https://doi.org/10.24840/978-989-54863-4-2_0347-0350

SMOKE CONTROL PROCESS IN AN UNDERGROUND PARKING LOT – CFD SIMULATION

Ana Carneiro[1], João Silva[2], Nelson Rodrigues[3], Senhorinha Teixeira[4], Pedro Lobarinhas[5]
[1] Mechanical Engineering Department, University of Minho, Campus de Azurém, Guimarães, Portugal; a85203alunos.uminho.pt; ORCID 0000-0003-1104-6170
[2] Mechanical Engineering Department, University of Minho, Guimarães, Portugal ; js@dem.uminho.pt ; ORCID 0000-0003-3518-7111
[3] University of Minho, Mechanical Engineering, Guimarães, Portugal ; nrodrigues@dem.uminho.pt; ORCID 0000-0002-9394-1357
[4] ALGORITMI, School of Engineering, University of Minho, Guimarães, Portugal; st@dps.uminho.pt; ORCID 0000-0002-7464-3944
[5] MEtRICs, School of Engineering, University of Minho, Guimarães, Portugal; pl@dem.uminho.pt; ORCID 0000-0002-1777-2061

Abstract
Background: Car parks present characteristics that can affect the evacuation process in case of a fire. To improve fire safety in these spaces, it is important to provide these areas with systems that control/remove the smoke released by the fire. **Objective**: The focus of this paper is the study of fire safety in underground car parks. Its purpose is to simulate a fire scenario in a car park, so to understand the behaviour of the designed smoke control system. To predict airflow behaviour and the temperature variation when a fire is developing, a Computational Fluid Dynamics simulation was performed using Ansys Fluent software. The paper also presents a brief description of the car park studied and explains the approach used to represent the fire scenario. It was verified that the system is efficient because the fire spreading is confined to the heat source location, even at the maximum reached temperature. The simulation results allowed the analysis of the velocity and temperature contours, during the development of the fire.

Keywords: Fire Safety, Underground Car Park, Smoke Control, CFD Simulation

Introduction

Nowadays, the number of vehicles entering cities is increasing, which causes various problems, specifically organizational problems. As it is necessary to find places to park these vehicles, one of the solutions found was to construct underground car parks. Usually, these spaces are under commercial or habitational buildings to promote good accessibility and accommodation to the users. These constructions are usually completely covered and in case of a fire, this aspect has a great impact on the smoke control/removal because of the large smoke release and the rapid rise of temperature. The main purpose of a smoke control system is to facilitate the evacuation of the occupants present in the space. Therefore, various studies should be performed to avoid complications during the evacuation process. In Portugal, there is legislation in place that must be followed during the design of the smoke control systems. The directive in question is *Portaria nº 135-2020*. This regulation states that all the underground, totally covered car parks must use an active smoke control system, which means that mechanical equipment must be used to perform the smoke control, as it is impossible to do so naturally. As Kallianiotis et al. (2022) state, mechanical ventilators can reduce the smoke effects significantly, which is a priority in case of a fire. The objective of this study is to simulate a fire event in an underground car park in order to predict the behaviour of the smoke control system installed in the car park.

Material and Methods

The subject of this study is an underground car park, totally covered, that has an insufflation zone and an extraction zone. The insufflation and extraction of air are only possible due to the presence of two ventilators. This park also has two induction ventilators, one in the centre and the other on the right upper side, that direct the airflow to a certain direction, near the extraction area. In Figure 1 the insufflation ventilator is represented by the inlet area and the extraction ventilator by the outlet. In yellow are indicated the two induction ventilators. The layout presented in Figure 1 shows the geometry and the divisions to generate the mesh and represents an approximation, as all the doors and gates are considered walls. This is due to the fact that when the system is activated, all those components close to prevent the fire from spreading.

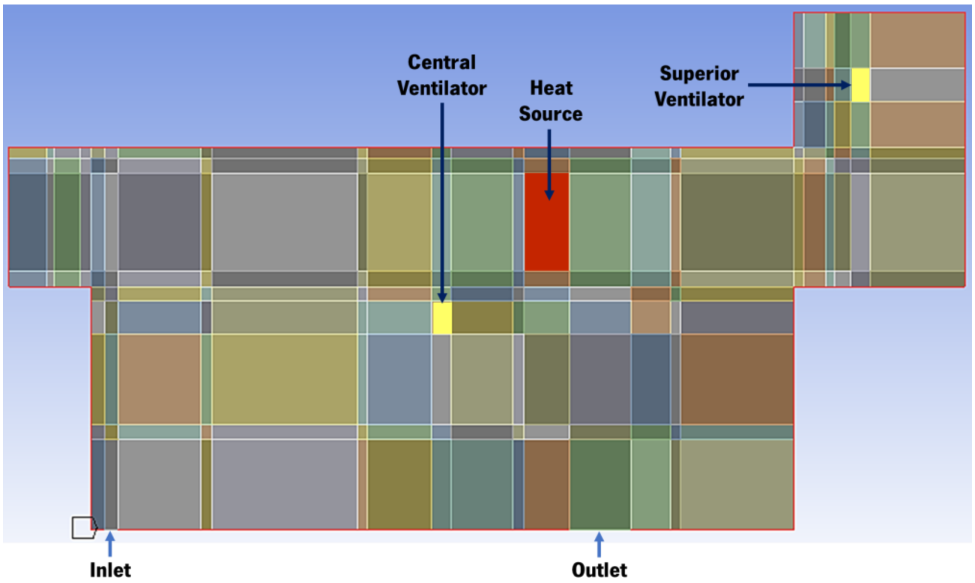
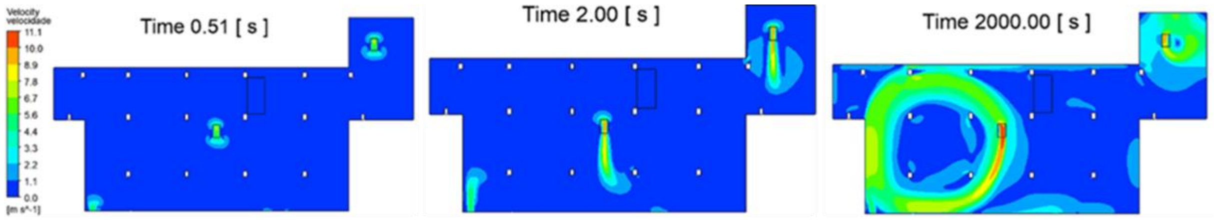


Figure 1. *Boundary conditions for the geometry of the car park and ventilators and heat source locations.*

Concerning pollution points, the directive previously mentioned considers the number of cars present in the park and so the extraction rate is $600m^3$/hora/vehicle. The worst-case scenario was considered, meaning that 20 vehicles were inside the car park when the fire began. The area chosen to represent the fire was the right side of the central induction ventilator, as shown in Figure 1, in red. According to Bernardo & Pessoa, 2022, the development of a fire can be divided into three stages: ignition, heating and cooling. Therefore, three time intervals were selected in order to define the phases mentioned. A study performed by Merci & Shipp (2013) presents a value for the heat release rate for one car burning. Using that data, a function that indicates the heat generation due to the fire was created and implemented in the simulation. The CFD simulation was performed using Ansys Fluent software. According to Barsim et al. (2020), the turbulence model k-ε can predict turbulent flow behaviour caused by fires so, the turbulence model chosen was the k-ε Realizable, as it is more adequate to relatively complex flows (ANSYS Inc., 2022). The energy model was activated to simulate thermal variations due to the fire. As the study intends to evaluate changes over time, the simulation was carried out in a transient state. The first simulation step is the drawing of the geometry. It is important to mention that it is simplified in 2D, and the induction ventilators entrance of air occurs through the superior edge and the exit through the inferior one, Figure 1. The mesh divides the whole domain into control volumes. The more refined the mesh, less errors occur during the calculations. However, a very refined mesh requires long calculation times, which is not desirable. A mesh optimization study was performed to encounter a compromise between calculation time and results accuracy. The mesh selected has good quality and calculation times are not very high. Concerning boundary conditions, the inlet was defined as a pressure-inlet and the outlet as a velocity-inlet with a negative velocity. Regarding the induction ventilators, a source term was defined to represent their operation, and because the values needed were on the technical sheet.

Results and discussion

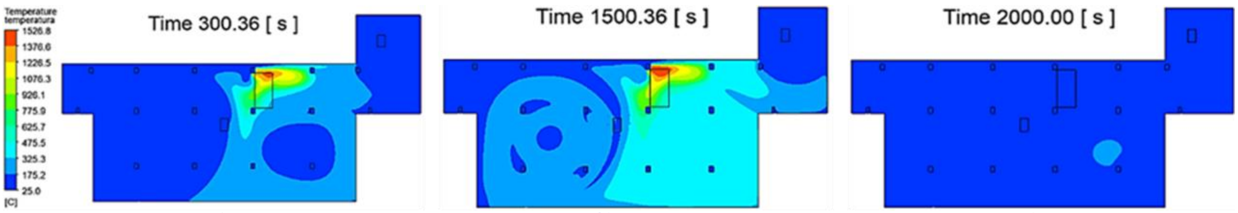
The analysis of the velocity field allows the understanding of the flow behaviour and to distinguish the areas where the air is more stagnated. This means that these areas have almost no air circulating and so they can be considered critical. Figure 2 shows the variation of the velocity field over time.



a) b) c)

Figure 2. *Velocity contours: a) beginning of ventilators operation; b) seconds after beginning of ventilators operation); c) full ventilators operation.*

By evaluating the results obtained, it is possible to see that after the activation of the system the velocity field is constant. In the contour regarding the 2000 s, it is visible that two air recirculations occur, one near the superior ventilator and the other between the air entrance and the central ventilator. This means that the smoke removal will not be facilitated as it is not directed to the extraction area. Furthermore, the existence of stagnation zones will affect the system efficiency if a fire happens in those areas. The temperature contours show the development of the fire and enable the understanding of the system performance. In Figure 3, the fire development is displayed over time.



a) b) c)

Figure 3 *- Temperature contours: a) beginning of the fire; b) maximum development of the fire; c) seconds after the end of the fire.*

As shown in Figure 3 temperature does not rise drastically in one area, which means that the airflow passes near enough the location of the fire and allows the sweeping of some of the smoke. After the fire ends, the air inside the park does not have a high temperature, meaning that the polluted air is extracted.

Conclusions

In this study, a CFD simulation of a fire in an underground car park was performed. This tool proved to be useful to predict the flow behaviour and to draw conclusions regarding the performance of the system. As mentioned, some areas of the park do not have air flowing through them, meaning that, in case of a fire, the smoke control system will not be very efficient in those zones. The smoke control system could be optimized with changes such as the directions to which the induction ventilators direct the air. The number of induction ventilators could also be increased to avoid the existing stagnation zones. Regarding temperature variations, the system can confine the high-temperature area and remove heat rapidly after the extinction of the fire. Future work should be performed to consider other variables and factors that may affect the system's performance. Numerical studies can be a valuable tool to optimize these systems during the design stage as most of the time experimental data is not possible to acquire. The simulation results allow the detection of dangerous situations and show which areas must be improved. The modifications make the system more reliable and thus ensure the safety of the occupants.

Acknowledgements

FCT has supported this work within the R&D Units Project Scope UIDB/00319/2020 (ALGORITMI) and R&D Units Project Scope UIDP/04077/2020 (MEtRICs).

References

ANSYS Inc. (2022). Ansys Fluent Theory Guide. http://www.ansys.com

Barsim, M. M., Bassily, M. A., El-Batsh, H. M., Rihan, Y. A., & Sherif, M. M. (2020). Performance of impulse ventilation system in an underground car park fire: Case study. Journal of Building Engineering, 29. https://doi.org/10.1016/j.jobe.2019.101162

Bernardo, C., & Pessoa, P. (2022). Sodeca, Decflex, Seminários Sodeca, Sistemas mecânicos de controlo de fumo.

Kallianiotis, A., Papakonstantinou, D., Tolias, I. C., & Benardos, A. (2022). Evaluation of fire smoke control in underground space. Underground Space (China), 7(3), 295–310. https://doi.org/10.1016/j.undsp.2021.07.010

Merci, B., & Shipp, M. (2013). Smoke and heat control for fires in large car parks: Lessons learnt from research? Fire Safety Journal, 57, 3–10. https://doi.org/10.1016/j.firesaf.2012.05.001

Portaria nº 135-2020. (2020). Portaria referente ao regulamento técnico de segurança contra incêndio em edifícios (SCIE). Diário da República Eletrónico (DRE).