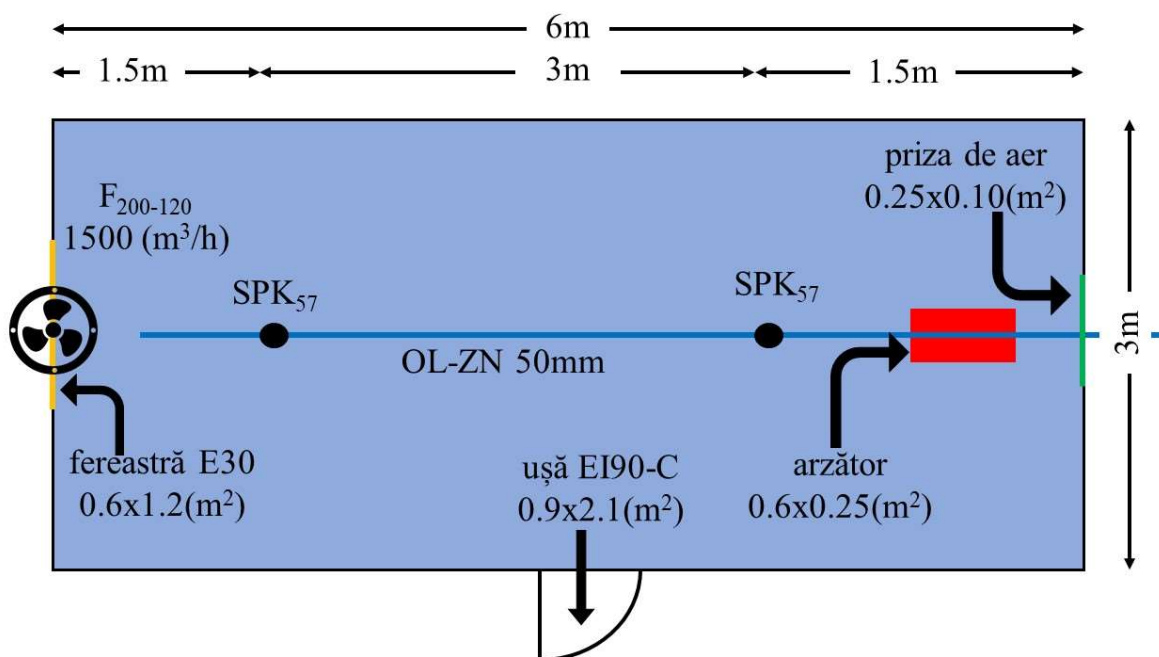


Fig. 2. Experimental Plan Stand.

Next to the experimental booth, only 1.50 (m) from it, an operator container was mounted. It has the following interior dimensions $3 \times 2 \times 2.7 \text{ (m}^3\text{)}$ (Fig. 25). In this office were arranged the equipment for monitoring the research, the equipment used for recording data during the experiment, the electrical panel, the computer used for downloading and processing data, the equipment for measuring and control, various small materials used in the research were temporarily stored.

In the experiments and numerical modelling, the fuel used was *LPG* gas (mixture of propane and butane) from a 60 (l)cylinder. The main components of the combustion plant are the following: 2 burners with controlled flow rate 3 (kW) and 12.2(kW), LPG cylinder 60 (l), metal gas supply line from cylinder to the burners, made of 20 (mm) steel, gas flow/pressure regulator, service or control valves.

The fan used is *PR-Q 456T AT*. This is a fan that is guaranteed to operate correctly up to air temperatures of maximum $200\ (^{\circ}\text{C})$. The fan can provide an air extraction flow from $0\ (\text{m}^3/\text{s})$ to $1,500\ (\text{m}^3/\text{s})$.

During the experimental tests, the temperature distribution over time was recorded. This was done using 35 temperature sensors, K-type thermocouples. The sensors were mounted in two orthogonal planes: one vertical, mid-space, and one arranged at the top, in the sprinkler plane (Fig. 3).

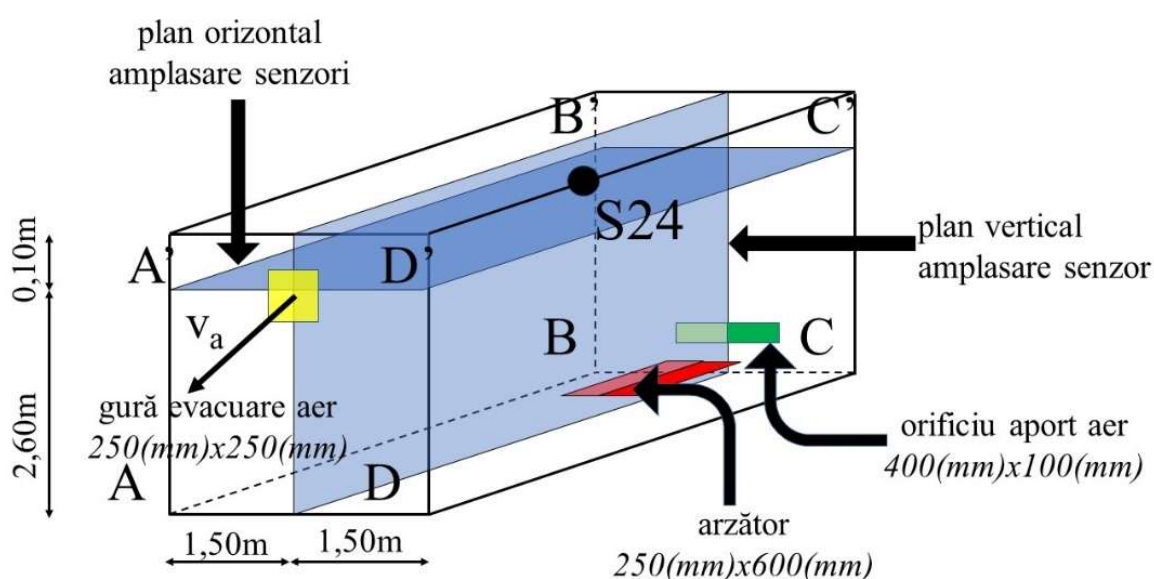


Fig. 3. Horizontal and vertical plane for the location of temperature sensors.

Two ALMEMO 710 data acquisition units were used to record data.

Two sprinklers were mounted inside the container, in the upper plane of the room. Sprinklers with a temperature trigger of $68\ (^{\circ}\text{C})$ were used. The sprinklers were mounted head up. The sprinklers were mounted at $0.1\ (\text{m})$ from the ceiling, with a distance of $3\ (\text{m})$ between them. They are located in the median plane of the room, at the intersection of the two main planes for the installation of temperature sensors.

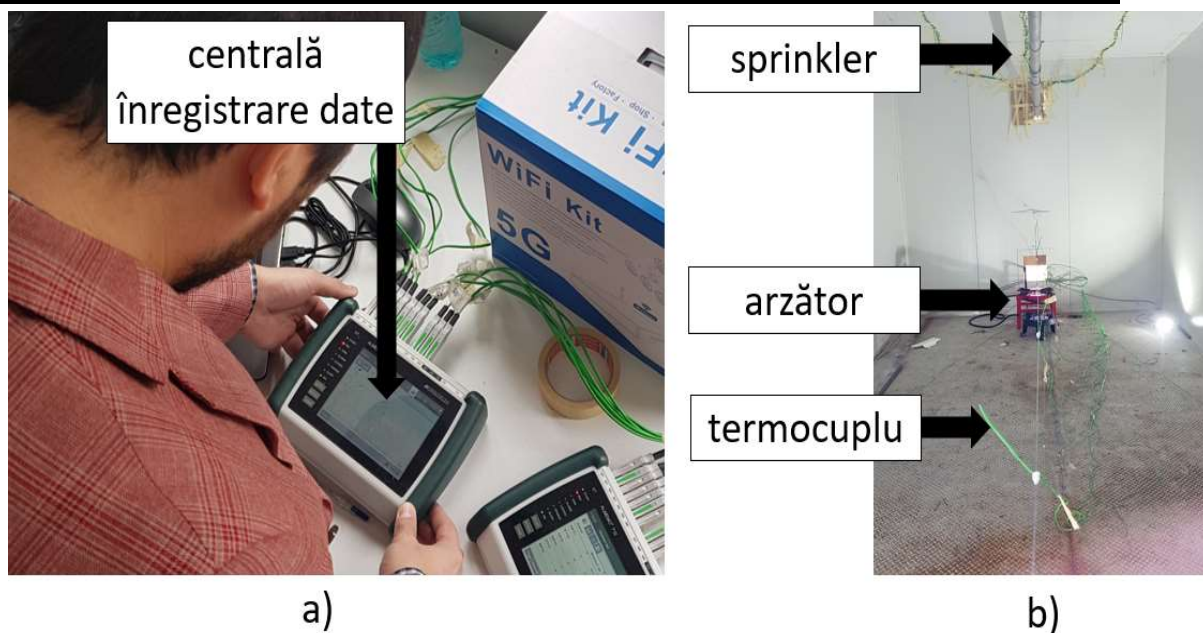


Fig. 31. Pictures during the experimental tests: a) ALMEMO central operation and b) experiment execution.

The following predetermined steps were carried out during each experiment:

- There were 2 operators: one in the office, for permanent surveillance with video cameras, and another for possible interventions or adjustments;
- the temperature sensors are connected to the ALMEMO710 recording units;
- the operation of the LPG sensor was checked in order to record any incomplete burns or accidental leaks (this was done using the gas between a lighter);
- the operation of the video recording cameras was checked;
- the LPG cylinder is placed on a calibrated scale; the weight is recorded before the start of the experiment and after the completion of the experiment;
- during the tests that involved the ignition the fan, the fan was triggered before starting the experiment and igniting the flame;
- one operator opens the supply from the cylinder, the tap on the gas supply line and adjusts the flow rate, while the second operator ensures the ignition of the burners; the pressure regulator is used to set the oil gas supply flow rate;
- the experimental booth is left, and the door is closed;

-
- an operator monitors the camcorders and the proper functioning of the data loggers from inside the container;
 - the second operator provides direct monitoring through the window located in the outer walls;
 - at the end of the experiment duration, the LPG supply is switched off by turning off the cylinder tap and subsequently the service tap;
 - the door is opened without any operator entering the space for 5 minutes to ensure the cooling of the air and the evacuation of hot gases (the temperatures in many cases have reached almost the boiling point of the water, so temperatures that could harm the human body); the cooling of the interior space is done naturally by opening the door, evacuating the hot air and introducing cold air;
 - the weight of the cylinder is measured after the experiment is completed;
 - the second experiment is carried out after the cooling of the air and the envelope, generally after at least one hour; if the outdoor temperature of 20°C is no longer ensured, it is resumed the next day;
 - data was downloaded to the computer at the end of each day of experiments.

3. DETERMINATION OF THE NUMERIC MODEL

The experimental case with which the numerical model was determined was noted with *CALIB1* (*v0,SQ1*). It is characterized by an energy release rate $HRRPUA=105$ (kW/m^2) (heat release rate per unit area) and an average foul air exhaust rate $v_a = 0$ (m/s), i.e. an air exhaust rate of $Q_v = 0$ (m^3/s). The parameter used for calibration was the change in indoor temperature at a predetermined point in space.

The second stage was the validation of the numeric model. This was done by comparing the results obtained experimentally in three other cases, noted with *CALIB2*, *CALIB3* and *CALIB4* with those resulting from the numerical modeling (**Tab. 1**).

Tab. 1. Characteristic elements of the experimental tests.

Cod experiment	Utilizat pentru	Cod model	HRRPUA (kW/m^2)	S_Q (kWh)	v_a (m/s)	Q_v (m^3/s)	n_a (h^{-1})
CALIB1	determinare	v0,SQ1	105	3.938	0	0.0000	0.0
CALIB2	validare	v0.5,SQ1	105	3.938	0.5	0.0313	2.3
CALIB3	validare	v1,SQ1	105	3.938	1	0.0625	4.6
CALIB4	validare	v4.5,SQ1	105	3.938	4.5	0.2813	20.8

The identification of the numerical model involves going through several stages to correctly establish the following aspects: the analysis field, the analysis time, the geometric modeling, the modeling of the experimental booth envelope, the setting of the optimal grid, the modeling of the source, the setting of the initial and limit conditions.

The criterion used was to achieve a minimum mean deviation, σ_{ERR} ($^{\circ}C$), between the experimental temperature distribution at point *S24* and that within the model. To this end, several likely scenarios have been iterated until the correct one has been identified.

The analyzed range was set 0.5 (m) higher than the outer limit of the experimental booth. Thus, the size of the analysis field is: $7.2 \times 4.2 \times 3.9$ (m^3), the extension of the analysis field proving necessary for the accuracy of the results.

The analysis time was determined according to the duration and experimental results obtained. It was thus found that for *CALIB1* an event duration of 760 (s) was sufficient to reach the triggering temperatures of the sprinklers.

The geometric modeling was carried out by direct measurement of the size of the experimental booth, the details, the position and the size of the equipment of the experimental booth. They were then reproduced inside the model.

The geometrical features, i.e. the surface, position, thickness, are considered known. But apart from these, the material characteristics of the envelope are of particular importance: material density, thermal conductivity and specific heat. For the density of the outer walls and the specific heat of the material from which the outer walls are made, the values from the facade panel data sheet were used: $\rho=120 \text{ (kg/m}^3\text{)}$ and $c_p=0.840 \text{ (kJ/kg.K)}$. It was found that by using these values the results obtained are appropriate. In the case of thermal conductivity it was necessary to determine an equivalent thermal conductivity $\lambda_{eqv} \text{ (W/m.K)}$. This has a very important role because of it the heat losses from the container to the outside [8]. The detail value for the equivalent thermal conductivity $\lambda_{eqv}=0.061 \text{ (W/m.K)}$ was established by determined by detail iterations using the FDS program, iterations that are not reproduced in this graph because the curves become very close and it would not be useful for explaining the research.

For modelling using FDS the analyzed space is divided into cells. In this case, the computational cells have the same size on all three axes. It has been established in this research by comparing the experimental results with the numerical ones a grid with a side size of 0.10 (m) .

The simulation by complex CFD methods involves the modelling of the fire by introducing into the space under analysis a certain amount of energy (heat) at a given time [9]. The source geometry refers to the position in space of the burner and the burning surface. In this case, starting from the real model, an area of the burner of $0.25 \times 0.60 \text{ (m}^2\text{)}$ is approximated, and its position is simulated with the position of the burners in the real experiment. In this case, for a fuel consumption of 0.315 (kg) of LPG there is an amount of energy introduced into the system during the experiment period of $HR=3,938 \text{ (kW)}$ and $HRRPUA=105 \text{ (kW / m}^2\text{)}$.

The modelling of a fan in the SDS was done by specifying the air flow discharged in one second through a surface.

The following initial conditions apply to *CALIB1*: the initial temperature throughout the space is similar and equal to $20 \text{ (}^\circ\text{C)}$; the initial speed at all points in the analyzed space is similar and equal to 0 (m/s) ; the initial concentration of oxygen in the air is equal to 0.208 (mol/mol) ; the initial concentration of carbon dioxide in the air is equal to: $0.000387 \text{ (mol/mol)}$; the initial visibility at all free points in the space is equal to 30 (m) .

The following limit conditions apply in the case of *CALIB1*: the air temperature at the burner surface is a set one, depending on the amount of energy introduced into the system; the air speed in contact with the surface of the envelope elements and with the rest of the burner

surfaces (not the one where the heat is introduced) is equal to 0 (m/s) ; the air speed in contact with the inner and outer surfaces of the exhaust outlet is equal to 0 (m/s) .

The process of validating the previously established numerical model is a necessary step to be able to identify whether it is valid for other situations as well. In this respect, the results of three other experimental tests were used: *CALIB2*($v0.5, SQ1$), *CALIB3*($v1, SQ1$), *CALIB4* ($v4.5, SQ1$) .

Compared to *CALIB1*, an experiment in which there was no air exhaust, $Q_{v1} = 0 \text{ (m}^3/\text{s)}$, the difference between the other cases is represented by the ventilation flow: $Q_{v1} = 0.5 \text{ (m}^3/\text{s)}$, $Q_{v2} = 1 \text{ (m}^3/\text{s)}$, $Q_{v3} = 4.5 \text{ (m}^3/\text{s)}$.

If we compare how the temperatures evolve over time in the 4 experimental cases, with the corresponding numerical models, we can observe that at the same thermal load ($HRR_{PUA} = 105 \text{ (kW/m}^2)$) for a ventilation at high flows rates (*CALIB4*) the temperatures do not exceed 50°C , about half of the values recorded when there is no ventilation or it has low flow rates (*CALIB1* and *CALIB2*) (**Fig. 4**).

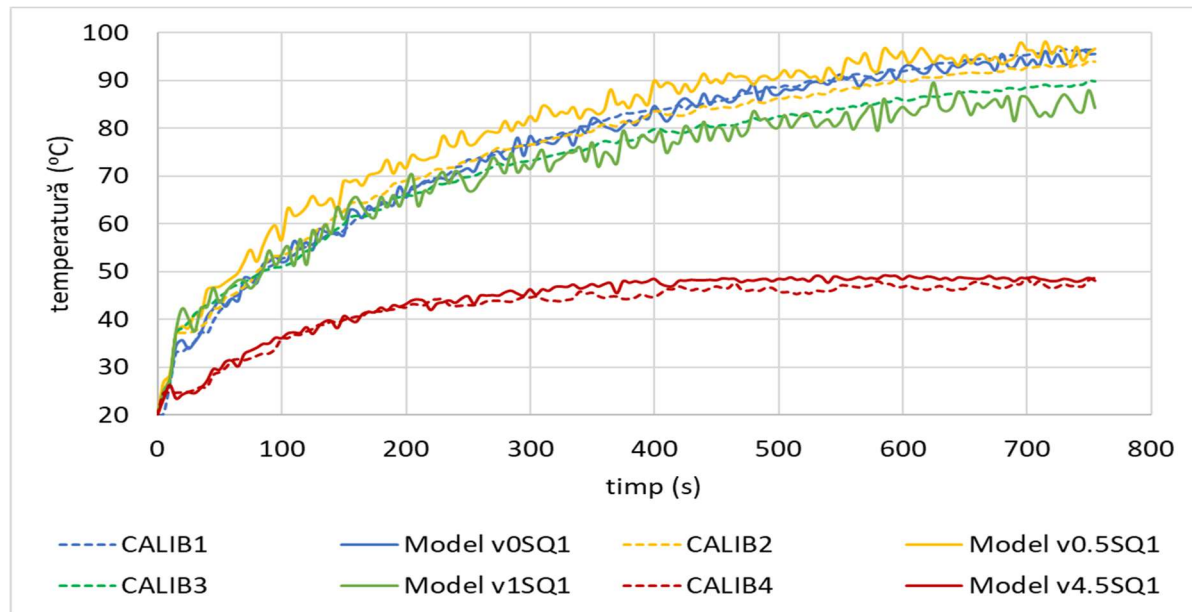


Fig. 4. Comparative graph of temperature evolution at point S24 for all 4 experimental situations and corresponding mathematical models.

It can be seen that the model established using *CALIB1* is also appropriate for *CALIB2*, *CALIB3* and *CALIB4* experimental tests. It is considered that the model is validated and can be used to extend the research to other ventilation flow rates or thermal load values.

4. SIMULATION AND RESULTS

Using the SDS program, 168 simulations were carried out, varying two parameters: the average air exhaust speed v_a , from 0 (m/s) to 9 (m/s) and the amount of energy introduced into the HRRPUA system, from 89 (kW/m²) to 10 500 (kW/m²).

The parameters which have been recorded and which will be used in the interpretation of the phenomenon are: indoor temperature at point S24; oxygen concentration at point S24; carbon dioxide concentration at point S24; visibility at point S24.

Different graphs could be established that help to understand the phenomenon such as the variation of indoor temperatures depending on the flow of the hot air exhaust fan, such as for example figure [65] below and which clearly highlights the important role of good ventilation in maintaining low temperatures that allow the survival and evacuation of the users of a building.

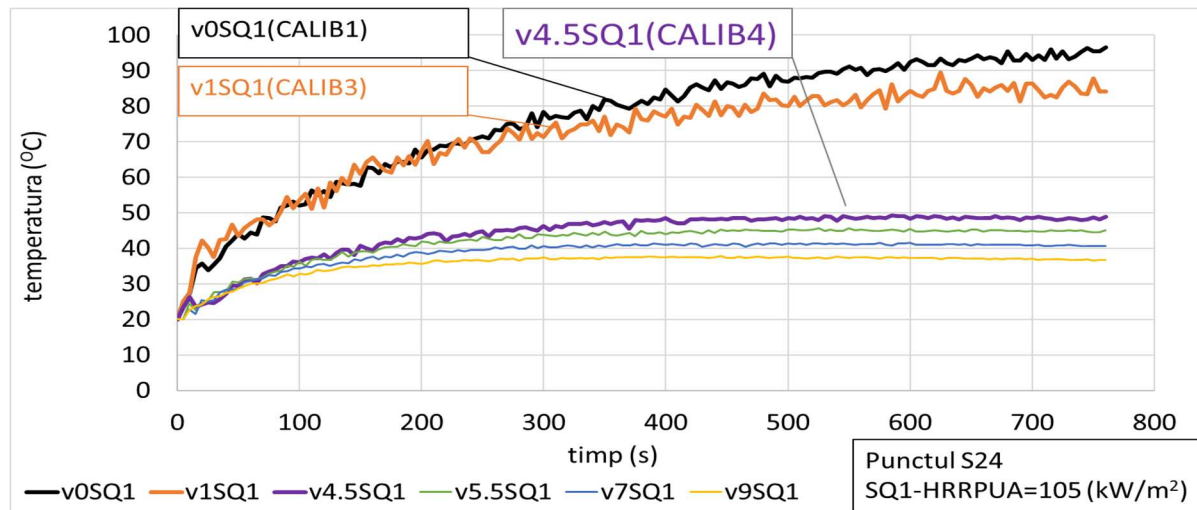


Fig. 5. Temperature trend charts at point S24 when the heat load is SQ1-HRRPUA =105 (kW/m²) at different ventilation rates characterized by air exhaust rates between 0 (m/s) and 9 (m/s).

Fig. 66 below shows how the indoor temperatures vary depending on how much combustible material is burned.

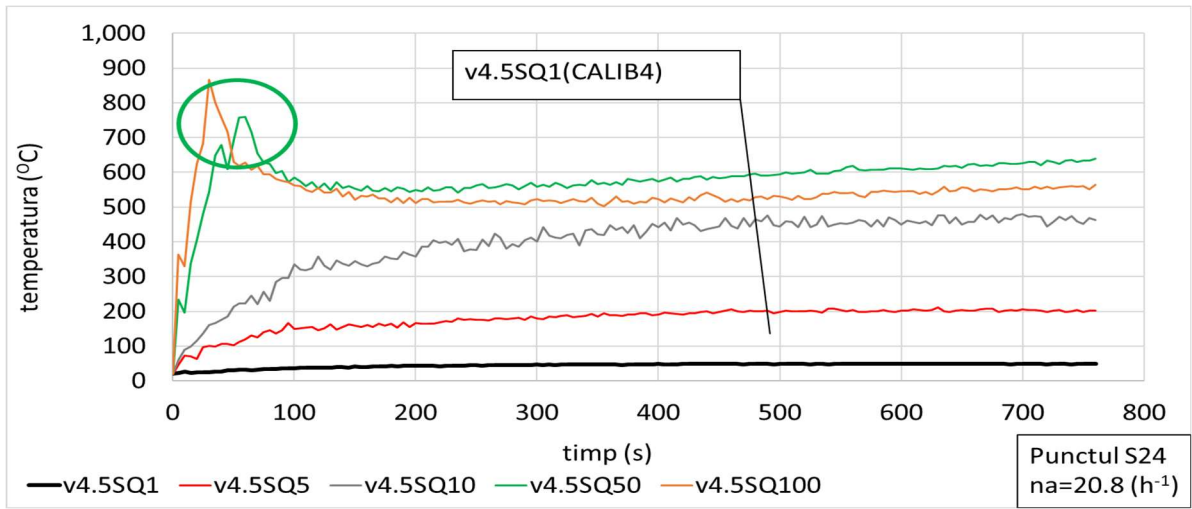


Fig. 6. Temperature evolution graphs at point S24 over time when ventilation is switched on at 20.8 hourly shifts per hour, at different heat load values between $SQ1-HRRPUA = 105 \text{ (kW/m}^2\text{)}$ and $SQ100-HRRPUA = 10\,500 \text{ (kW/m}^2\text{)}$

By knowing the evolution of indoor temperatures, it was possible to determine the time at which a particular sprinkler could be triggered (**Fig. 75**).

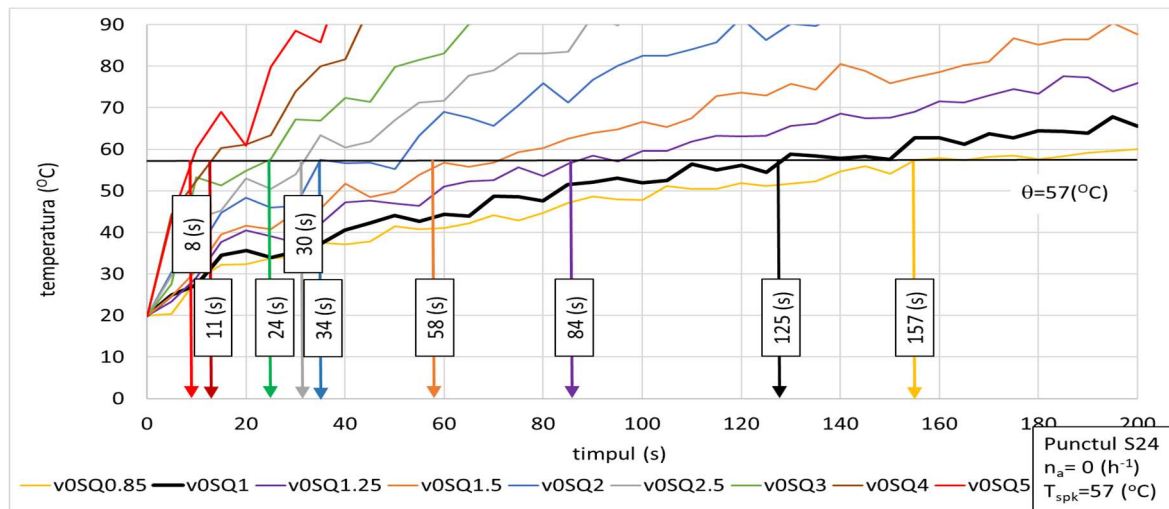


Fig. 7. Detailed graph of the evolution of the temperatures with the marking of the sprinkler trigger point with $T_{spk} = 57 \text{ (}^\circ\text{C)}$.

The numerical simulation allowed the draw a graph showing the correlation between the triggering times of the sprinklers and the amount of fuel burned (heat load), for certain ventilation flow rates (example **Fig. 79**).

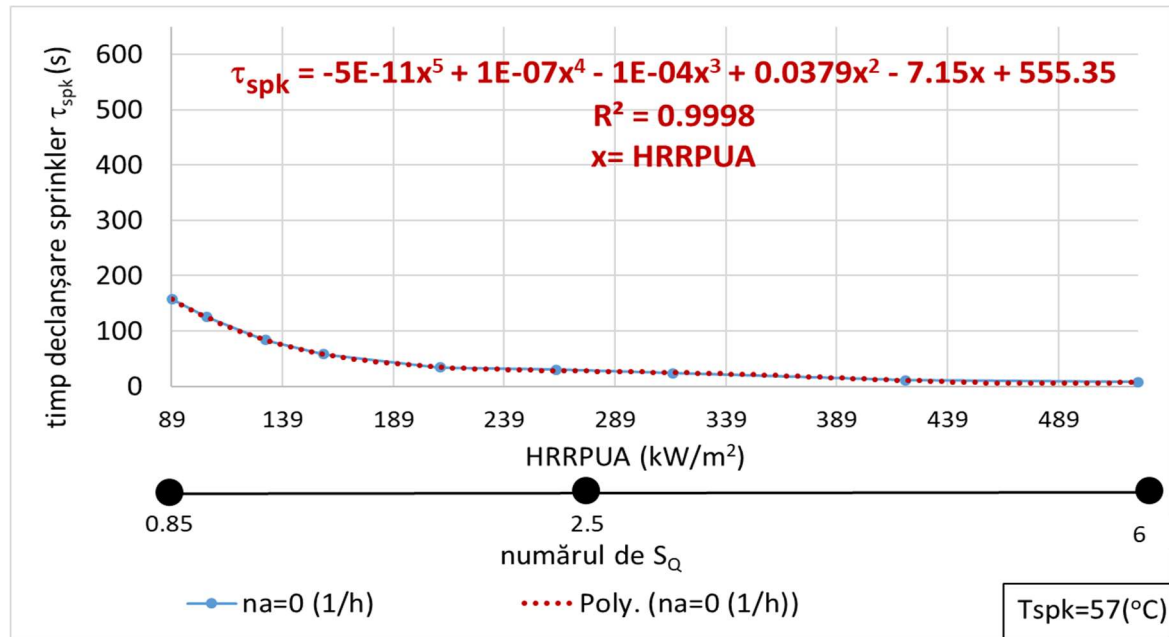


Fig. 8. The evolution graph of the triggering time of sprinklers τ_{spk} depending on the number of HRRPUA for $n_a=0(h^{-1})$.

It was possible to compare the functions that define the triggering times of the sprinklers according to the heat load, at different ventilation flow rates (**Fig. 83**).

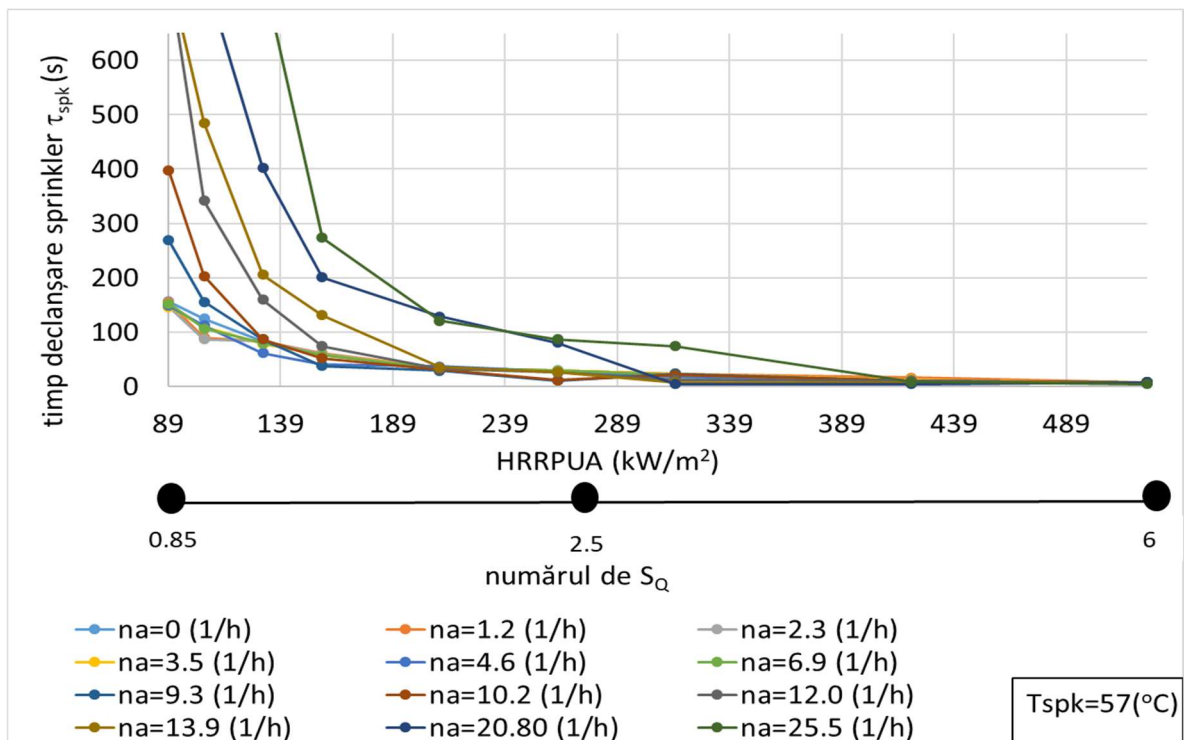


Fig. 9. The evolution graphs of the triggering time of sprinklers τ_{spk} depending on the number of HRRPUA for different hourly shifts.

The way in which other important parameters such as oxygen concentration change over time, and which plays an important role in ensuring the vital functions of the users of a building were also analyzed. For example, **Fig. 95** shows how the oxygen concentration decreases over time to values that no longer allow survival.

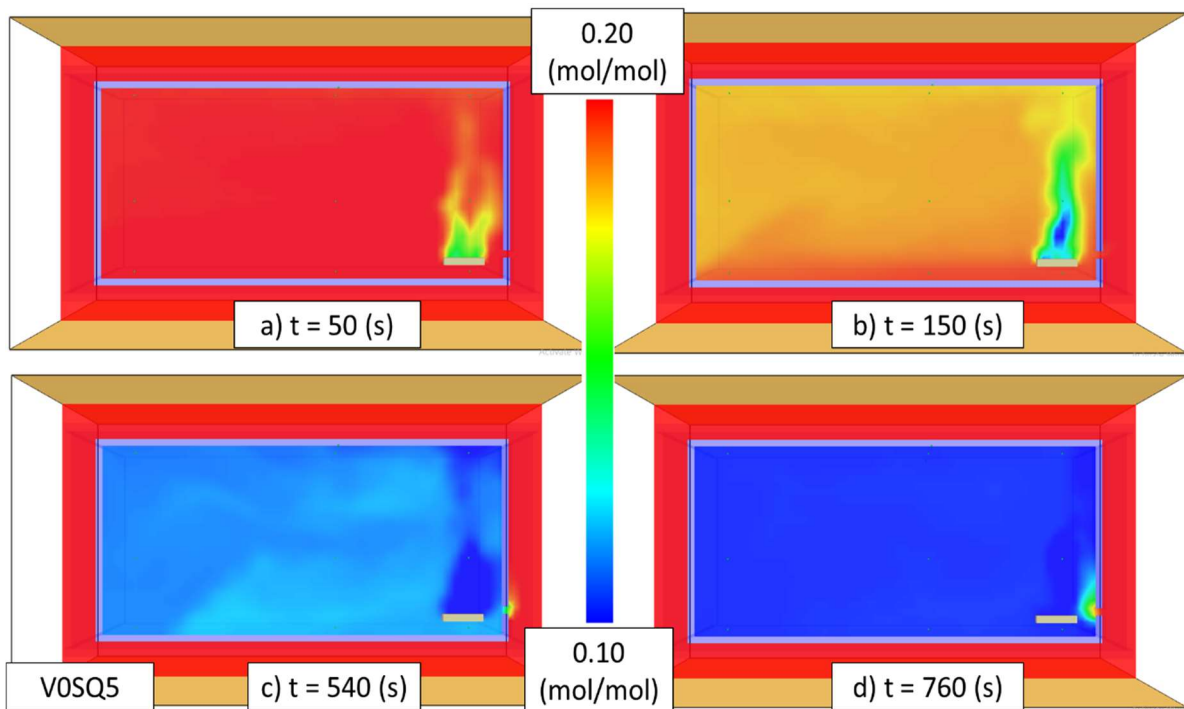


Fig. 10. The distribution graphs of the oxygen concentration in the mid-plane $Y=2.1$, for the model v0SQ5, at times: a) $t=50$ (s), b) $t=150$ (s), c) $t=540$ (s), d) $t=760$ (s).

Graphs have been plotted showing the importance of ventilation during a fire and that high ventilation rates lead to a higher concentration of oxygen, having a favorable contribution in maintaining evacuation and survival conditions. It can be seen in **Fig. 105** that proper ventilation maintains high oxygen concentrations, while in unventilated spaces we identify a rapid decrease as the oxygen converts to carbon dioxide by fuel combustion.

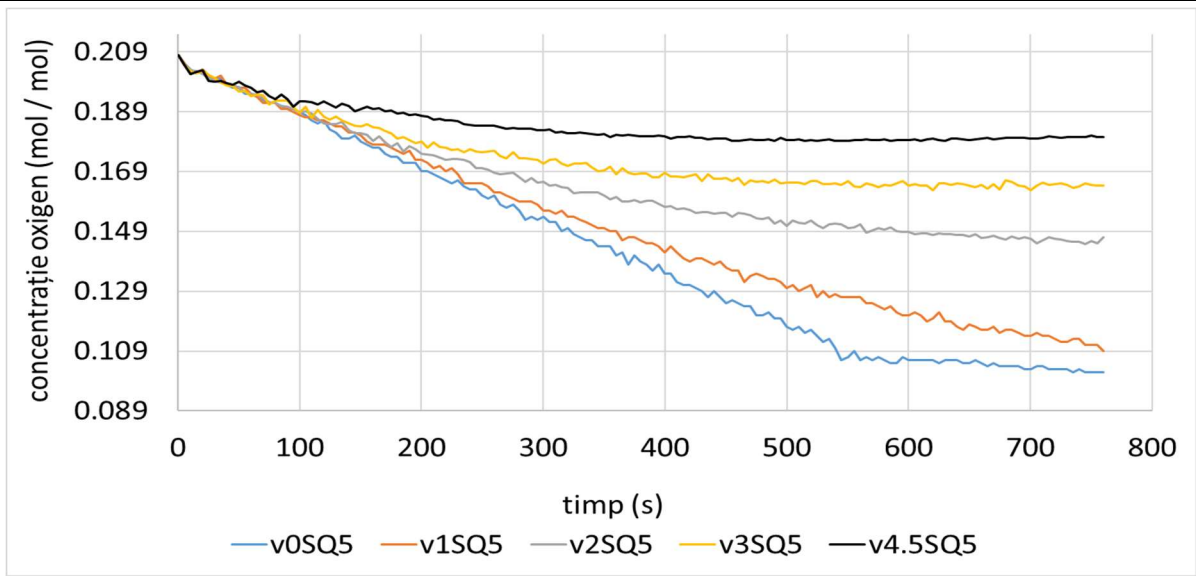


Fig. 11. Oxygen concentration trend chart at point S24, at different exhaust rates and constant heat load $SQ5-HRRPUA = 525 \text{ (kWh/m}^2\text{)}$.

The variation of visibility over time and the rapidity with which it decreases over time to values at which evacuation is no longer allowed was also identified (**Fig. 116**).

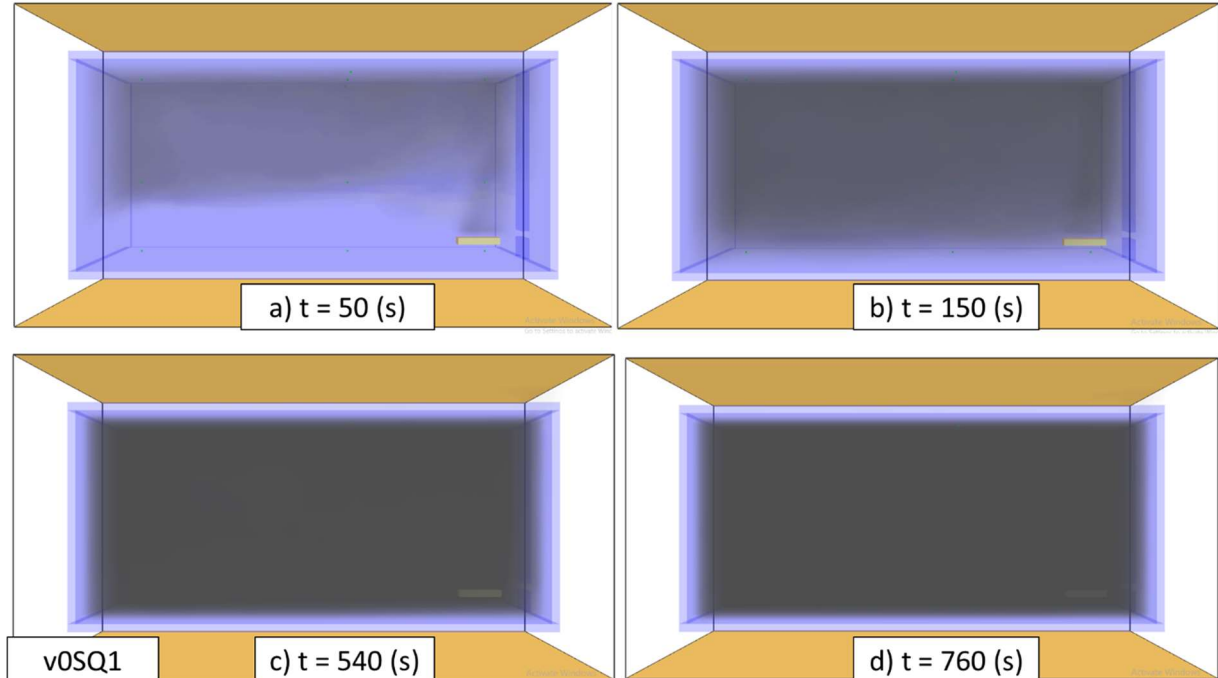


Fig. 12. The distribution graphs of the visibility in the mid-plane $Y=2.1$, for the model v0, SQ1, at times: a) $t=50 \text{ (s)}$, b) $t=150 \text{ (s)}$, c) $t=540 \text{ (s)}$, d) $t=760 \text{ (s)}$.

Good ventilation plays an important role not only in maintaining a high oxygen concentration but also in maintaining good visibility. The evacuation of hot air charged with smoke particles and the compensation with cold air without combustion residues significantly changes the conditions in the space (**Fig. 131**).

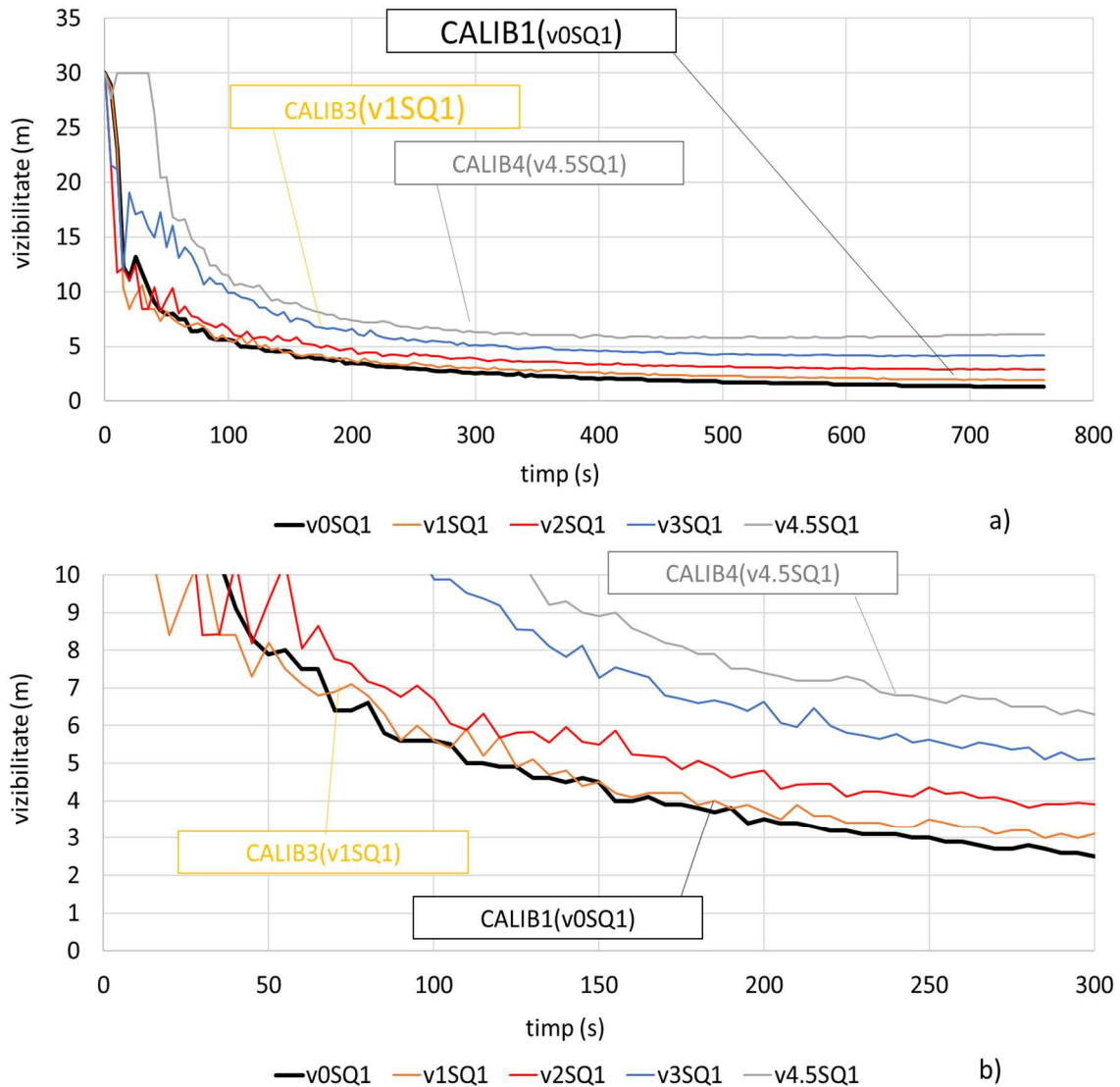


Fig. 13. Visibility evolution graph at point S24, for variable exhaust flow and constant heat load SQ1-HRRPUA = 105 (kW/m²), plotted in a) overview and b) detail.

Graphs could also be plotted showing how long a good visibility (10 m) can be maintained depending on the discharge flow rate of the foul air (Fig. 136).

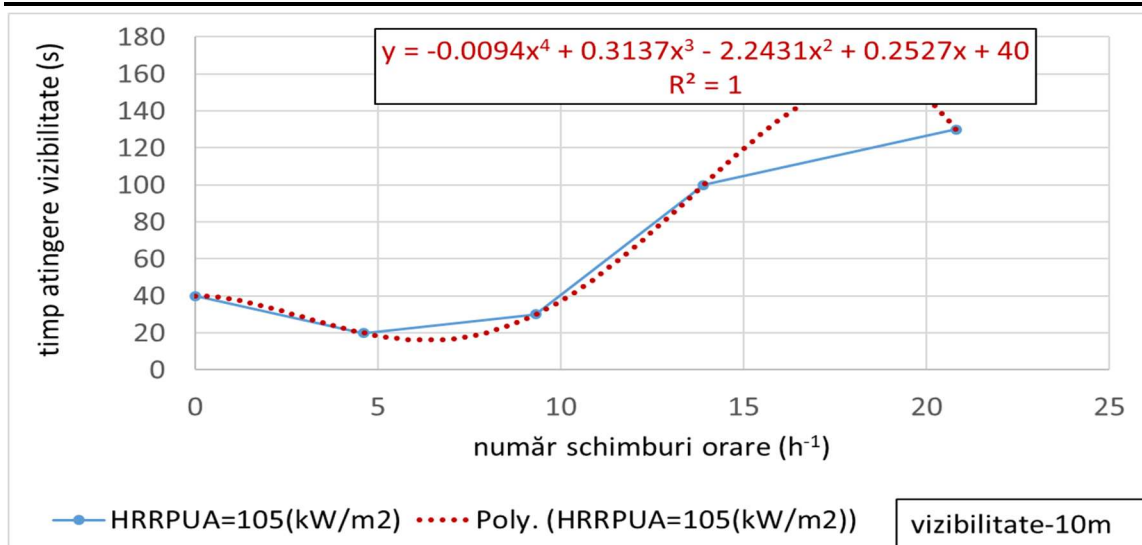
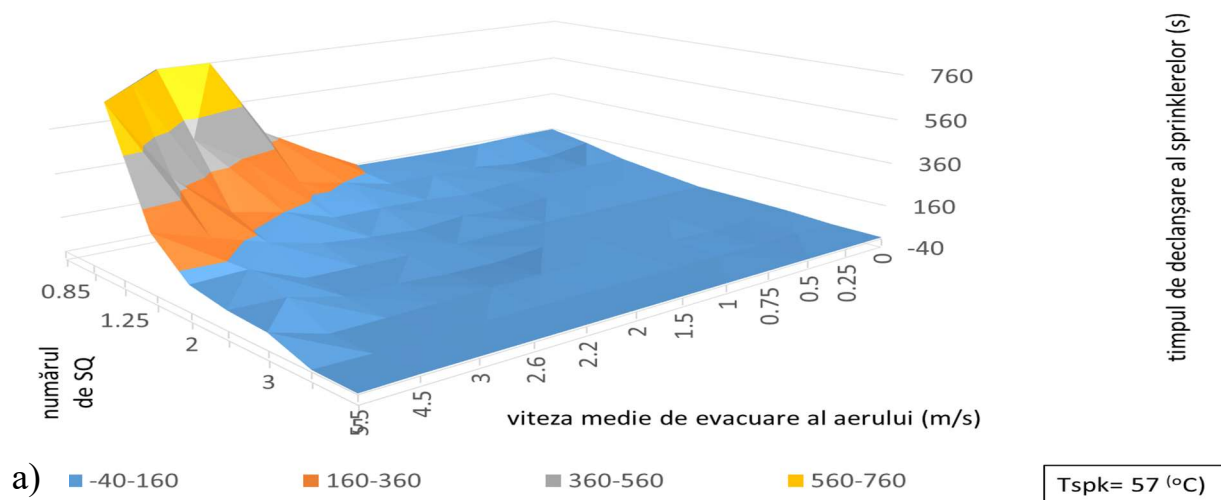


Fig. 14. Variation graph of the time at which visibility drops below a limit value, depending on the number of hourly shifts, for a given heat load value.

Based on the numerical models, graphs have been established that connect the average exhaust air speed (fan flow rate) to the time at which a certain temperature is reached, leading to the triggering of the sprinklers.

Considering the known values for the triggering times of the 4 types of sprinklers studied and their dependence on HRRPUA/SQ and on the average air exhaust velocity/number of hourly shifts, spatial graphs could be drawn defining the phenomenon. In the following 4 pictures, the phenomenon is observed for the 4 types of sprinklers studied.



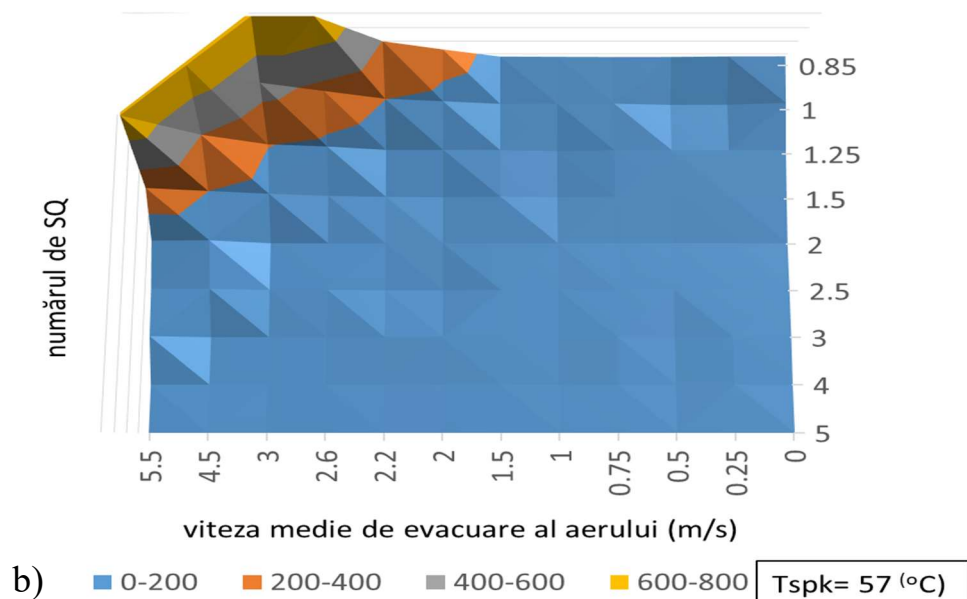
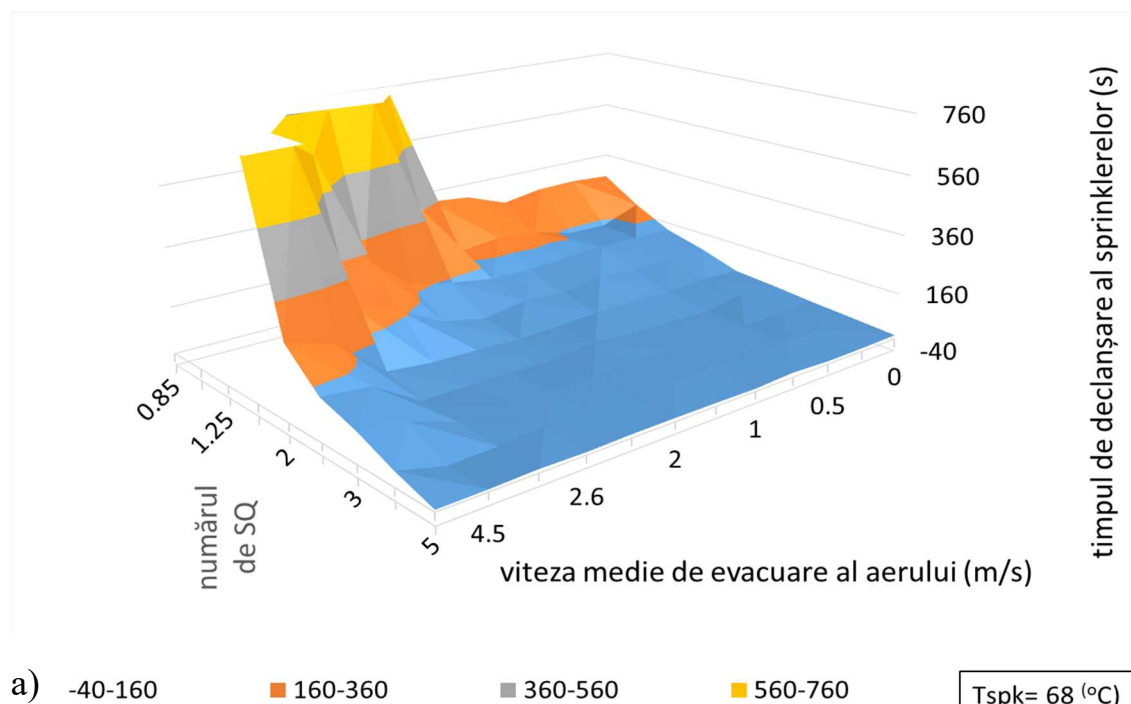


Fig. 15. The evolution graph of the sprinkler triggering time $T_{spk} = 57(^{\circ}\text{C})$ depending on the heat load and the average air exhaust rate – view under two different angles.



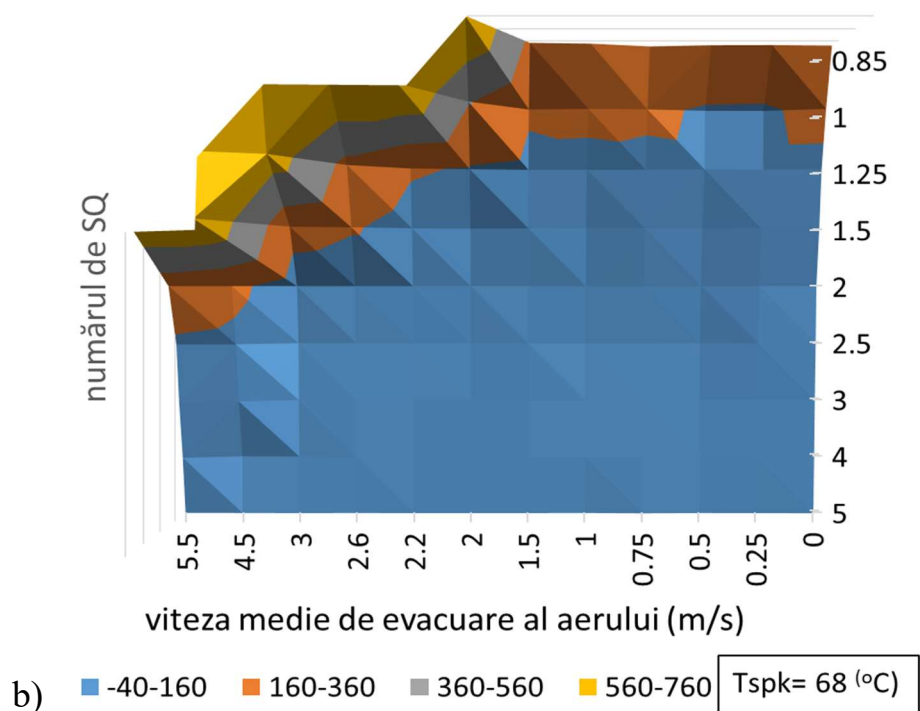
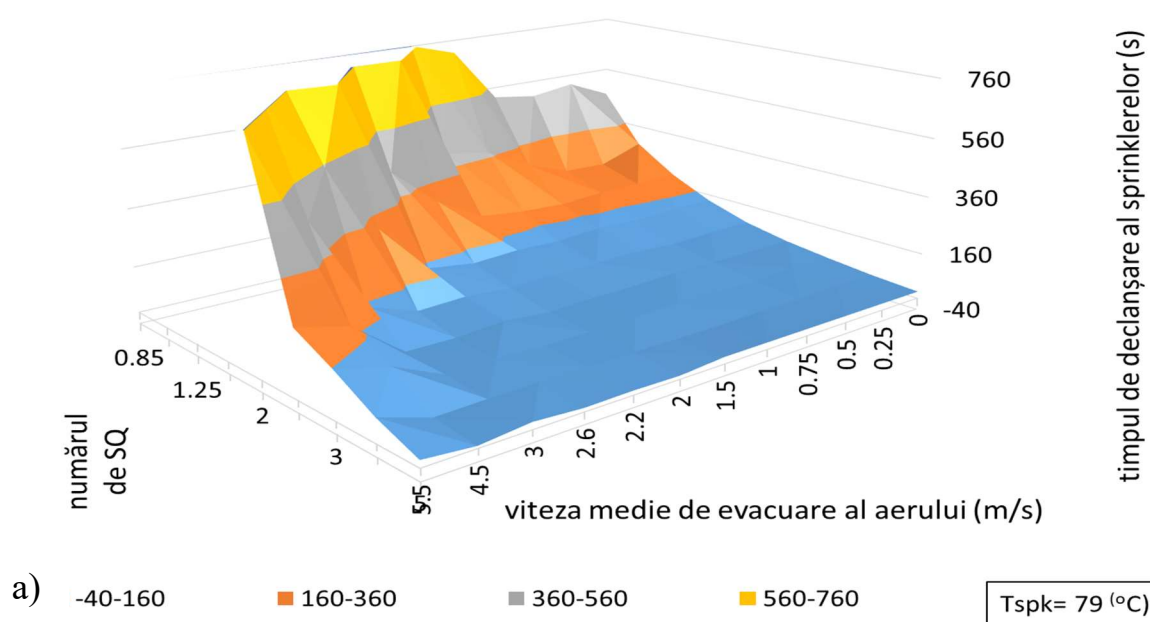


Fig. 16. The evolution graph of the sprinkler triggering time $T_{spk} = 68$ (°C) depending on the heat load and the average air exhaust rate – view under two different angles.



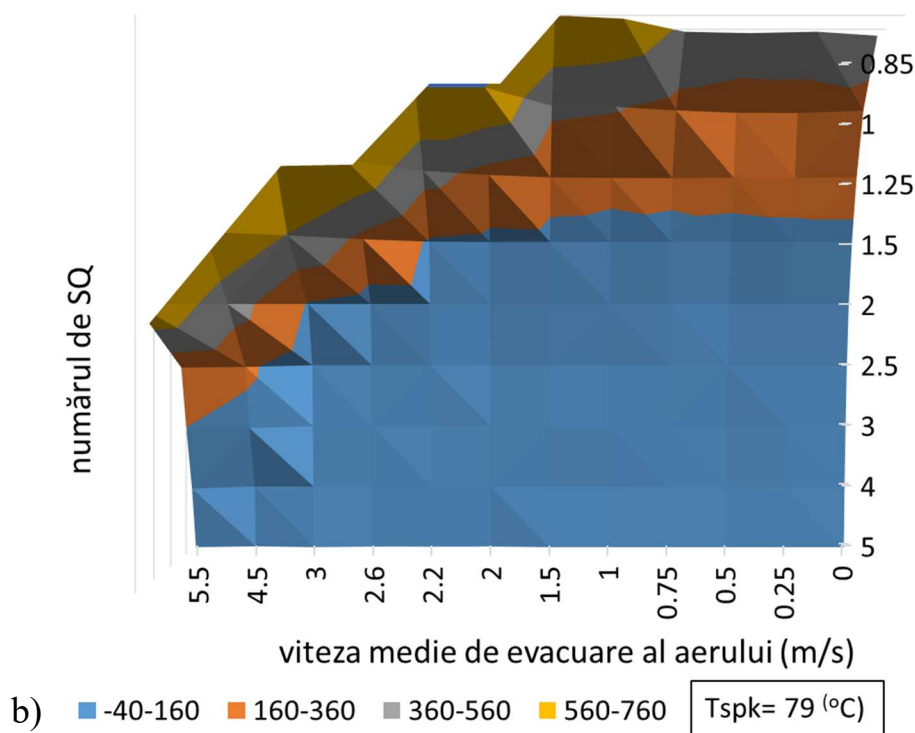
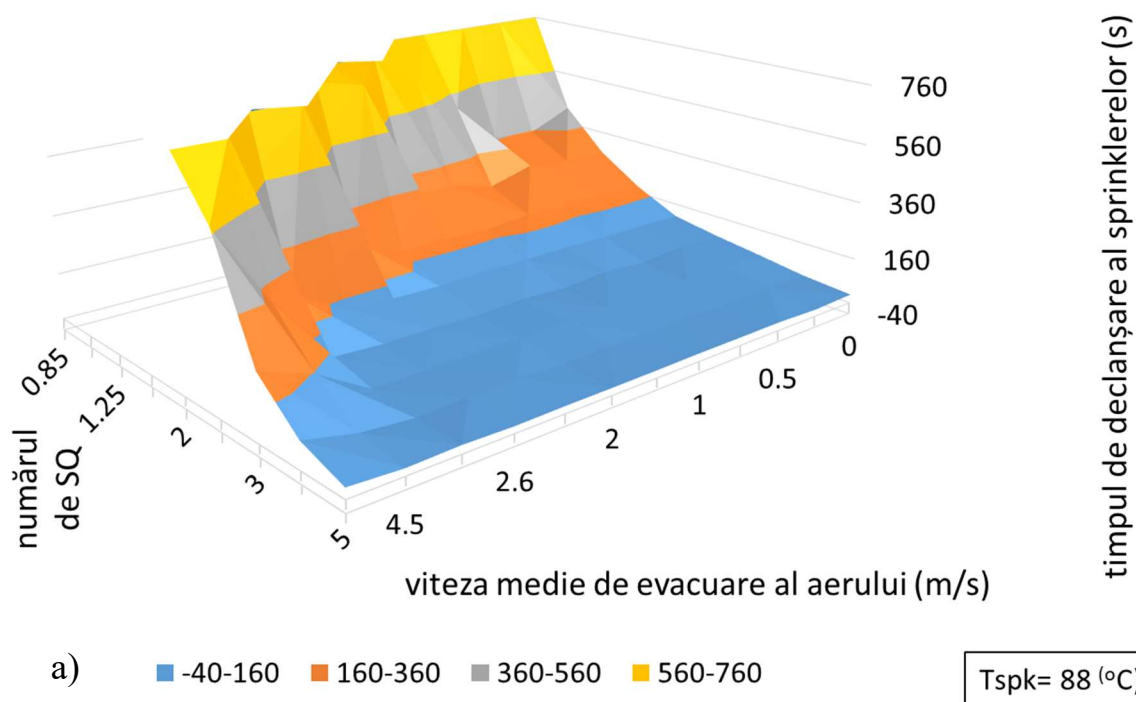


Fig. 17. The evolution graph of the sprinkler triggering time $T_{spk} = 79$ (°C) depending on the heat load and the average air exhaust rate – view under two different angles.



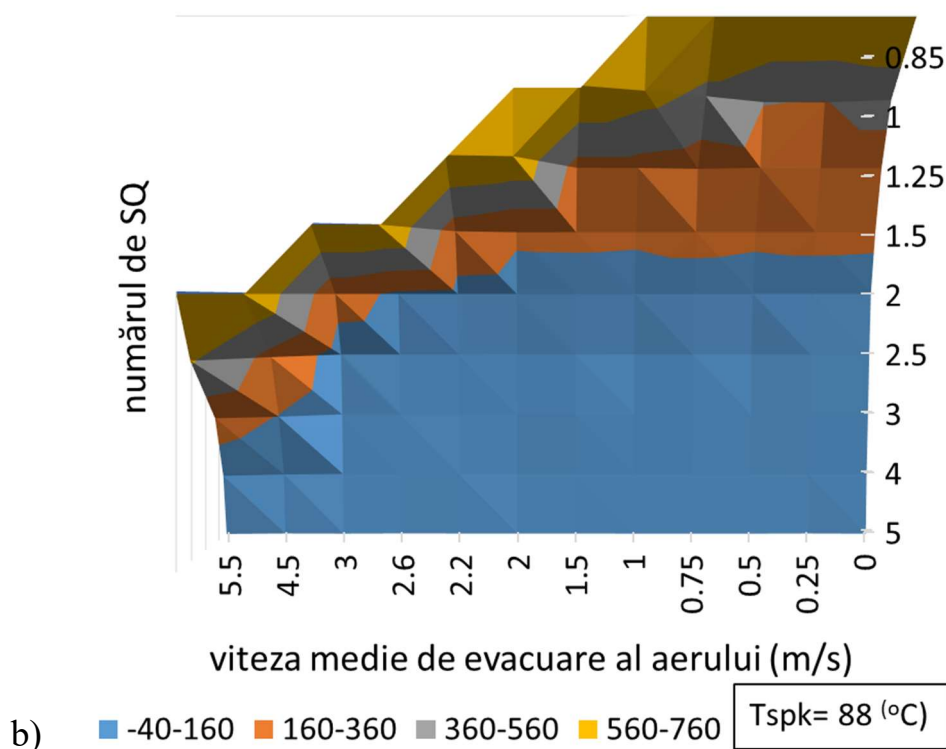


Fig. 18. The evolution graph of the sprinkler triggering time $T_{spk} = 88$ (°C) depending on the heat load and the average air exhaust rate – view under two different angles.

This research thesis takes a step forward and, using the numerical simulation, has established mathematical graphs and functions that relate the triggering times of the heat load sprinklers on the one hand and the ventilation flow rate on the other hand.

Several scientific papers were published during the research period.

In *E3S Web of Conferences*, EENVIRO 2017, the article "*Fire modeling in a nonventilated corridor*" was published. The article studies how fire evolves inside a corridor that is not ventilated by means of the parameters: temperature, oxygen concentration and visibility. Co-authors of this article are Professor Engineer Vlad Iordache PhD and Lecturer Engineer Ilinca Năstase PhD.

The article entitled "*Simularea incendiilor prin utilizarea a trei metode diferite de introducere a fluxului de căldură (HRR-Heat Release Rate)*" was published in the *Romanian Journal of Civil Engineering*. Co-authors of this article are Professor Engineer Vlad Iordache and Lecturer Engineer Ilinca Năstase PhD. The article analyzes the influence of the HRR

introduction on indoor temperatures, considering that the total value is simulated in all three cases analyzed.

At the annual conference of the doctoral school organized by UTCB in 2019, the article *"Numerical and experimental study to determine the thermal conductivity of an envelope for a room set on fire"* was published. Co-authors of this article are Professor Engineer Vlad Iordache and Lecturer Engineer Ilinca Năstase PhD. The article shows how the equivalent heat conductivity of the experimental booth envelope was determined. Data obtained from experimental tests and values determined by numerical modeling were used.

The article *"Experimental and Numerical Study on the Influence of Mechanical Ventilation on the Survival Conditions in a Room Set on Fire"* was submitted for analysis to the *Fire Safety Journal*. Co-authors of this article are Professor Engineer Vlad Iordache and Lecturer Engineer Ilinca Năstase PhD. The article investigates how the mechanical ventilation system influences the survival and evacuation conditions of users of a space subject to fire action.

The article entitled *"Experimental and numerical study on the performance of different fire ventilation systems"* was published in the *Romanian Journal of Civil Engineering*. In this article the performance of different types of fire ventilation systems is studied.

The article *"Ventilation on the Survival Conditions in a Room Set on Fire"* was submitted for publication in the journal *Mathematical Modelling in Civil Engineering*. The study conducts an analysis of how the ventilation rates of a space on fire affect the evacuation and survival conditions of the users.

5. CONCLUSIONS

The main purpose of this research was to analyze how different installations with a role in fire safety influence each other. In this case, it concerns the automatic ventilation system for the exhaust of hot air charged with noxious gases, flue gases and smoke and the automatic water sprinkler system. It was observed how the two types influence each other, but also how the minimum parameters necessary to ensure the survival people in case of fire change.

The influence of the ventilation system on automatic sprinkler systems is found to be significant.

What this research adds is the establishment of graphs and polynomial functions of dependence of parameters describing the fire, such as indoor temperatures, oxygen concentration, concentration of carbon dioxide and visibility of HRRPUA (the amount of energy introduced into the system) and the number of hourly shifts for certain reference intervals and under given conditions. It has been established in the present study that polynomial functions best describe this dependence.

Further research is needed to establish a correlation between how the two types of installations operate. An automation allowing the instantaneous changes of the air flow rate according to the indoor temperatures so as to allow the sprinkler installations to be switched on for fire containment and suppression is to be analyzed.

At present, the design rules simply relate the ventilation flow rate only to the surface of the space that is protected by the mechanical ventilation system. The research shows that this simplified link is not sufficient, as the parameters that define the fire depend on the heat load, i.e. the quantity and type of combustible material in the space subject to fire action.

Future research should find a method for determining the characteristics and mode of operation of the mechanical ventilation system that takes into account not only the surface of the area but all its geometric characteristics as well as the quantity and type of combustible material existing in the space subject to fire action.