V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

SMOKE CONTROL IN AN UNDERGROUND CAR PARK WITH IMPULSE VENTILATION

João L. Aveiro[1], *João C. Viegas[1]

Laboratório Nacional de Engenharia Civil,
Av. do Brasil, 101, 1700-066 Lisboa, Portugal

e-mail: joaveiro@netcabo.pt, jviegas@lnec.pt

Key words: CFD simulation, Underground car park, Impulse ventilation, Experiments

Abstract. *The covered car parks need to have means of ventilation, natural or mechanical, that provide the exhaust of the combustion products released by the engines of the vehicles passing through and to control the smoke released by fire. In last years, a new mechanical ventilation system appeared, based on the use of axial ventilators (jet fans) suspended under the car park ceiling. Jet fans generate the momentum necessary to promote the internal ventilation flow. In this way, the inlets and outlets may be concentrated in some points of the underground car park. Due to the general geometry of car parking, the flow can be rather complex, so parameters like the jet fans position, orientation and velocities have to be carefully chosen [1; 2]. While widely accepted design recommendations for this kind of projects are missing, the use of CFD software, as complement and auxiliary tool in the evaluation of ventilation design, can be of great interest. In this context, CFD simulations were carried out, using a freeware code (the FDS - Fire Dynamics Simulator), to evaluate and confirm the performance of an existing ventilation system, installed in an underground car park, in a fire scenario. The CFD results were compared with experimental values.*

1 INTRODUCTION

The underground parking parks are places where, due to the kind of occupation, the hazard of having a fire is great and must be considered. These spaces have especial characteristics because they usually have reduced height, what makes smoke control difficult in a fire situation. In order to prevent this risk, the Portuguese regulation allows solutions like ventilation, natural or forced, to exhaust the smoke to the exterior, and the internal subdivision of the car park, using fire resistant walls, to prevent the fire propagation.

The use of this kind of solutions has showed to have some drawbacks:

- The partition of the total area implies closing the communications access;
- Implies the installation of a duct system to promote the exhaust of smoke and the entry of new air;
- The space required to accommodate this kind of solutions could be used for parking places.

So, to overcome these drawbacks new solutions have been proposed and tested. The one which has showed best results and is commonly used is impulse ventilation. In last years, this new mechanical ventilation system appeared, based on the use of axial ventilators (jet fans) suspended under the car park ceiling. Jet fans generate the momentum necessary to promote the internal ventilation flow. In this way, the inlets and outlets may be concentrated in some points of the underground car park. This solution is supposed to be less expensive than the traditional smoke control systems based on ductwork.

Due to the general geometry of car parking, the flow can be rather complex, so parameters like the jet fans position, orientation and velocities have to be carefully chosen [1; 2]. While widely accepted design recommendations for this kind of projects are missing, the use of CFD software, as complement and auxiliary tool in the evaluation of ventilation design, can be of great interest.

In this context, CFD simulations were carried out, using a freeware code (the FDS - Fire Dynamics Simulator), to evaluate and confirm the performance of an existing ventilation system, installed in an underground car parking, in a fire scenario.

The flow inside the car parking, equipped with an impulse ventilation system for smoke control, is strongly influenced by the action of the forced convection, even in a starting fire scenario.

The first simulations were carried out without a heat source, in isothermal conditions. At this time the validation was carried out; the CFD results for the velocity field were compared with the experimental values measured during isothermal test in this underground car parking.

The fire simulations were carried out using a heat source with a heat release rate of 4 MW (convective part only), this heat release rate is close to the worst scenario for a car fire. There are no available experimental results for tests using a real heat release source, to confirm the final results obtained by the simulations, because the heat and smoke released in this kind of tests would damage the construction of the underground car park.

2 NUMERICAL MODEL

FDS is a Computational Fluid Dynamics (CFD) model of fire-driven fluid flow [3]. The model solves numerically a form of the Navier-Stokes equations appropriate for low-speed, thermally-driven flow with an emphasis on smoke and heat transport from fires. The partial derivatives of the conservation equations of mass, momentum and energy are approximated as finite differences, and the solution is updated in time on a three-dimensional, rectilinear grid.

The most distinguishing feature of any CFD model is treatment of turbulence. In this study FDS uses one of the main techniques: the Large Eddy Simulation (LES). LES is a technique used to model the dissipative processes (viscosity, thermal conductivity, material diffusivity) that occur at length scales smaller than those that are explicitly resolved on the numerical grid. This means that the parameters μ, k, D in the conservative equations cannot be used directly in most practical simulations. They must be replaced by surrogate expressions that "model" their impact on the approximate form of the governing equations. Following the analysis of Smagorinsky [4], the viscosity μ is modeled as:

$$\mu_{LES} = \rho\left(C_s\Delta\right)^2\left(2\bar{S}_{ij}.\bar{S}_{ij} - \frac{2}{3}\left(\nabla.\bar{u}\right)^2\right)^{\frac{1}{2}}$$

where C_s is an empirical constant. In previous studies [2; 5; 6], where the influence of this parameter assessed in the simulation of impulse ventilation in underground car parks, this value was set to 0.4. In the previous equation Δ is a length on the order of the size of a grid cell. The bar above the various quantities denotes that these are the resolved values, meaning that they are computed from the numerical solution sampled on a coarse grid. The other diffusive parameter, the thermal conductivity, is related to the turbulent viscosity by the equations:

$$k_{LES} = \frac{\mu_{LES} c_p}{\Pr_t}$$

The turbulent Prandtl number $\Pr_l$ is assumed to be constant for a given scenario. In this study was used the default value.

3 EXPERIMENTAL METHOD

3.1 Ground parking park description

The car parking level used in this study is the -6 floor, in a total of 6 underground floors of a parking park. This floor was chosen because experimental values were available and because the conclusions taken for this floor could be generalized to the other floors. The underground park largest dimensions are 220 m for the length and 71 m for the width, the park has a medium height of 3 m. In the next figure is presented the geometry of the park, with the respective existent divisions in fire zones.

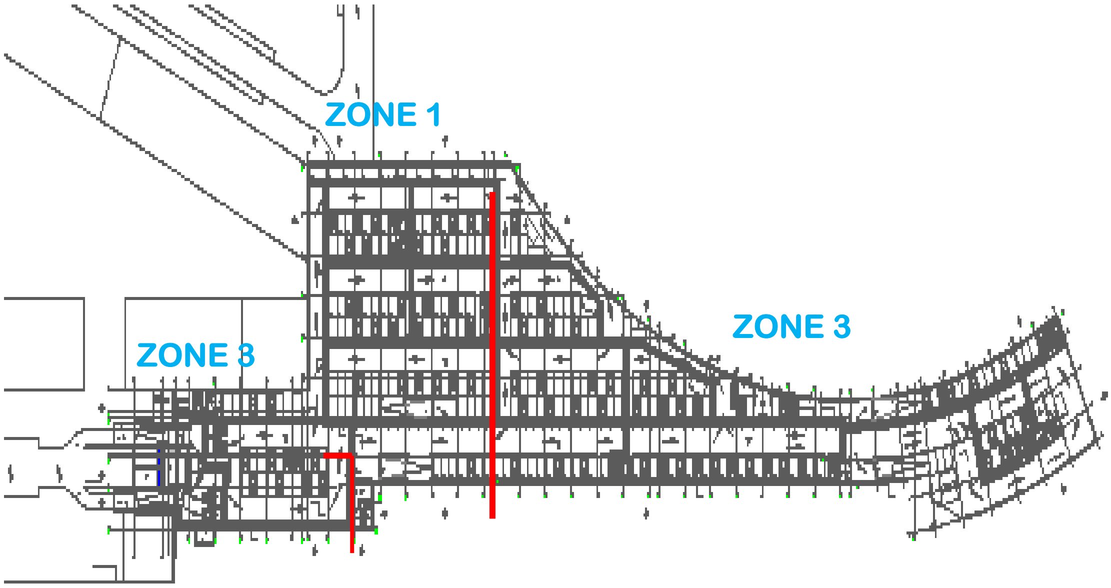


Figure 1: Geometry of the car park.

The car parking has a total area of 9603 m² and it is divided in three distinct fire detection zones. The detection of a fire, in a fire zone, activates a specific set of jet fans that impose a flow pattern and velocity field thought to be adequate for the smoke control. The current ventilation and the smoke control, in a fire situation, are carried out by 22 jet fans (figure 2), suspended below the ceiling of the park. Their function is to generate the necessary thrust to promote the combustion products flow towards the ventilation external openings or to the exhaust fans. 5 axial exhaust fans installed in shafts, being reversible, can supply air from the exterior or can exhaust the combustion products to the outside of the car park.

The reversible jet fans have a maximum power of 1,3 kW, and the unidirectional ones a maximum power of 1,1 kW. They are able to generate a maximum thrust of 50 N. The axial exhaust fans have a maximum power of 40 kW, allowing the exhaust of a maximum volume flow rate of 170.000 m^3/h.

The ventilation openings are completed with the access ramps for the cars. As they are in communication with the exterior they can be used, depending of the existing fire scenario, for incoming of new air or for extraction of the combustion products. In order to increase the efficiency of this configuration, there are jet fans installed in the ramps to promote the air flow.

Figure 2: Car park overview with a jet fan suspended under the ceiling.

3.2 Test method

The tests had a double objective, to proceed with a visual verification of the flow obtained through the use of the smoke ventilation system and to allow registering experimental values to support the validation of CFD simulations.

The use of a fire source similar to a real underground car park fire situation wasn't easy because the heat and smoke released in this kind of tests would, for sure, damage the building. Instead the tests were carried out in isothermal conditions, cold smoke was generated by a smoke source used by firemen training.

There is a disadvantage of this solution: it does not allow obtaining experimental data about the disturbance caused by the buoyancy driven flow due to fire in the flow due to jet fans. However, previous studies shown the satisfactory performance of the computer code used in this study in solving flow problems originated only by jet fans [5] or only by a buoyancy driven flow [6].. It is supposed that the computer code is adequate to carry out simulations involving the two types of phenomena together. Moreover, this computer code is largely used by fire science community to carry out fire simulations.

The experimental work carried out on the underground car park included two types of tasks: the use of a cold smoke source to visualize of the isothermal flow and the measurement of velocity and flow direction in the most important places of the fire detection zone.

For the measurements hot wire anemometers AIRFLOW TA-5 model were used. The accuracy of the measurements was ± 0,31 m/s, according to the manufacturer. The hot wire anemometer was always lined up with the directions of the velocity vector, using for this purpose a tiny line placed in the device extremity. The information given by this line, direction of the flow, was also register. For this type of measurement was estimated an accuracy of ± 20º. The turbulent character of the flow made difficult to evaluate its direction correctly. The difficulties increased to low velocities. In each position 11 readings of the absolute valor of velocity were made (one reading every 2 seconds). The average value was considered. The measurements were made at a height of proximally 2.00 m.

In figures 3 to 8 are showed the flow patterns as well as the positions and values measured for velocities. The position of the smoke source is indicated by a red circle with a point inside. Every figure represents a different fire detection scenario.

Figure 3: Flow pattern in the tests of fire detection zone 1.

Figure 4: Measured velocities and directions for fire detection zone 1.

Figure 5: Flow pattern in the tests of fire detection zone 2.

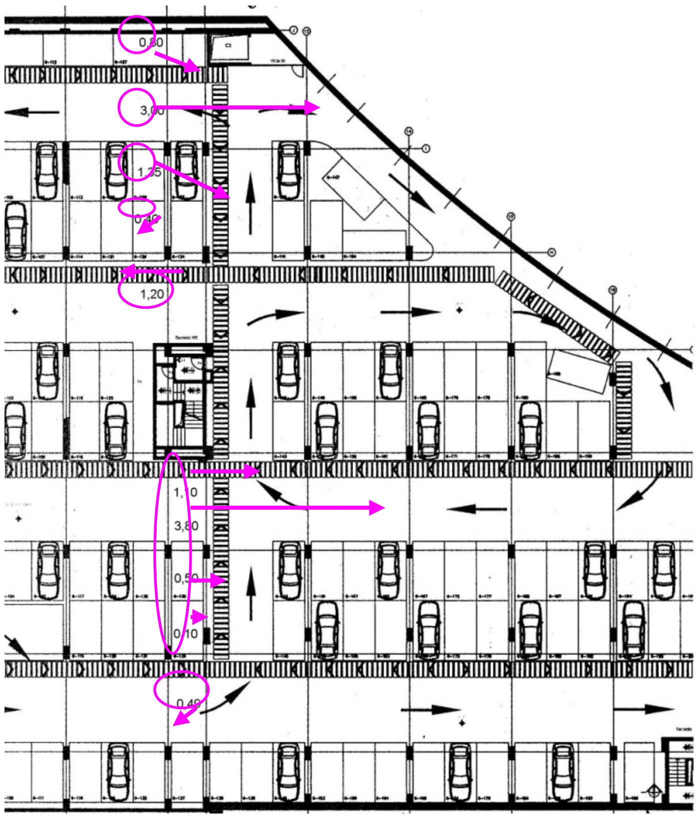


Figure 6: Measured velocities and directions for fire detection zone 2.

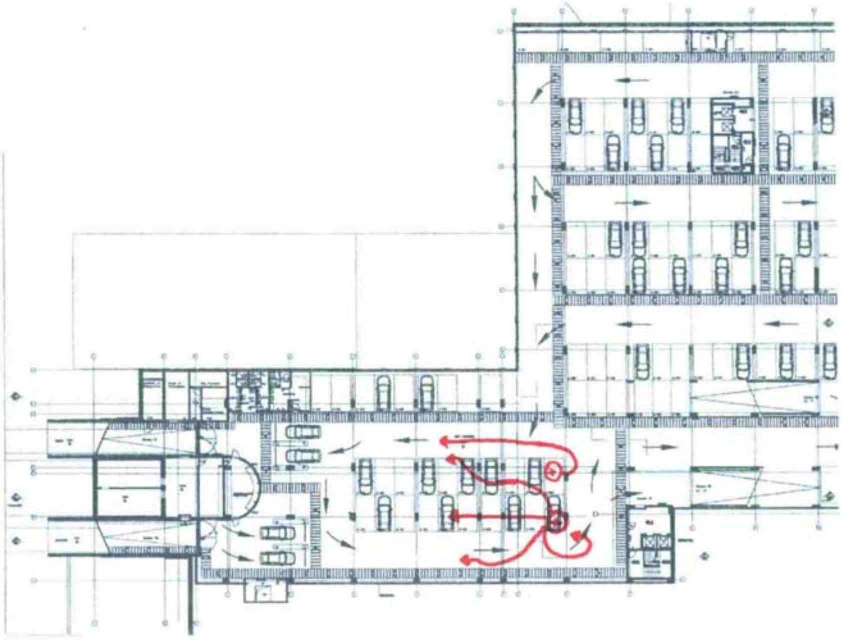


Figure 7: Flow pattern in the tests of fire detection zone 3.

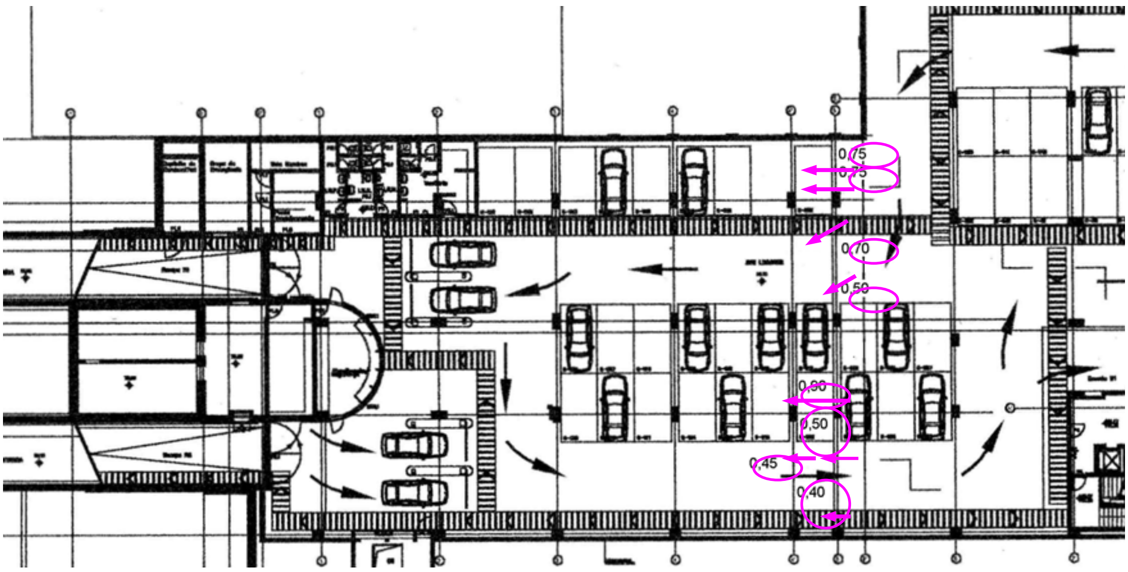


Figure 8: Measured velocities and directions for fire detection zone 3.

4 COMPARATION WITH THE COMPUTACIONAL RESULTS

4.1 Numerical model

Due to the great dimension of this car parking level, it was not necessary to include all his extension, with all the three zones, in the calculation domain of the CFD analysis. For that reason, there is a section of zone 2 that was not taken into account in all the simulations. This calculation domain simplification allows using a more fine mesh instead of a coarse mesh that would have negative reflexes in the calculation's performance. Therefore, the results quality is better and it reduces substantially the time needed to carry out the simulation. The calculation domain external boundaries were located as far as possible from the fire zone in order to reduce their influence on the simulation of the most important physical processes.

The calculation domain considered is equal in all the simulations (see figure 9), only the fans positions and function mode (on/off) are changed according to the fire detection table. In the left boundary of the domain was imposed a free opening condition, it simulates an access zone to outside of the car park. A similar condition was imposed in the right boundary of the calculation. The remaining part of fire detection zone 2 was not considered in the simulations.

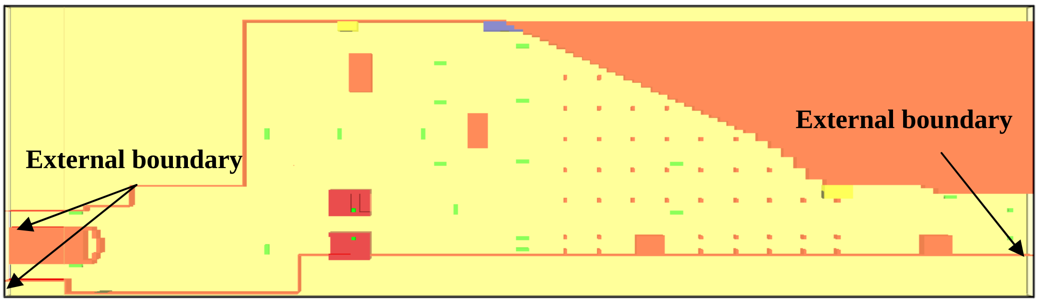


Figure 9: Calculation domain used in the simulations.

The calculation domain is 240.6 m long (coordinate x), 75.0 m wide (coordinate y) and 3,16 m high (coordinate z). It was discretized by an irregular Cartesian grid. In the zones nearest the fans a more fine mesh was used in order to improve the simulation of the jet. The mesh was defined according to previous studies [2; 5; 6]. The domain is divided in to 729x225x11 cells.

The simulations were running in a computer equipped with a AMD Athlon™ 3500+ processor with 2,21 GHz and 1 GB of memory. The required time to perform each simulation was about 3 days.

4.2 Fire detection zone 1

In figure 10 is presented the velocity field predicted for steady and isothermal conditions.

It is possible to identify the direction of the flow towards the duct C1 (extraction ventilator), as well as the zones where the flow reaches higher velocities.

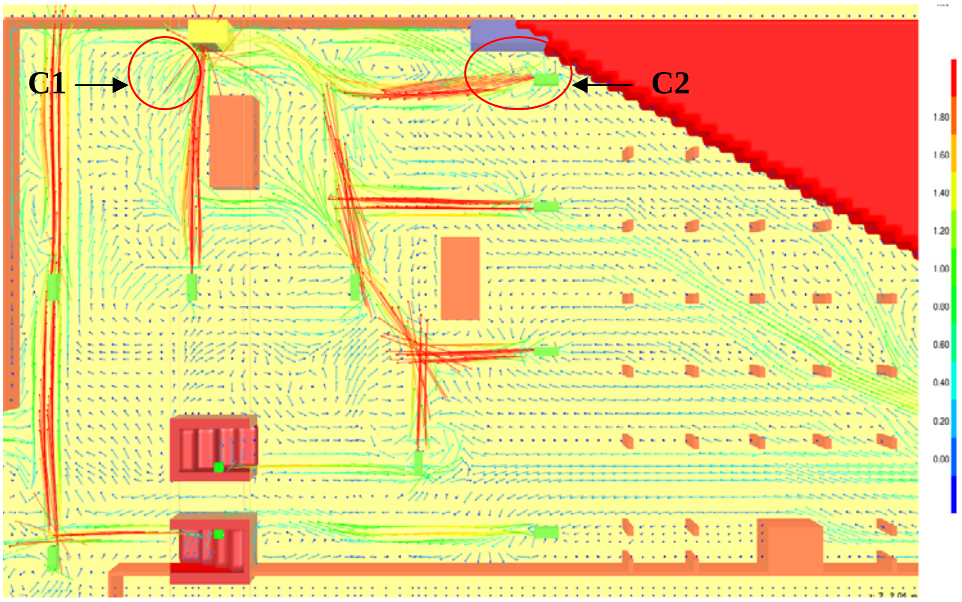
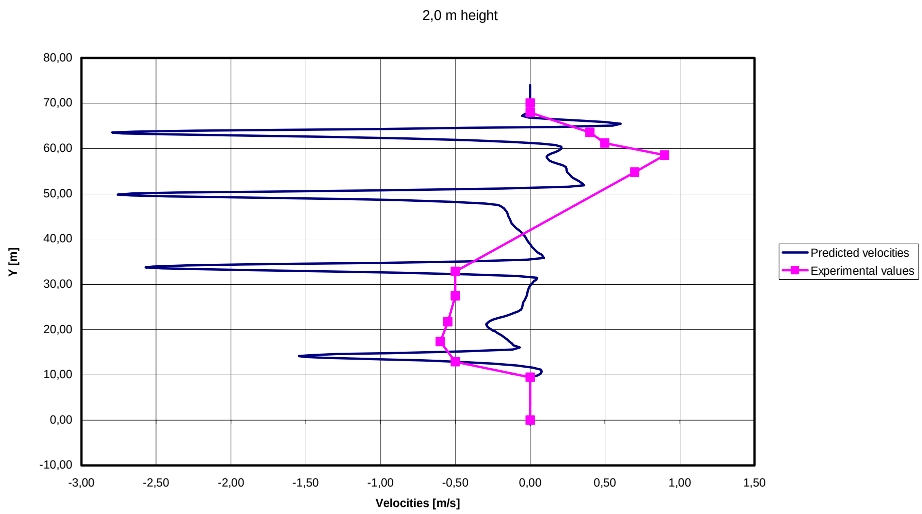


Figure 10: Predicted velocity field for fire detection zone 1at 2,01m height.

In figure 11 the predicted velocities for fire detection zone 1 are compared with the results of the measurements. As showed previously in figure 4, there are two different places where experimental values are available; the graphics in the figure correspond to these two places.

It is possible to say that in general the foreseen velocities are higher than the experimental values. This difference can be explained by the type of conditions imposed in the jet fans nozzles: small inaccuracies in the placement of the anemometer correspond to big variations of velocity. However, having in mind the type of analysis that is wanted (macroscopic analysis in the entire fire detection zone), it can be admitted that this difference does not correspond to a very significant errors. In fact, it is possible to see that the computational results show well the recirculatig flows near de exhaust fan C2.

In the second graphic the computational results are similar to the measurements. To be noted that this measurement zone is not in the vicinity of any jet fan.



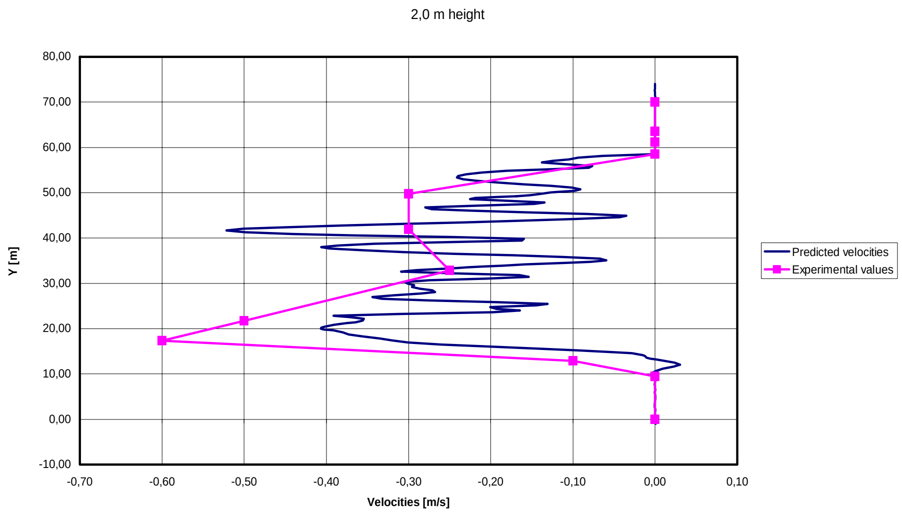


Figure 11: Comparison of the predicted velocities with the experimental values for fire detection zone 1.

4.3 Fire detection zone 2

In figure 12 is presented the velocity field predicted for steady and isothermal conditions.

It is possible to identify the direction of the flow towards the ducts C4 e C5 (exhaust fans), as well as the zones where the flow have higher velocities. The inclusion of the park columns in the simulation was necessary to dissipate the flow towards the exhaust. It is possible to see that the domain simplification (cut of the calculation domain at the right side of C4 and C5) does not have great importance in the relevant area (left side of the exhaust fans C4 and C5). The boundary conditions were built in order to simulate the flow coming from the missing part of the car park.

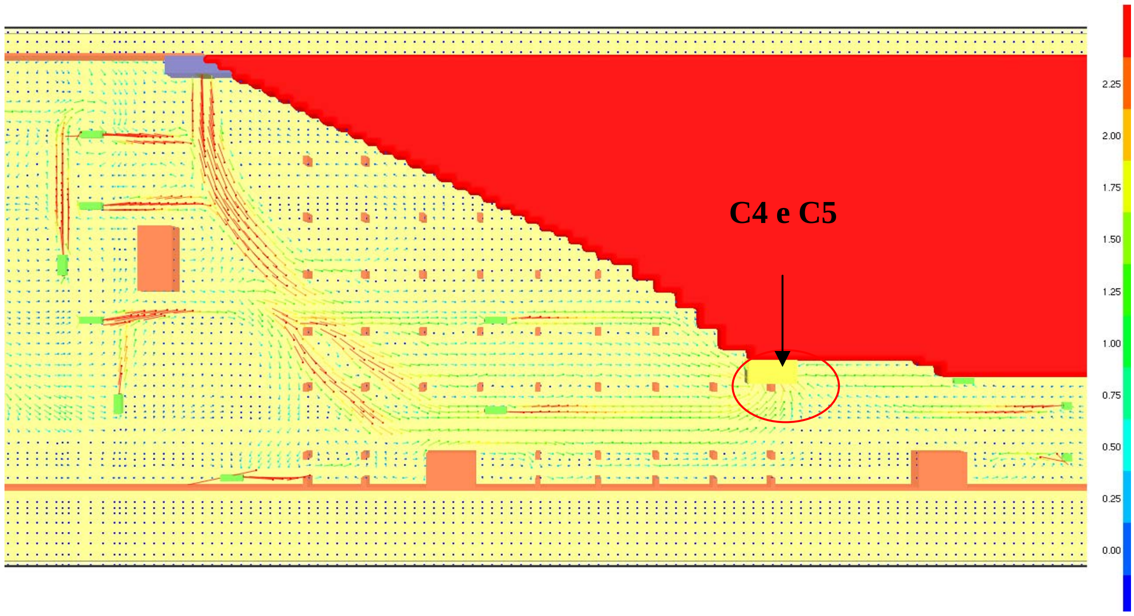


Figure 12: Predicted velocity field for fire detection zone 2 at 2,01m height.

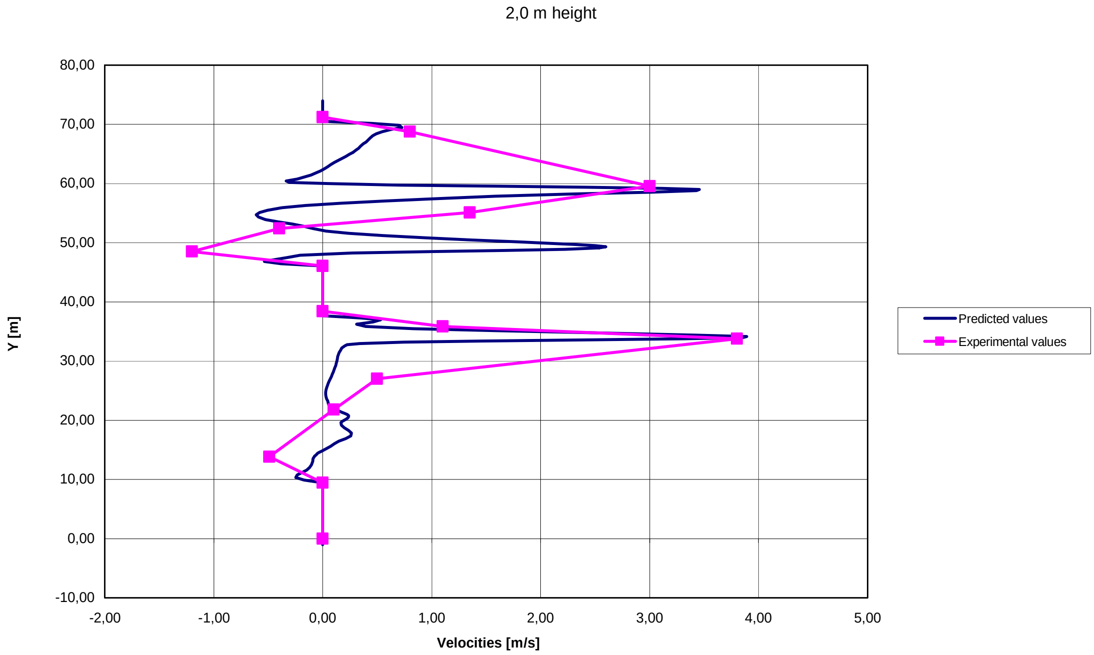


Figure 13: Comparison of the predicted velocities with the experimental values for fire detection zone 2.

In figure 13 the predicted velocities for fire detection zone 2 are compared with the results of the measurements.

It is possible to say that the results show an acceptable between the experimental values and simulations values. Both directions and absolute value of the velocity are in agreement. The minor differences can be explained because measurements have been made in points with some distance in between, giving it a discrete form, while the predictions have a more continuous shape.

4.4 Fire detection zone 3

In figure 14 is presented the velocity field predicted for steady and isothermal conditions.

It is possible to identify the direction of the flow towards the access ramps to outside R1 and R2, as well as the zones where the flow have higher velocities. The jet fans in the ramps are reversible; in this case they are working forcing the flow to outside.

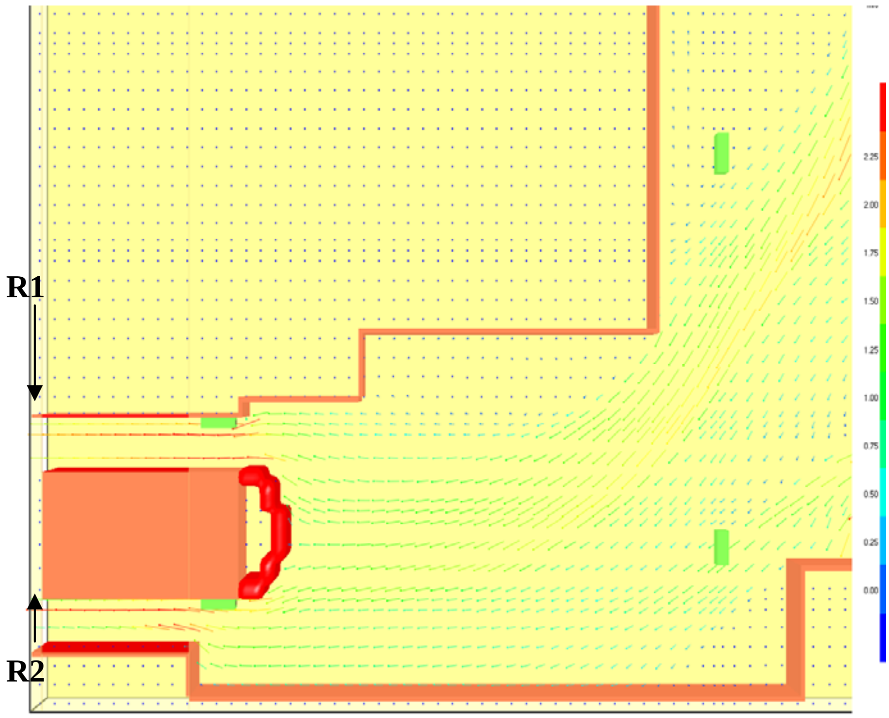


Figure 14: Predicted velocity field for fire detection zone 3.

In figure 15 the predicted velocities for fire detection 3 are compared with the results of the measurements.

It is possible to see that the differences between simulation results and experimental values can be divided in two areas. One of that is a zone, close to the inferior wall of the calculation domain, where the measured velocities are slightly higher. However, it is possible to see that both curves have the same tendency. In the zone near the superior wall of the calculation domain it is the reverse: the measured velocities are lower than the computational results. This inaccuracy was introduced by the jet fan placed in the ramp to access the level -5 above. This jet fan is used to avoid the smoke released during a fire to contaminate -5 level. The flow entrained by this jet fan is very difficult to simulate accurately because the ventilator is tilted as the ramp.

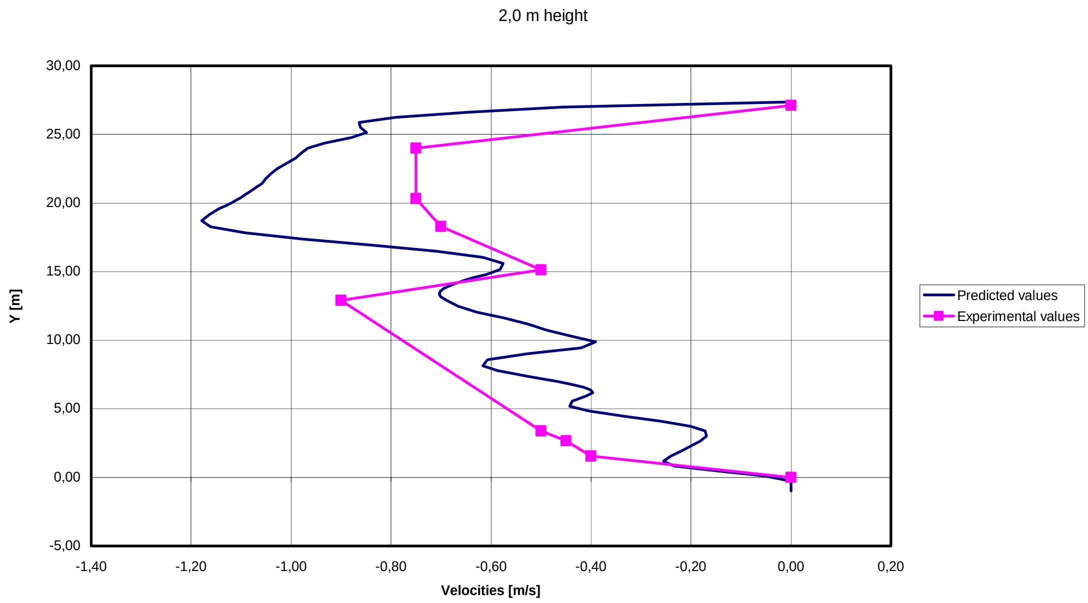


Figure 15: Comparison of the predicted velocities with the experimental values for fire detection zone 3.

Despite some differences observed it is possible to say that the predicted results do not show significant errors. While the simulations do not introduce false critical situations, like new stagnation points or recirculating flows, it is expected that the efficiency of the smoke ventilation system is not adversely affected in simulations.

5 FIRE SIMULATION

To simulate the performance of the smoke control system in the event of fire, the same calculation domain used for isothermal simulations was adopted. It was considered a heat source with a convective heat release rate of 4 kW covering an area of 4 m x 2 m, corresponding approximately to a burning car. This source was positioned on the places thought to be more problematic for each fire detection zone.

The simulations were carried out in transient conditions until the steady state was reached. Only steady results are presented. To be able to identify the zones contaminated by smoke released during the combustion, the temperature field is presented at a height of proximally 2,01m.

5.1 Fire detection zone 1

To evaluate the efficiency of the ventilation system the source of heat was located in two different places, being both places close to the boundary with the fire detection zone 2. In figure 16 is possible to see the placement of the heat source and the predicted results.

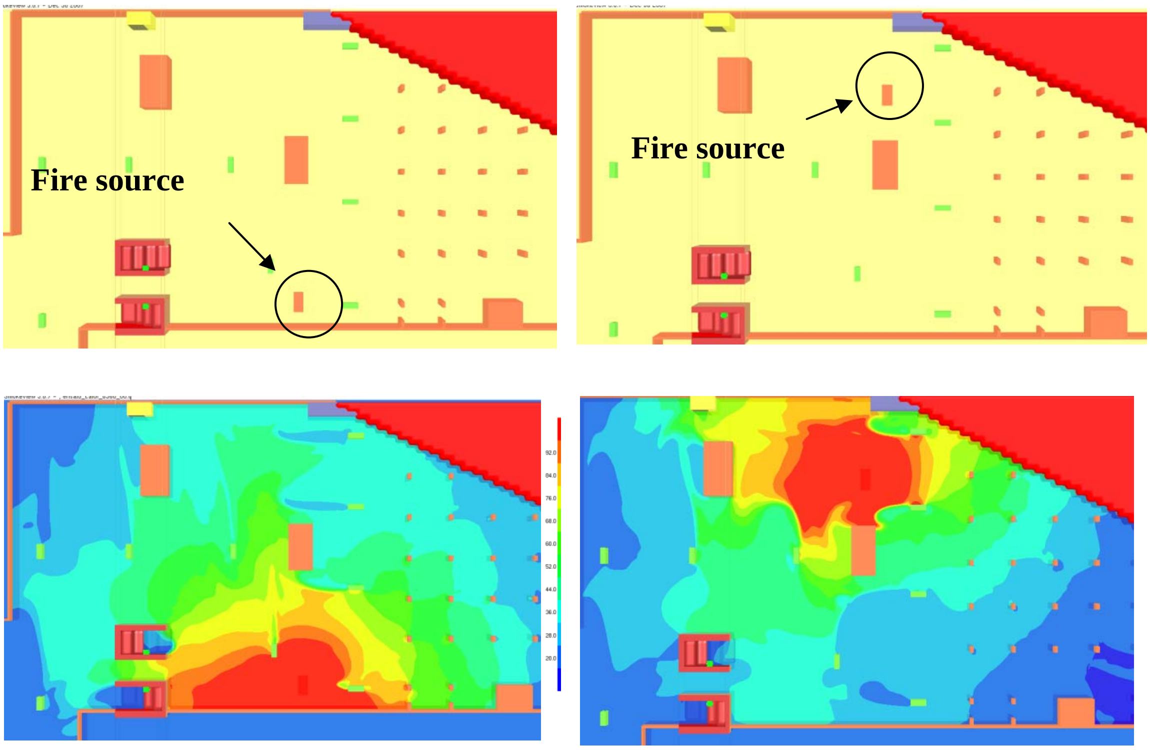


Figure 16: Simulations with heat source, position of the source and respective results of the temperature fields for fire detection 1.

As predicted through the previous simulations, in isothermal conditions, there is a significant flow towards the fire detection zone 2. However, it is possible to see that there is a tendency for the smoke to be kept in the vicinity of the boundary between zones 1 and 2. In the other boundary (between detection zones 1 and 3 – in the left side of the calculation domain), the flow driven by the buoyancy is not strong enough to overcome the jet fans action. The jet fans constrain significantly the smoke flow and promote his dilution, contributing for a temperature reduction. In the more significant part of the domain, the temperatures do not exceed 45ºC an only near the heat source reach over 90ºC.

5.2 Fire detection zone 2

In figure 17 is possible to see the placement of the heat source and the predicted results.

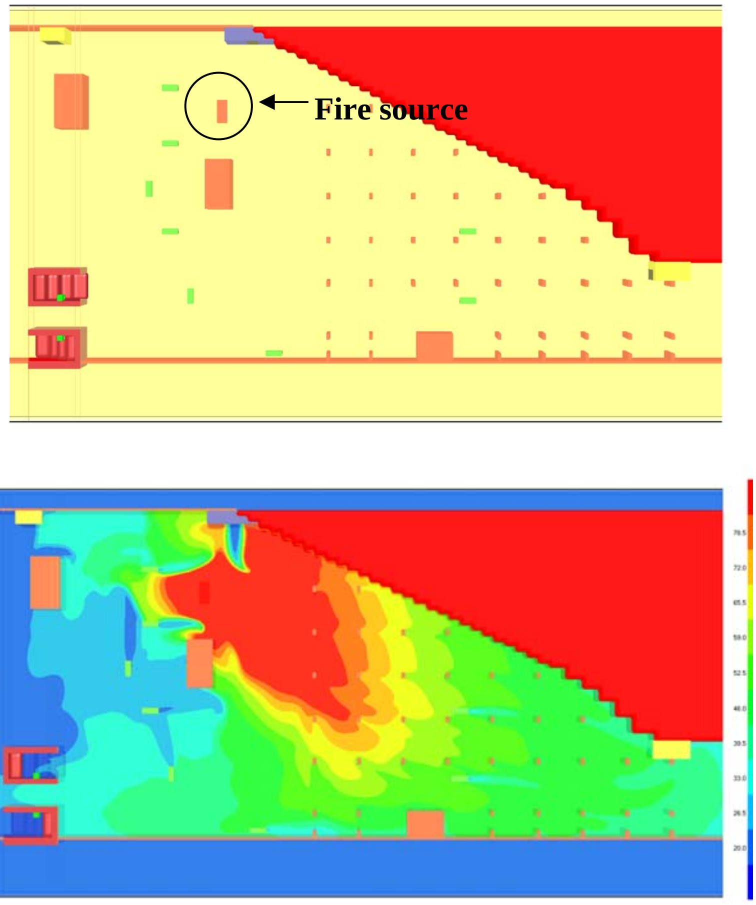


Figure 17: Simulations with heat source, position of the source and respective results of the temperature fields for fire detection 2.

By positioning the heat source over the boundary between the fire detections zones 1 and 2, the simulation represents the worst case; therefore, the conclusions will be conservative (in the safe side). The results have showed to be very consistent to the real phenomena. As it can be observed there is a tendency for the smoke to overcome the jet fans effect near the upper wall and near the bottom wall. Despite this tendency, it is also possible to see that most of the smoke flow (and the zone with the highest temperatures) is kept inside the fire detection zone 2.

In the more significant part of the domain, the temperatures do not exceed 45ºC an only near the heat source reach over 90ºC.

5.3 Fire detection zone 3

In figure 18 is possible to see the placement of the heat source and the predicted results.

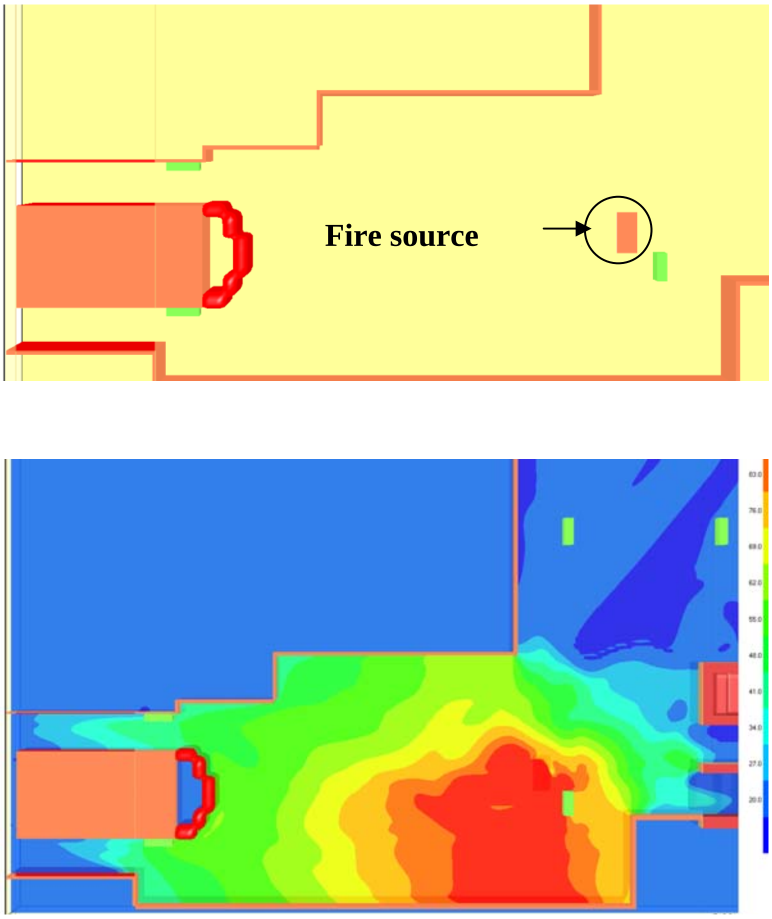


Figure 18: Simulations with heat source, position of the source and respective results of the temperature fields for fire detection 3.

It is possible to see that smoke was confined by the vicinity of the fire detection zone 3 boundary, even in the zone between the two ramps very closed the source position. The exhaust through the ramps to outside seems to be very efficient.

In the more significant part of the domain, the temperatures do not exceed 45ºC an only near the heat source reach over 90ºC.

6 CONCLUSIONS

- The computational results, despite of some inaccuracies with the experimental values in some of the cases studied, show an acceptable agreement with the velocity fields observed during the experimental tests.
- Considering this adequate performance of the computer code applied to this calculation domain, it is expected that the predictions of velocity and temperature fields in case of fire simulate adequately the real case.
- The fire simulations show that the design of the impulse ventilation system of this underground car park is adequate.
- The jet fans have an important role constraining the smoke flow released by the fire, allowing the appearance of almost smoke free zones were people can egress as well as the firemen can fight the fire.

- Other advantage of using this kind of solutions is related with the strong dilution of the smoke that reduces drastically the smoke temperature and toxic and irritant combustion products concentration.
- The used methodology and the use of CFD is a good aid for the correct evaluation of the efficiency of ventilations and smoke control systems based on jet fans,, avoiding the need of realistic tests (more expensive and difficult to carry out).

ACKNOWLEDGEMENTS

This research was supported by Fundação para a Ciência e Tecnologia (Research Project PTDC/ECM/68064/2006).

REFERENCES

[1] J. C. Viegas and J. G. Saraiva, CFD Study of smoke control inside enclosed car parking, Proceedings of Interflam 2001, 9th International Fire Science & Engineering Conference. Edinburgh: Interscience communications (2001).

[2] J.C. Viegas, "Utilização de ventilação de impulso em parques de estacionamento cobertos", LNEC (2008).

[3] K. McGrattan, Fire Dynamics Simulator (Version 4). Technical Reference Guide, NIST, NISTSP 1018 (2005).

[4] J. Smagorinsky. General Circulation Experiments with the Primitive Equations. I. The Basic Experiment. Monthly Weather Review, 91(3): 99-164, March 1963

[5] J. C. Viegas, The use of impulse ventilation to control pollution in underground car parks. *The International Journal of Ventilation*. Vol. 8, No 1. Coventry: VEETECH, Ltd. (2009). (http://www.atypon-link.com/VEET/doi/abs/10.5555/ijov.2009.8.1.57)

[6] J. C. Viegas, The use of impulse ventilation for smoke control in underground car parks. *Tunnel. Underg. Space Technol*. Vol. 25 (2010), pp 42-53. doi:10.1016/j.tust.2009.08.003.