PROCEEDINGS, Fire and Evacuation Modeling Technical Conference (FEMTC) 2018
Gaithersburg, Maryland, October 1-3, 2018

EFFICIENCY OF THE FIRE SAFETY SYSTEM, BASED ON FIRE DYNAMICS SIMULATOR (FDS)

Wilson Nobre Lima
Aristides Lopes da Silva

University of Cape Verde
Department of Science and Technology
Palmarejo, Praia CP 279, Cabo Verde
e-mail: wilson.nlima@hotmail.com | aristides.silva@docente.unicv.edu.cv

ABSTRACT

The efficiency of a fire safety system constitutes a concern to public and technologically advanced buildings. Still today, real simulations and, exercises are used in order to analyze, evaluate and validate these systems' efficiency. Considering that these exercises entail expensive simulation costs, this work focuses on designing a virtual model which is based on Computational Fluid Dynamics 3D, that allow for the analysis of the efficiency of the fire safety system and decision-making for emergencies in the Cape Verde Data Center managed by Operational Nucleus for Information Society (NOSi).

This article reports the results of a simulation scenario based on the Fire Dynamics Simulator (FDS) code, showing that this model would significantly influence decision making, particularly in the case of a fire, which would avoid expensive costs and possible damage in a real simulation.

Keywords: Fire, Virtual Simulation, Data Center, CFD, FDS

INTRODUCTION

With the emergence of technologically sophisticated public buildings, security becomes such a critical issue that it must be set up, with practical and executable solutions, primarily for fire events. The concern for occupant safety and existing technologies in the building is a problem that deserves strong attention from researchers.

The risk of fire is present, practically in all places built and frequented by humans. Therefore, it is important to adopt measurements to prevent fires and to guarantee the safety of people in buildings. Both the equipment and facilities are at high fire risk, and must be designed and built so that, in case of fire, they can be easily isolated. [1]

Fire in Data Centers, are events that commonly occur due to high temperatures from existing machinery operating within these Data Centers. As an example of one of these events, we can consider the fire that occurred in the Samsung datacenter on April 2014, where there was an interruption of data for several hours before being restored, making the service of Samsung devices stop. [6][3]

In this regard, the present work consists of designing a model, based on the Fire Dynamics Simulator, which allows for analyzing the efficiency of the fire safety system of the Data Center of Cape Verde, examining temperature predictions and local pressure measurements.

The results of this study propose that this model would significantly influence decision-making, particularly in the case of a fire within the safe room, which would avoid expensive costs and possible damage in a real simulation. Likewise, the results of the grid sensitivity analysis can be used to enhance the performance on future simulations, saving rendering time and reducing the computational effort.

PROJECT METHODOLOGY

The methodology was based on modeling processes regularly used. First, the geometric model was designed and subsequently the parameters for results extraction were added. Fig.1, shows briefly the methodology used.

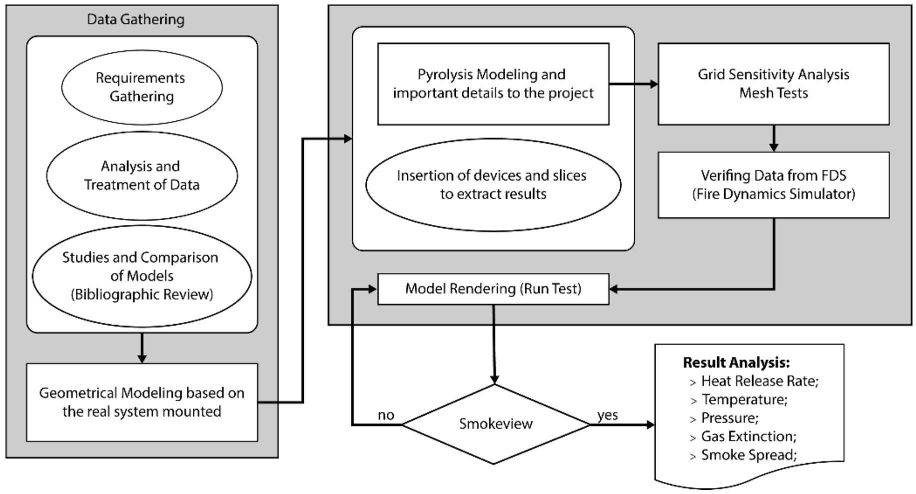

Fig. 1 - Methodology Fluxogram

Grid Sensitivity Analysis

In the Large Eddy Simulation (LES)[1] it is not possible to obtain perfect independence of cells, although the slight variations that can be theoretically found between cells are tolerable. [2]

Having said that and giving emphasis to the grid sensitivity tests, the data analyses strongly influence the model results, as well as the computer performance.

For this grid sensitivity analysis, only the mesh of the safe room was examined (see Fig. 6), as the area of greatest turbulence due to the fire source location. Five tests were performed and analyzed: three with cell dimensions of 0.20 m, 0.16 m and 0.12 m, another one with a dimension of 0.12 m (for obtaining different cells divisions, having discrepant results among themselves) and a last one of 0.08m. A simulation was performed, initially with a fire power of 300kW/m^2, with a simulation time of 50s and a time step estimated by the software of 0.14s.

Tab.1 shows the temperature variations close to the plume and Fig. 2 presents the temperature trends at 30s of simulation measured by the devices, according to the different cells dimensions analyzed.

[1] **LES -** numerical technique for integrating equations filtered from the movement that describes the time evolution of high-Reynolds number and three-dimensional turbulence. [4]

Tab. 1 - Temperature close to the fire source

Time (s)	Temperature (ºC)				
	0.20 m	0.16 m	0.12 m	0.12 m	0.08 m
10	21.0	22.0	22.5	24.5	24.5
20	14.0	14.5	13.8	8.4	15.0
30	7.0	9.0	12.0	13.0	12.5
40	8.5	7.0	9.0	12.0	12.0
50	5.0	5.5	5.0	28.0	15.5

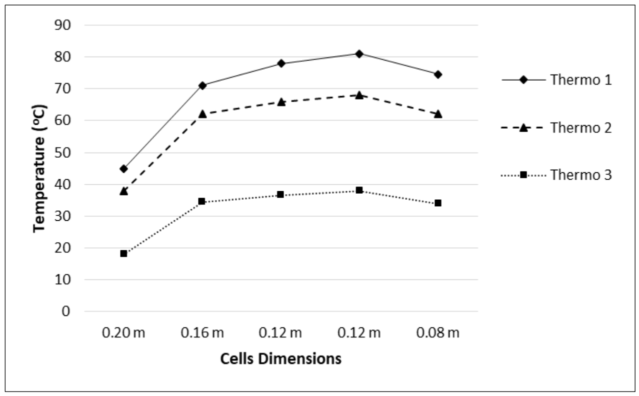

Fig. 2 - Temperature variations at 30s of simulation,
See device mapping for understanding the positioning, Fig.7 and 8

It can be observed that, in these trends, there is a convergence of values, regardless of the placement of the referred devices.

The machine used is a server with the following characteristics: Windows System Server 2012 R2 Standard based on a 64bit, Intel(R) Xeon(R) CPU X5675 @ 3.07GHz, processor with 16GB of RAM.

From the results, we can see that the CPU load is constantly running at maximum capacity, ranging from 98% to 100%, while the RAM and the render time have varying performance due to the decrease of cells dimensions, i.e. during the render, the values of these variables tend to increase when the cell dimensions are reduced. These oscillations occur between 30 minutes and 4 hours of rendering, having registered a case where there is an unexpected increase in time of 13:28:37 achieved by the cell with the dimension of 0.08 m.

It should be noted that this average was calculated based on the results from the resource monitor of the computer used. Tasks and processes that cannot be turned off for proper operation of the machine should be considered.

You can still see improvements in the performance of the machine, using two meshes with cell dimensions of 0.12 m, by varying the cells divisions for each mesh, maximizing computer performance. Tab. 2 present the cell divisions, while Fig. 3 shows the gains gathered during the rendering.

Tab. 2 - Mesh 0.12m - Cells Divisions

Tests	Cells Divisions		
	X axis	Y axis	Z axis
Test 3	108	45	28
Test 4	112	48	27

Fig. 3 presents results comparison of the computer performance during the rendering of different analyzed tests.

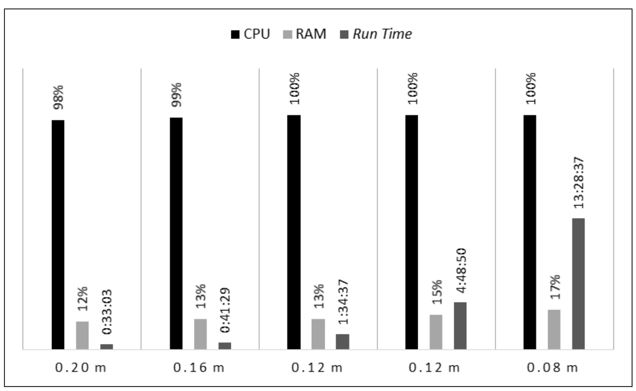

Fig. 3 - Computer Performance during the rendering

Comparing the results obtained, one can see that the cell dimensions of 0.20 m and 0.16 m have better performance in relation to the others, due to having the least render time, RAM and CPU usage. Based on the results, Fig. 3 and Fig. 4 show that the cell dimension of 0.16 m is the optimal solution found.

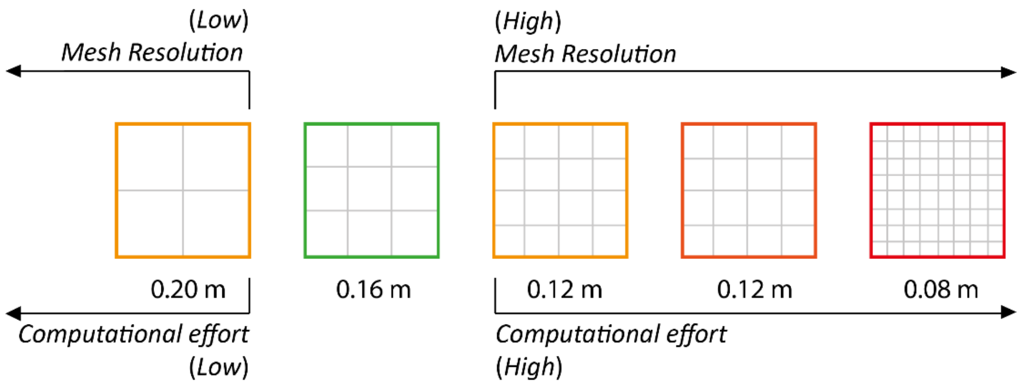

Fig. 4 - Best Resolution Found

FDS, NUMERICAL SIMULATION

Physical Model

This work focused on the study of the Data Center in Cabo Verde, a technological infrastructure that hosts equipment for processing and storing State data with strong potential ability to provide services to other national and international entities, Fig. 5.

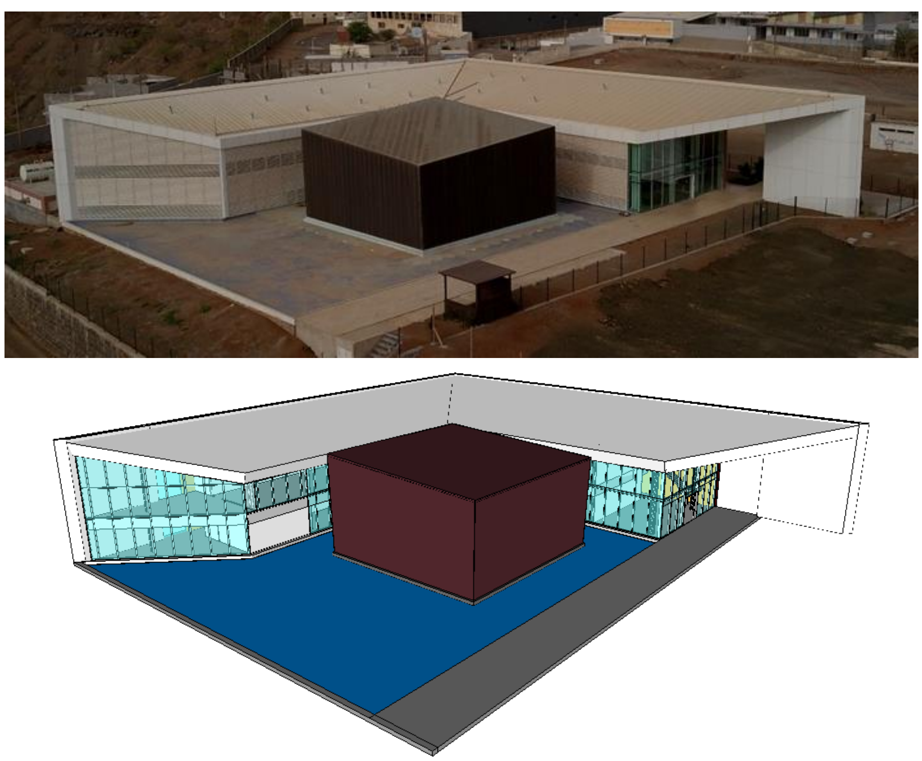

Fig. 5 - Geometric model under study, (real picture - FDS Model)

The construction of this building is based on a steel structure and is divided into two parts, being:

- The Network Operating Center (NOC) is part of the L-shaped building, which houses the team responsible for managing the Data Center, that is, it provides technical support to the building and equipment contained therein;
- The data center (The Cube) is the safest part of the building and houses the technologies of data storage and processing. The Cube comprises two rooms: the main room and the safe room;

Security becomes of great concern, because of the integrity of the data stored and the operation of equipment, since there are risks such as fire, increased indoor temperature, humidity, and electrical overload, among others. Having said that, the need for decision-making for certain cases depend on real or virtual simulations. The virtual simulations often stand as the most viable option, since real simulations can be expensive.

<u>Simulation Scenario</u>

The interior of the Cube, in the Safe Room specifically, was the subject of the scenario, which was initially designed with a fire power of 300kW/m^2, originating from a short circuit in one of the UPSs (Uninterruptible Power Source). It was examined through INIT REGIONS, with an initial environment temperature of 22^{o}C within the Cube and 26^{o}C outside of it, in a 30s simulation.

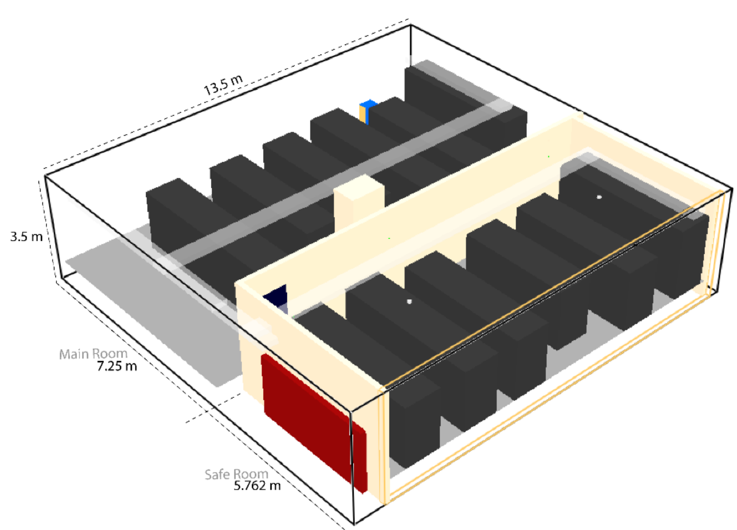

Fig. 6 - Cube Geometry and mesh dimensions

The model was designed according to the existing fire safety system, based on IG-55 extinction gas. To model the gas IG-55, parameters were set with mass fractions distributed at 0.5 to argon (Ar) and at 0.5 to nitrogen (N2), being an Inert gas composed of 50% of each of these chemical elements. To inject the gas into the space, a surface type supply was used providing an airflow of 0.5 m³/s, replacing the nozzles. The injection of particles was represented by small dots of grey color, allowing the visualization of the dynamics of gas into the space.

To extract the results, devices were distributed accordingly to the existing system and additional devices were inserted to extract results. Fig. 7 and 8 show the device mapping, as well as Tab. 3 which presents the nomenclature and labels used.

Tab. 3 - Devices (Nomenclature and Labels)

Devices	Thermocouple	Gas-Phase	Supply (Nozzles)	Smoke Detectors	Heat Detectors
	Thermo_	Pre_	NC_SR	SD(C/F)_SR	HeatD_
Label					

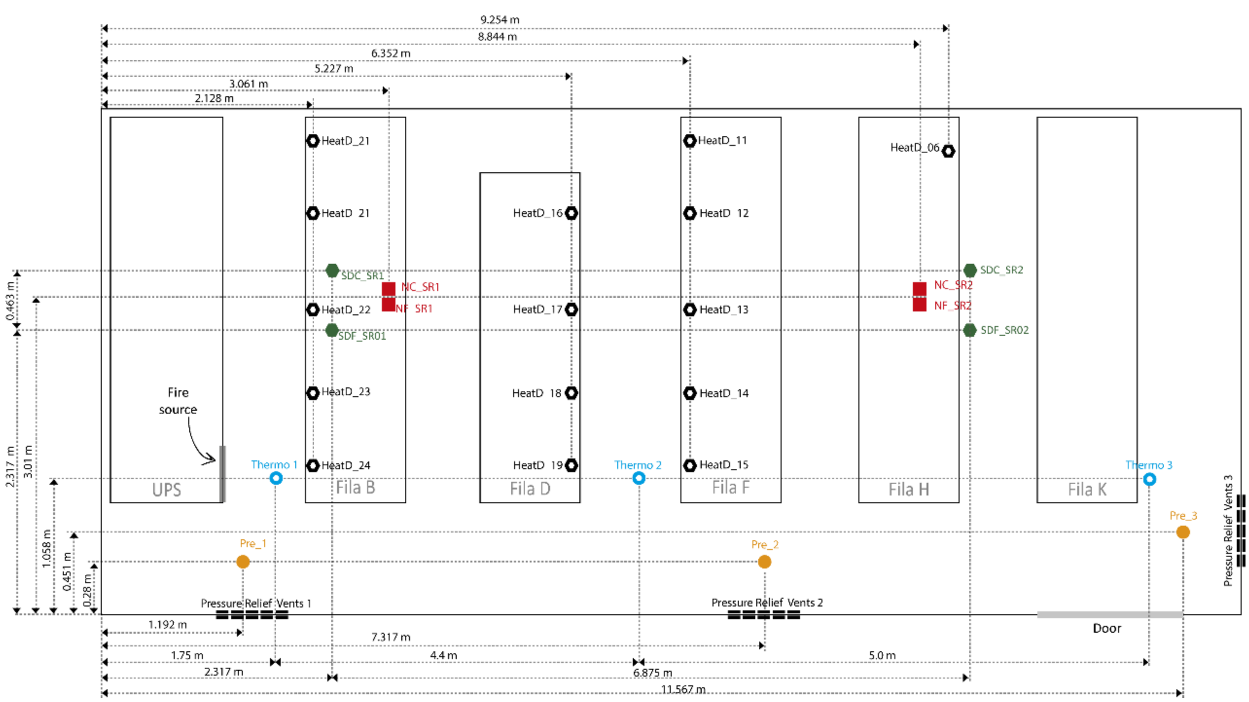

Fig. 7 - Device Mapping - Top View

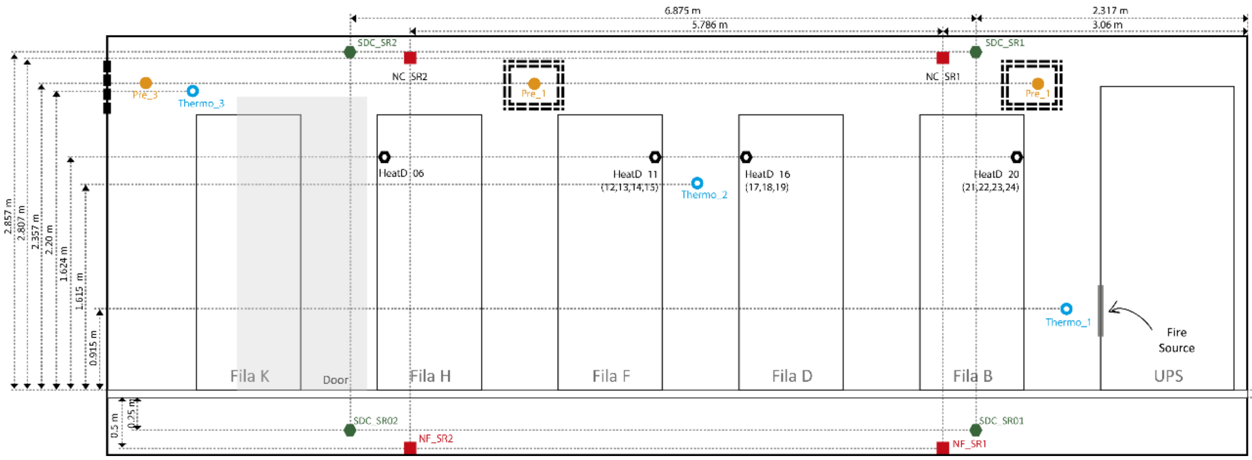

Fig. 8 - Device Mapping - Side View

The pressurization system was also examined in the model where the pressure relief vents trigger according to devices. To achieve this, holes were modeled, that only activate when the pressure value is equal to or greater than 270Pa measured by the gas-phase device. [7]

RESULT ANALYSIS

Note that the mesh size of 0.16 m and the time step of 0.14 s, used in this model can influence the results. Thus, it becomes crucial to review these aspects for future studies that will be conducted on this model.

Regarding the temperature, initially in the safe room it was 22°C. In the middle of the simulation, swings in temperature can be noted, caused by the spread of smoke. Theoretically larger temperature values was expected close to the fire source, although the different levels of positioning devices, Thermo 1, 2 and 3 have influenced these measurements. It can be justified by one of the principles of thermodynamics, the principle of convection, where upper areas have higher temperatures and lower areas have lower temperatures.

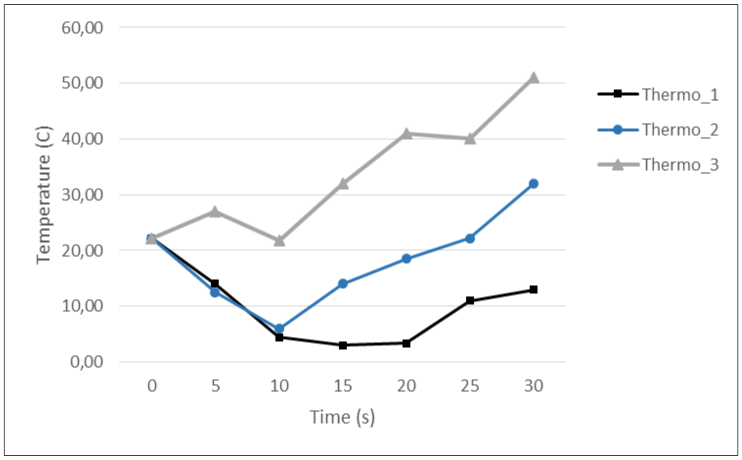

Fig. 9 - Temperature Measurements from Thermocouples

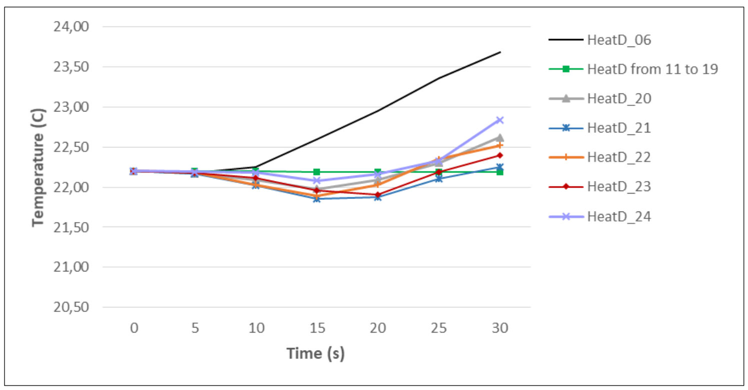

Fig. 10 - Temperature Measurements from Heat Detectors

In relation to the temperature analyses, from the graphs above, it can be seen that there is an increasing temperature trend at these specific points, and these results can be used as benchmarks for future studies and analysis.

Analyzing the spread of smoke, and gas activation, there is evidence of consistent results, signifying that as a research tool the model is useful in some situations, in particular fire situations.

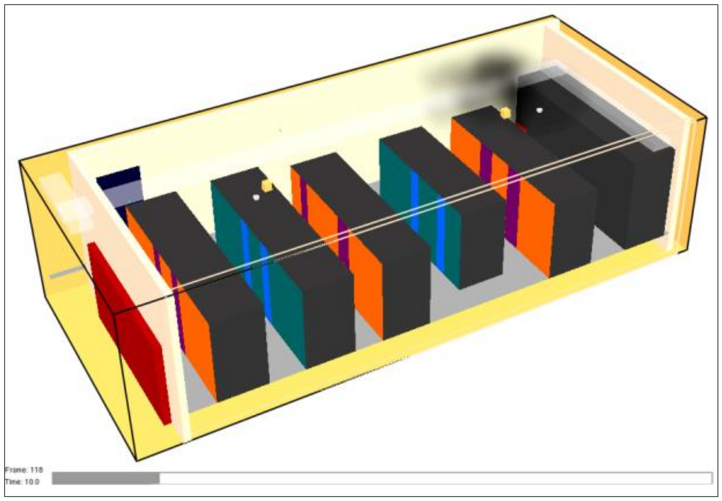

Fig. 11 - Smoke Spread at 10s

Fig. 12, shows the controller graphic indicating the activation time when the gas was injected into the safe room. This allows the activation of the gas by measuring obscurity by smoke detectors, specifically, the controller works after obscuration reaches the smoke detectors, where an obscuration threshold value of 3.31%/m was established as is seen in Fig. 13. Therefore, the activation of the controller occured at 12.3s. See Fig. 12.

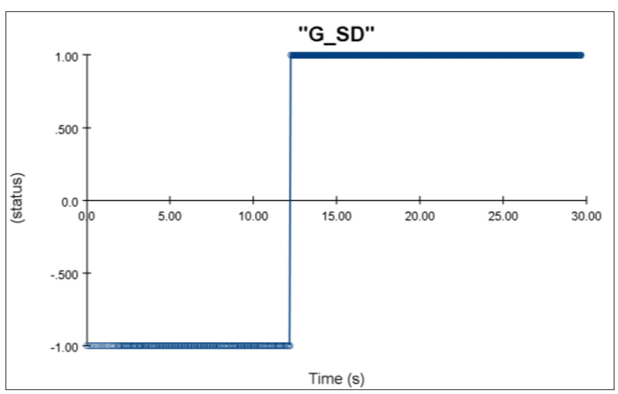

Fig. 12 - Control Plot, IG 55 Gas activation

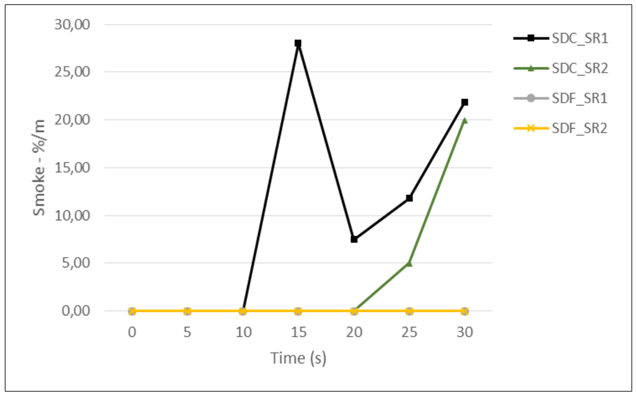

Fig. 13 - Smoke Detector Measurements

A perspective of gas injection is shown in Fig. 14 and Fig. 15 shows the variation of the heat release rate over the course of the simulation. Note that this rate remains almost constant during simulation, and it tends to decrease from 28s until the end.

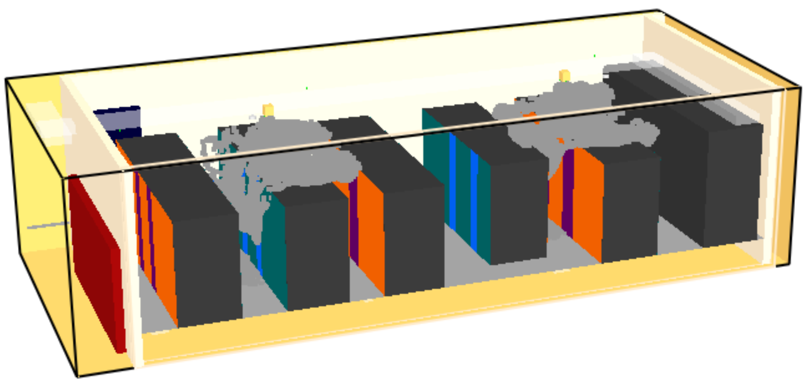

Fig. 14 - IG 55 Gas Injection into the Safe Room

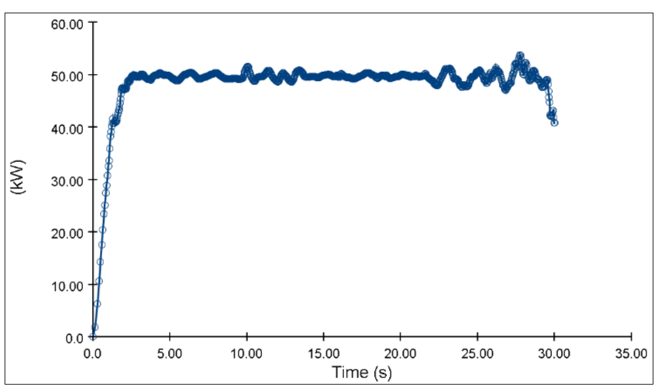

Fig. 15 - Heat Release Rate (HRR Plot)

According to a comparative study between a real simulation and a model based on the FDS, it is shown that the results obtained from the FDS overpredict the results from a real simulation, meaning that a discrepancy of virtual results at the beginning of the simulation was found. [5]

Regarding local pressure, theoretically, an increase in pressure after the gas injection was expected, since there was no system of pressurization parsed inside the Cube. Three gas phase devices were introduced, according to Fig. 7 and 8. However, just one of these devices were analyzed due to results similarity.

(a) (b)

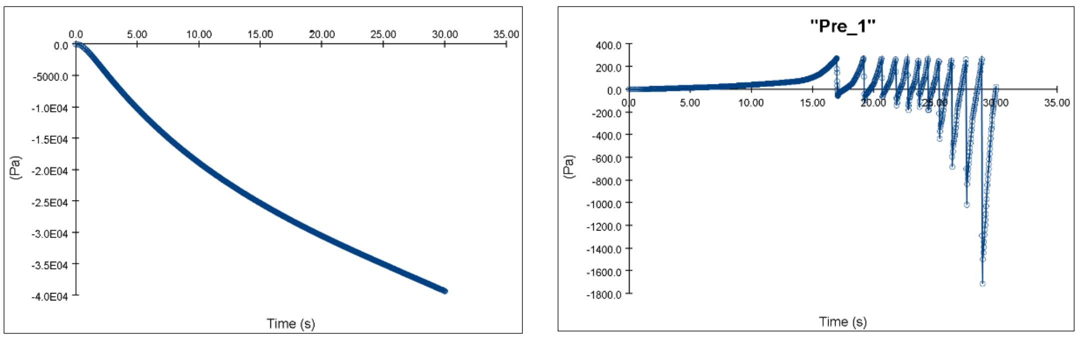

Fig. 16 – Local Pressure after the gas injection
(a) Model with the exhaust activated; (b) model with the exhaust deactivated

The pressure results measured by device Pre_1 is shown in Fig. 16, where in Fig. 16a it shows clearly negative pressure that decreases continuously. Having mentioned that, to gain a better understanding of this result, a scenario was examined in which the exhaust and fans for hot and cold air were disabled. According to this model, there are some concerns related to internal pressure, since the pressure results shows inverse values, with some expected oscillations after the activation of pressure relief vents. Analyzing the results, there was an increase of pressure at the beginning of the simulation and at approximately 18s there was a pressure drop, accompanied by oscillations until the end of the simulation. See Fig. 16b. This can be justified based on Bernoulli's principle, where the sum of the kinetic energies (velocity), pressure flow (pressure) and potential is a constant, meaning, the negative pressure (Fig. 16a), can be assumed based on a velocity increase in areas where the GAS-PHASE DEVICES are, offsetting the depression and keeping the sum value constant. This increase in velocity is justified by the movement of air caused by machinery.

In order to explain the negative pressure in Fig. 16a, a velocity vector slice is shown in Fig. 17.

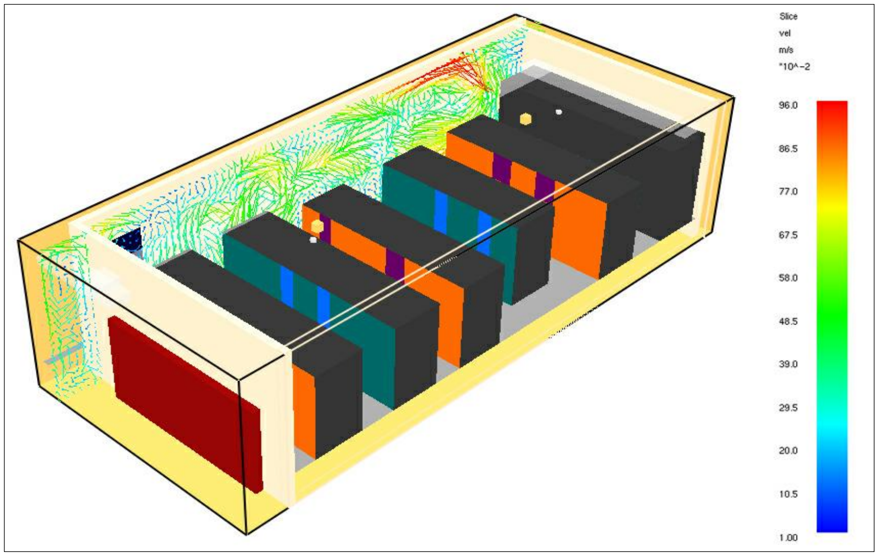

Fig. 17 – Velocity Slice (Vectorial Slice)

CONCLUSION

The model proposed in this study allows decision-making in some security cases, particularly fire cases in the safe room, avoiding costs and possible damage caused by a real simulation. Likewise, the results of the grid sensitivity analysis can be used to enhance performance in future simulations, saving rendering time and reducing computational effort. In addition, temperature and pressure were numerically analyzed based on the designed model, where the pressure results oscillate alarmingly, increasing the need for further studies, explaining the influence of the cooling system on the inner pressure.

The results analysis from smoke detectors were predictable and justified by their placements, which ultimately confirm the consistency of the designed model regarding the actual system installed.

FUTURE WORKS

Although this model responds positively to the concerns and objectives previously mentioned, however, in order to give continuity to the validation of this model, further work is recommended, particularly, comparison of experimental data with numerical simulations to introduce significant improvements in solutions for future needs in this field.

Based on the methodology used, we can take advantage, of grid sensitivity analysis in future work, comparing the results, improving the machine performance, and therefore get better results in future models.

Another possible future work would be to design a model based on a Pathfinder and EVAC module from FDS that allows for the analysis of the building structure and influences decision making for emergency cases, especially evacuation, in this data center.

ACKNOWLEDGEMENTS

This research is part of a study supported by Department of Science and Technology at the University of Cape Verde and Operational Nucleus for Information Society.

REFERENCES

[1] Costa, A. D. (2009). Meios de Extinção de Incêndio: Extintores Portáteis. Porto, Portugal: Faculdade de Engenharia da Universidade do Porto (FEUP).

[2] McGrattan, K., Baum, H. R., & Rehm, R. (1998). Large eddy simulations of smoke movement. Scientific Article. Fire Safety Journal.

[3] Miller, R. (April 20th, 2014). Data Center Fire Leads to Outage for Samsung Devices. Journal Post. Obtained from Data Center Knowledge: http://www.datacenterknowledge.com

[4] Moeng, C., & Sullivan, P. (2015). Large-Eddy Simulation. Scientific Article, 2nd Edition. Boulder, Colorado, United States of America: Elsevier: Encyclopedia of Atmospheric Sciences.

[5] Silva, A. L., Xiong, S., & Aamir, H. (2015). Effect of Different Air Settings Over Fire-induced Condition in a Stairwell. Scientific Article. United Kingdom: Journal of Structural Fire Engineering (JSFE).

[6] Sombers Associates Incorporation & Highleyman W. H. (2014). Fire Knocks Out Samsung. Obtained from http://www.availabilitydigest.com/

[7] Swenson, D. (18th of November, 2014). Modeling a Pressure Relief Vent. Obtained from Thunderhead Engineering: https://www.thunderheadeng.com/2014/11/modeling-pressure-relief-vent/