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

Indoor Car Parks – CFD Application

R. Fernandes; D. Henriques

France Air Portugal Lda Av. Casal da Serra I-4, Escr. 3, 2625-085
Póvoa de Santa Iria, Portugal

email:
ricardo.fernandes@france-air.com
daniel.henriques@france-air.com

Key words: CFD, Jet fan, smoke control, underground car park.

Abstract. *In the last few years, ventilation projects have been object of several changes and numerical simulation (CFD) has been a remarkable tool in this evolution.*

Jet fans, usually used in tunnel's ventilation, started being applied in car parks and contributed for the numerical simulation's increase to support all the solutions. CFD became an important resource when there are doubts regarding to the efficiency of smoke extraction in underground car parks or when the situation gets out of scope of Portuguese legislation. It's use allows the optimization of all projects, both the level of equipment as the array command, ensuring an effective smoke extraction.

There are two different cases in the several studies we follow with two particular characteristics.

In the first project, numerical simulation CFD has shown the importance of using jet fans to virtual partitioning different zones in a car park and to evidence the advantages of using 100% reversible fans. With those factors, it is guaranteed the escape of who is in the park and the access of the fire department to the disaster area.

In all car parks projects it's also essential to ensure ventilation when the concentration of CO (carbon monoxide) is high. In this study there weren't zones with stagnant air, so all park is well ventilated.

In the second project, numerical simulation CFD helped identify the interaction of the ventilation system with the wind effect in the openings of the inlet air. When the project has mechanical extraction and natural air compensation, wind effect can change completely the smoke extraction efficiency. In this specific project, in normal conditions, the existing mesh of unidirectional jet fans should be enough to forward all

the smoke to the extraction zone, but the wind effect was significant, not allowing the perfect ventilation.

With the use of numerical simulation it was possible to obtain one array command able to react effectively whatever the effect of wind. It was concluded that it should be installed anemometers in the natural openings measuring speed and direction of air. This way, it was possible revert the direction of extraction. It could be made through the "original" extraction fans or through the natural openings.

Validation tests made in the job proved that the ventilation system wouldn't work safely for people and for the fire department if the effect of wind had not been considered.

CFD is getting an important relevance in the optimization of car parks ventilation solutions. It's a resource that allows testing several situations without additional costs or without using expensive and long practical tests.

1 INTRODUCTION

It is well known that the major cause of deaths during a fire is the hot toxic smoke, rather than the fire itself. The control and essential removal of this smoke from the building is, therefore, a vital component in any fire protection scheme. As our knowledge of the behavior of fires increases, the traditional methods of exhausting the fire smoke are sometimes shown to be inadequate; and systems using positive and readily controllable jet fan units are often used instead. This paper will bring same examples of this application, showing the need of Computational Fluid Dynamics - CFD for a correct project.

The outbreak of a fire results in the immediate production of hot toxic smoke. If left to its own devices, the suction action of the fire will fill an underground car park with smoke in minutes and anyone caught in the parking would not be able to see or breathe. The first objective of any smoke venting system is to keep the escape passages free from smoke to assist in the evacuation of people.

In addition a well designed ventilation system can assist the Firemen by making it easier to find the source of the fire, and reach the fire trough a clean zone.

Enclosed and underground car parks present a very special problem for the smoke control engineer.

Car park ventilations systems have two functions. First to remove the fumes, mainly carbon-monoxide, produced by vehicles moving within the car park. Second, have the capacity to at least remove the smoke produced by a car on fire in the car park.

Portuguese Fire Regulations [1] prescribed for this two requirements flow rates of $300m^3/h$ per car for concentrations of 50 ppm CO or higher (until 100 ppm CO), and 600m3/h per car for concentrations of 100 ppm CO or higher as well as in case of fire [1]. As we know, when we use jet fans instead duct network, jet fans increases the total moving air volume, and the extraction does not have the capacity to absorb this total air flow. This occurs, because a jet fan induces approximately 10 times his air flow. All combine, total extraction flow is very superior of values on Fire Regulations. Computational Fluid Dynamics was an important tool to demonstrate this conclusion.

This technique was originally used in road tunnels to control smoke and pollution and recently applied to car park.

This technique allows avoiding subdivisions of the space and ventilation duct network. However, airflow behavior is very difficult and complicated to predict.

Complex flow patterns may be generated by the jet fans, the geometry of the car park, and some obstacles to the flow. This may produce the spread of the smoke for all zones, making very difficult person's evacuation.

Computational Fluid Dynamics – CFD – is a helpful tool to predict the expected flow patterns and to verify if spread of smoke occurs and if exist some stagnant zones very critical to control CO concentration.

CFD ensures system optimization and, more importantly, that occupants safety is not compromised. This avoids making a project extremely expensive by using too many fans, or an under performing system by specifying too few.

In the present work, we will see two projects where were evolved CFD.

In the first project, CFD has shown the importance of using jet fans to virtual partitioning different zones in a car park and to evidence the advantages of using 100% reversible fans. With those factors, it is guaranteed the escape of who is in the park and the access of the fire department to the disaster area. In this study there were not zones with stagnant air, so all park is well ventilated.

In the second project, CFD helped identify the interaction of the ventilation system with the wind effect in the openings of the inlet air. When the project has mechanical extraction and natural air compensation, wind effect can change completely the smoke extraction efficiency.

2 MATHEMATICAL MODELS

Flow simulations are performed using the version 3.0.16 of AIRPAK (FLUENT-ANSYS) commercial program [2]. The numerical code is based on the finite-volume method to solve the RANS/URANS equations [3].

2.1 Features of the source energy

CFD's are carried out to determine the influence of the volume of fire in the spread of hot and toxic gases as well as the temperature field inside the computational domain. We use for all CFD energy release by the fire of a modern car 4MW, with energy source volume of 7.7m3. This value corresponds to the convected part of energy produced by the fire [4].

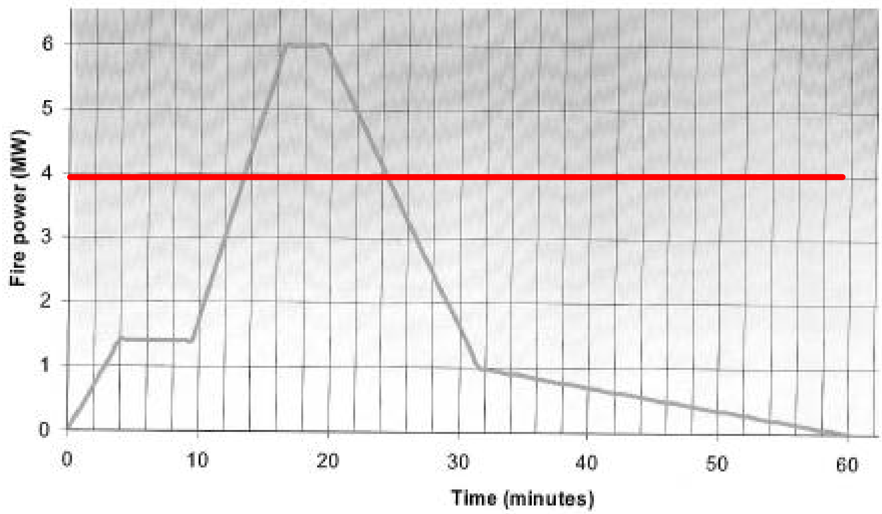

Figure 1 - Typical heat release rate by a fire of a modern car in underground car park: European Commission; Red line represents the power used in steady numerical simulation

Steady numerical simulation allows a good compromise between accuracy and CPU time (steady simulation require smaller CPU time than unsteady simulation).

Table 1 – Steady-state design fires

Fire parameters	Indoor car park without sprinkler system	Indoor car park with sprinkler system
Dimensions	5 m × 5 m	2 m × 5 m
Perimeter	20 m	14 m
Heat release rate	8 MW	4 MW

Table 1 - BS7346-7 [5] defines the power to use on steady simulation.

Fire is modeled through an energy source volume located in the computational domain. Smoke particles are not simulated in the present numerical simulations. However, smoke concentration and temperature field develop similarly results and so that, field temperature is a good parameter to calculate the dispersion of smoke in the car park.

2.2 Turbulence model

The turbulence model used in the numerical simulations is the standard k-e turbulence model.

2.3 Numerical model for simulating isothermal flow – ventilation CO

Steady simulated using RANS equations. Air is considered as an incompressible fluid. Coupling between pressure and velocity is solved using the SIMPLE algorithm. A standard second order pressure scheme is used for the pressure and convective terms are discretized by a second order upwind scheme. Closure of the model is performed using the k-e turbulence model in its standard formulation with current wall functions. Under-relaxation parameters are 0.3 for the pressure and 0.7 for the momentum.

2.4 Numerical model for simulating fire in the car park – smoke extraction

Steady flow is simulated using RANS [6] equations and energy equation. Change in air density is calculated using the ideal gas equation. Coupling between pressure and velocity is solved using the SIMPLE algorithm. Convective terms are discretized by a second order upwind scheme, like turbulence kinetic energy and dissipation rate and energy. Turbulence is simulated using the standard buoyancy-modified version of k-e model, which allows taking in account the buoyancy effects, with the current wall functions. Gravity forces are also calculated. Under-relaxation parameters are 0.3 for the pressure and 0.7 for the momentum.

2.5 Numerical model of jet fan

A simple numerical model of jet fan is developed to take into account the volume flow rate and the swirl flow induced by the fan.

Generally, jet fans are placed on the ceiling. They consist on a box of variable geometry (cylindrical, rectangular), an axial fan positioned within and through the cylindrical body and an electric motor.

The numerical model of jet fan has the configuration presented in Figure 2, similar to the dimensions of jet fan used, the AXALU TR 40 jet fan of France-Air [7] (fan diameter, 0.4m; unit height, 0.504m; unit length, 2.365m; unit width, 0.504m and maximum flow rate, $2.3m^3/s$). It consists of two square faces, where the useful two circular faces of inlet and outlet, and a rectangular box.

The following simplifications are considered for the present numerical jet fan model:

? Representation of the outside fan impulse: the inside part is not analyzed (shovels and interior conduct).

? Outside rectangular box, for simplifying the construction of the mesh around the jet fan.

? Circular section at each end of the jet fan, where velocity conditions are imposed: inlet and discharge with their boundary conditions.

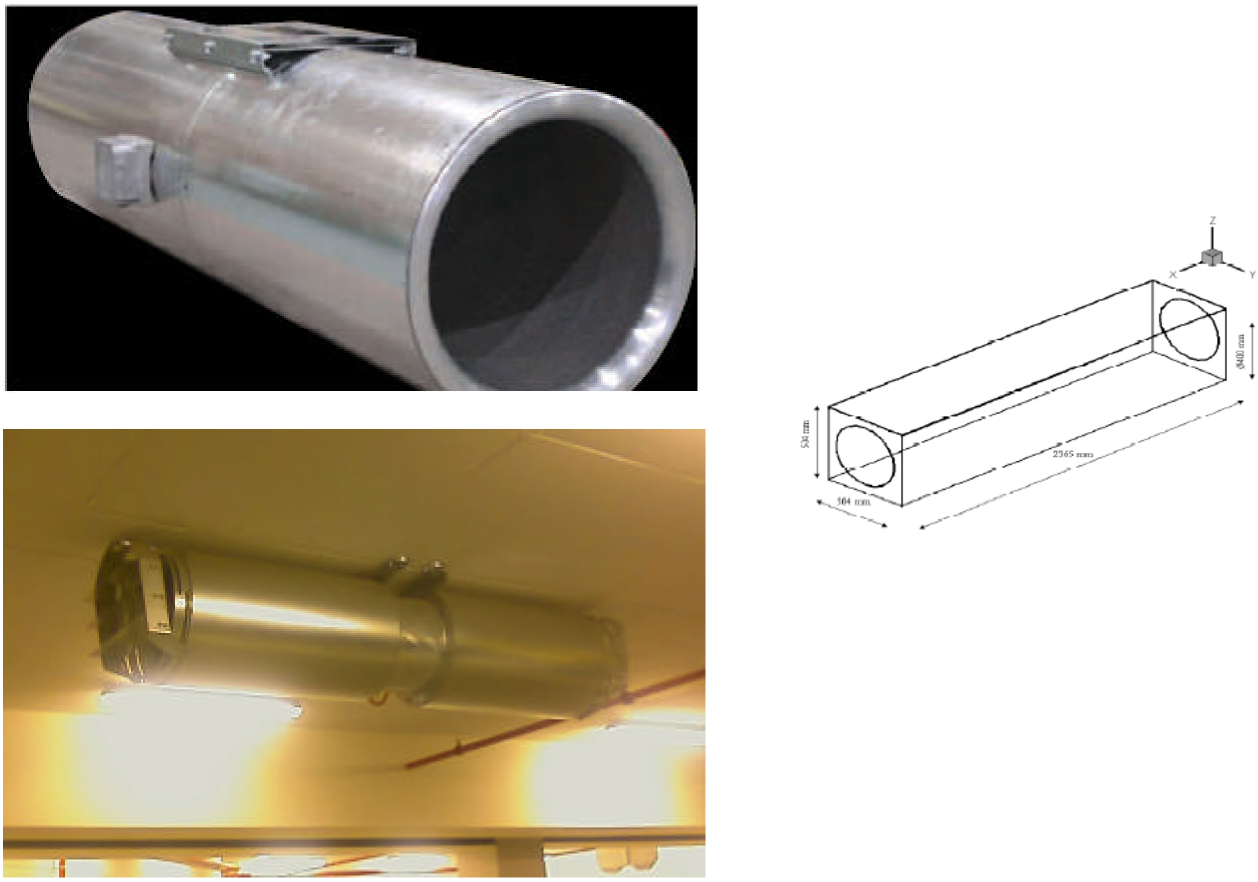

Figure 2 - Jet fan in an underground car park and numerical jet fan model.

The axial fan placed inside the physical body of the jet fan causes a flow with an axial velocity of 18.3m/s (corresponding flow rate equal to $2.3m^3/s$). This velocity is imposed on the aspiration face as well as the supply face of the jet fan, to respect the flow conservation through and flow induced by the jet fan. Swirl is imposed at the discharge of the jet fan model with maximum tangential velocity equal to 11.25m/s.

3 USING FANS 100% REVERSIBLE

In recent years jet fan or impulse technology has established itself as the new standard in car park ventilation. Impulse ventilation systems became a standard instead ducted mechanical extract systems. Series of jet fans, mounted under the ceiling, induce air movement from the air inlet openings towards extract points, moving smoke with it.

In the logic of this technique, we can extract the smoke near the fire avoid the spread smoke to all zones. With fully reversible ventilation systems we have an effective smoke control and limit the smoke dispersion in the car park.

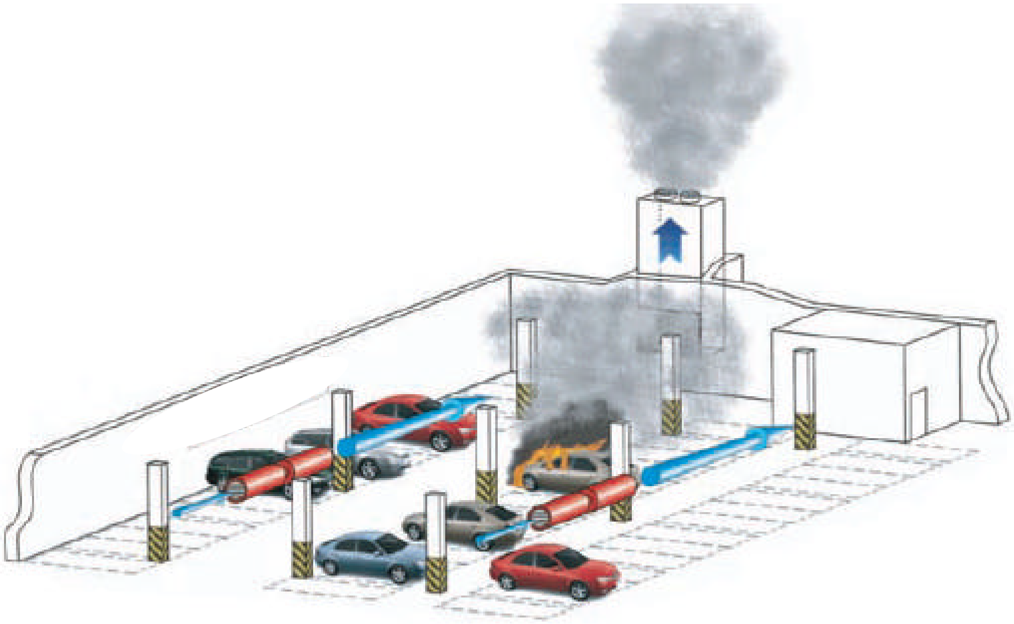

Figure 3 - Jet fan system in an underground car park.

CFD is used to verify the efficiency of the ventilation system in the level -3 of an underground car park. Figure 4 shows the car park, the virtual smoke control zones that allow defining the direction of operation of the reversible ventilation system, the localization of the 17 jet fans (8un. 100% reversible's and 9un. unidirectional) and the position of the 2 100% reversible axial fans (exhaust or inlet fans, according the direction of the ventilation system).

Axial fan ventilation is achieved by one grid, with area 2x2m on the exhaust and on inlet. Low velocity on inlet air allows a homogenous velocity in the car park.

Is also very important to check the start-up times of fans, because in case of fire close the curtains first, then connect the inlet and exhaust fan, and only at the end of this sequence, when it's already created a plume of stratified smoke start-up jet fans. This occurs because when you start-up the jet fans they will generate a lot of induction and damage smoke stratification, thus spoiling the field of vision early in the fire.

Is different when we talk about CO control because here first start the jet fans and then the inlet and exhaust fans. This happens because we take the ability of jet fans to mix and dilute the concentration of carbon monoxide.

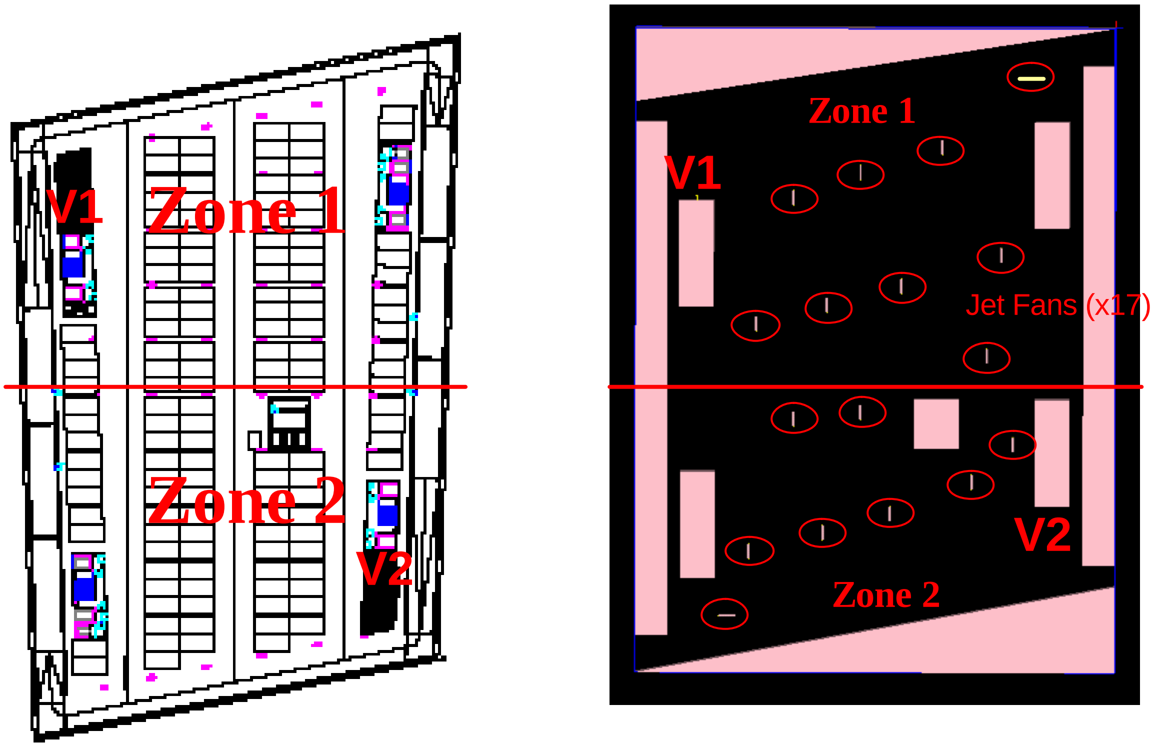

Figure 4 - Underground car park with identification of virtual smoke control zones, axial fans and jet fans.

The car park is 6856m^2 with 3m height. Circular pillars, with diameter 0.8m, are not taken into account. The ventilation system is equipped with jet fan Axalu TR 40, with fan interior diameter 0.4m, and rate flow equal to 2.3m3/s (maximum velocity used in case of fire), i.e. an axial velocity equal to 18.3m/s. Swirl is imposed at the outlet face of the jet fan model with maximum tangential velocity equal to 11.25m/s. Maximum rate flow of axial fan is 117000m^3/h. Regulate [1] flow was 75600 m^3/h, multiplies 126 cars per 600 m^3/h. Insufficiently flow rate to give good results, because don't have capacity to absorb all jet fans momentum. This kinetic energy is necessary to drag smoke to exhaust fan. All the ramps are closed when a fire occurs in the car park. The energy release by the fire of a modern car is 4MW, with energy source volume of 7.7m^3. The fire is located fist in the smoke control zone 1, and next in the smoke control zone 2, as indicated in Figure 6 and 12.

The temperature of supply air and ambient temperature is 20 degrees.

Figure 5 presents a detail of the mesh, with a refinement near the more critical zone, i.e. the jet fan, the grid of exhaust / inlet fans, around the fire source and near the ceiling. The mesh is composed by 1196400 control volumes.

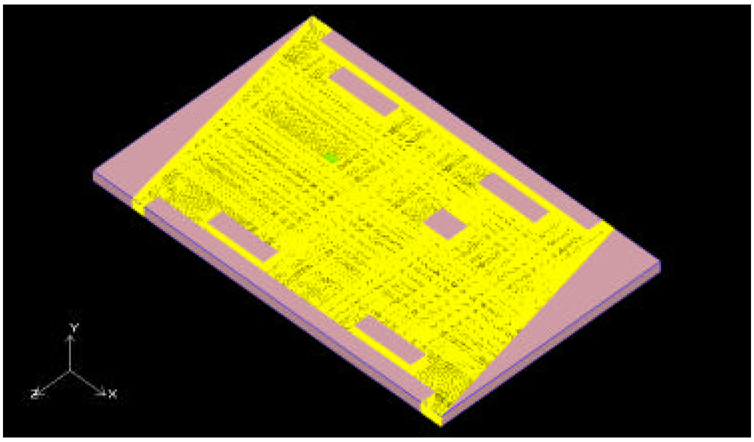

Figure 5 - Detail of the mesh.

3.1 Steady flow simulations of smoke control system on zone 1

CFD is used to verify the efficiency of the ventilation system in the level -3 of the underground car park and to compare two ventilation systems for smoke removal in each zone, only changing the position and number of jet fans. The exhaust fans operate at maximum capacity (100% of the extraction power) and jet fans operate in smoke control zones 1 and 2. Some jet fans are stopped in this zone.

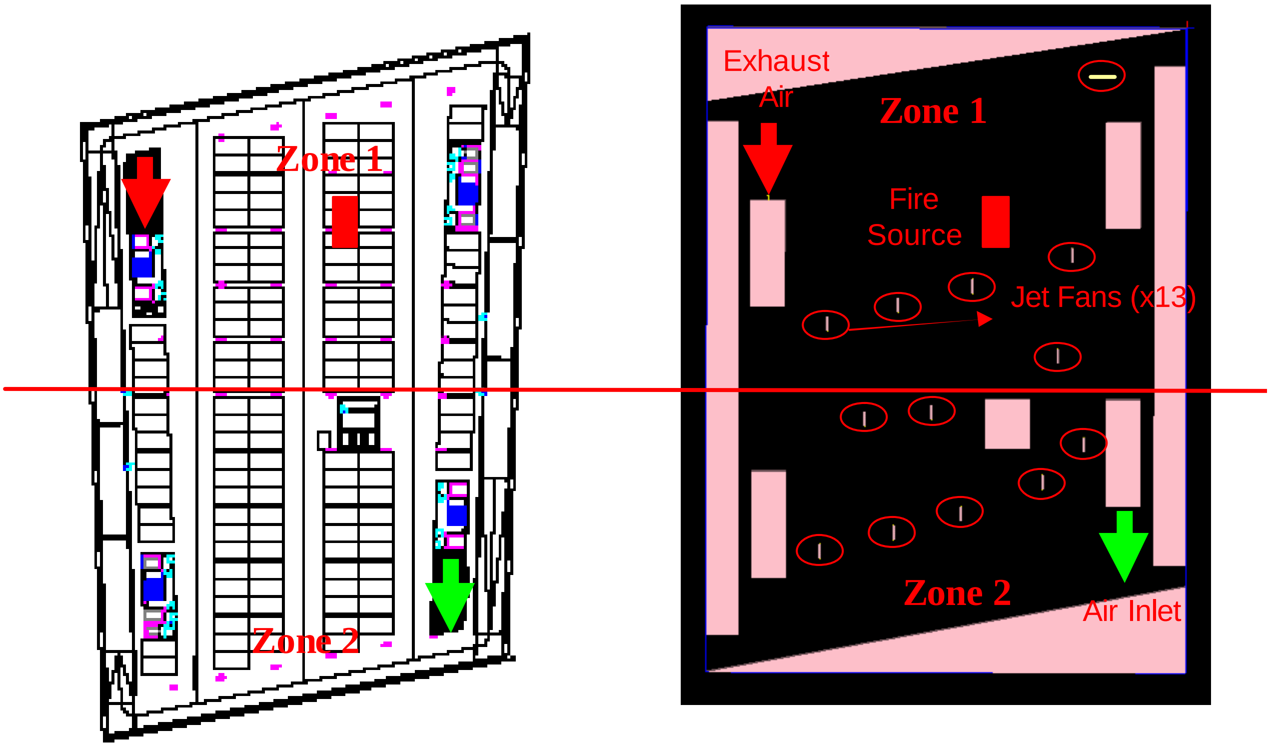

Figure 6 - Underground car park with identification of virtual smoke control zones, axial fans and jet fans. Fire is indicated by the red rectangle on zone 1.

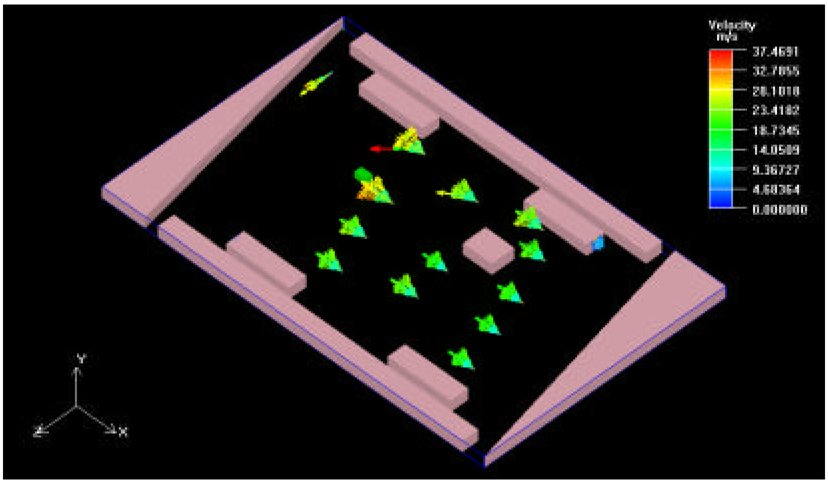

Figure 7 - Velocity vectors from jet fans and axial fans.

We can see jet fans with higher velocities and air inlet with a residual velocity. Air inlet velocity is very important avoid big vortices. Figure 8 shows importance of that.

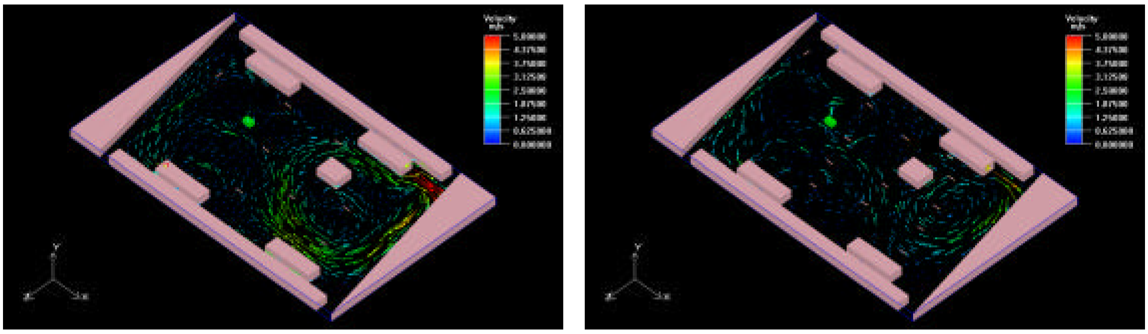

Figure 8 - Velocity pattern of 1.8m on Y axe.

On left, it's very clear existence of a big vortice that pass trough smoke zone and returns to clean zone. On figure we can see also this effect. Reducing the inlet velocity reduces such effect.

On next results we can compare the fire particles trace and temperature plan. We pass trough 5 simulation to rich the optimum. On next figures we can compare the fist result with the final result, it's an enormous difference. On our right we have final solution.

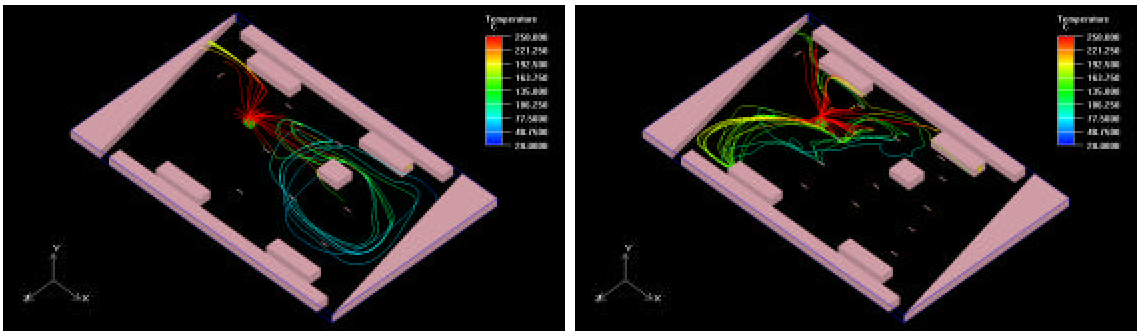

Figure 9 – Path of particles source

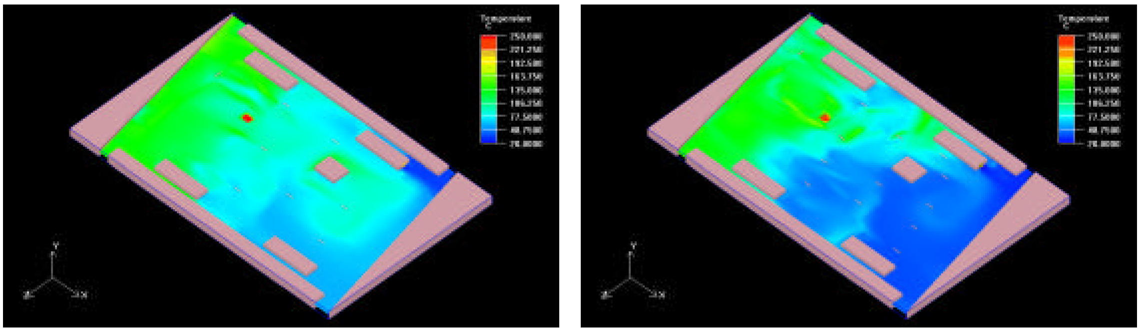

Figure 10 - Temperature in plane z=1.8m using steady numerical model.

The smoke are contained downstream of the local fire which allows the evacuation of occupants and ensures the appropriate environmental conditions for fire-fighters intervention with good visibility.

Control of CO concentrations was efficiency also like we can see on figure 11.

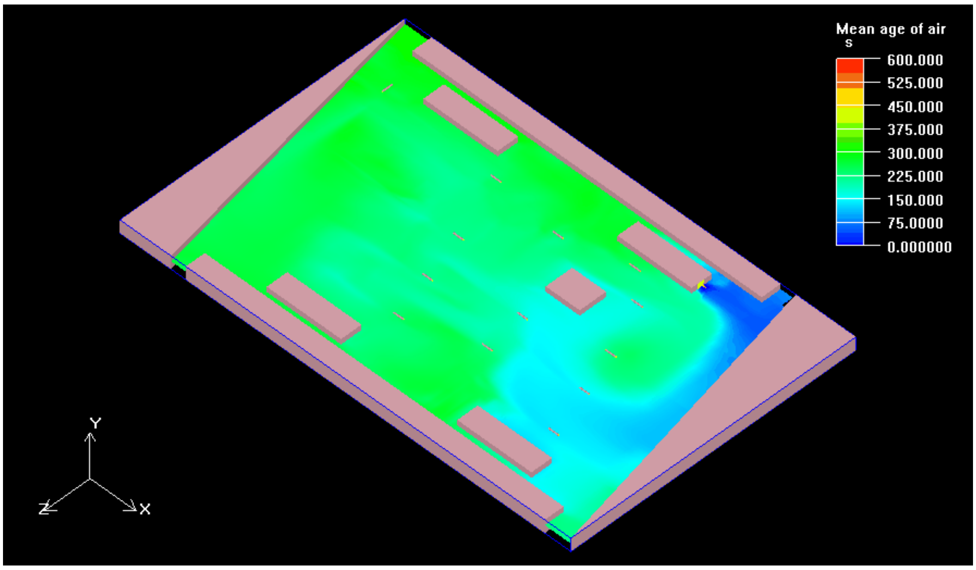

Figure 11 - Mean age of air in plane Y=1.8

It is guaranteed the escape of who is in the park and the access of the fire department to the disaster area. In this study there were not zones with stagnant air, so all park is well ventilated.

3.2 Steady flow simulations of smoke control system on zone 2

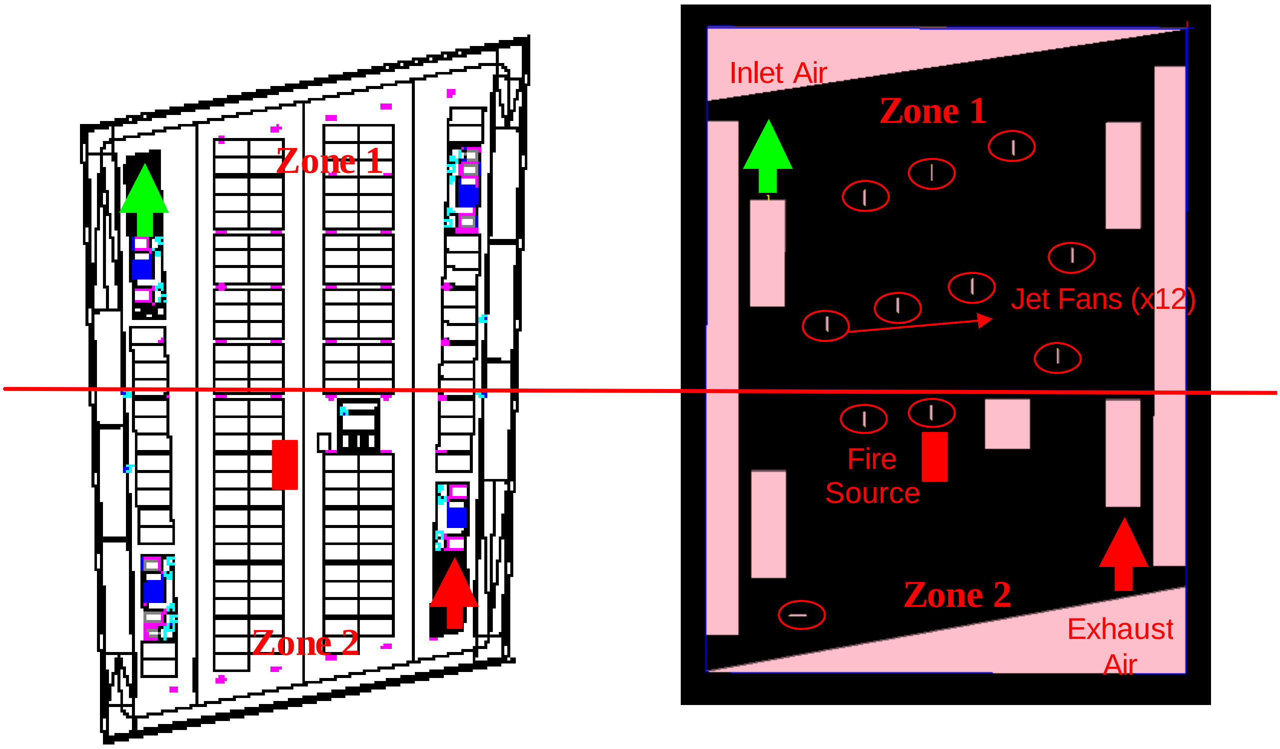

Figure 12 - Underground car park with identification of virtual smoke control zones, axial fans and jet fans. Fire is indicated by the red rectangle on zone 2.

When fire is detected on zone 2, axial fan V2 start exhaust and V1 start supply. After starts 12 jet fan units. CFD is used to verify the efficiency of the ventilation system in the same level -3, but now in reverse.

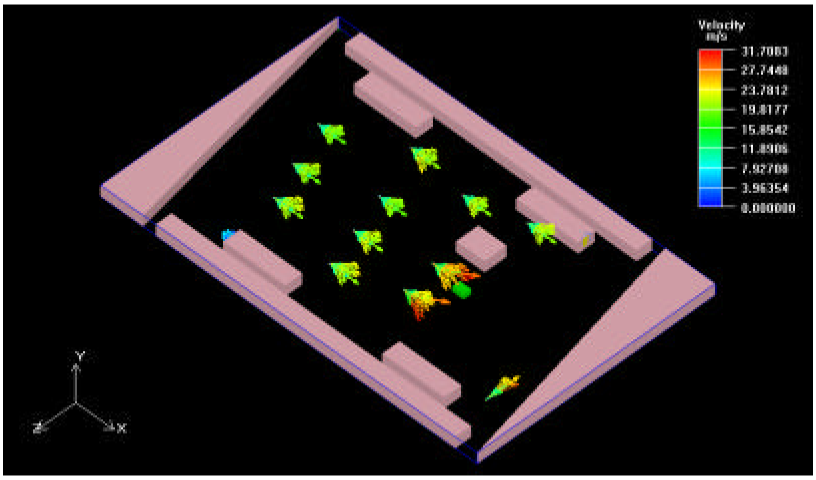

Figure 13 - Velocity vectors of jet fans and axial fans.

We can see jet fans with high velocities and air inlet with a residual velocity. Air inlet velocity is very important avoid big vortices.

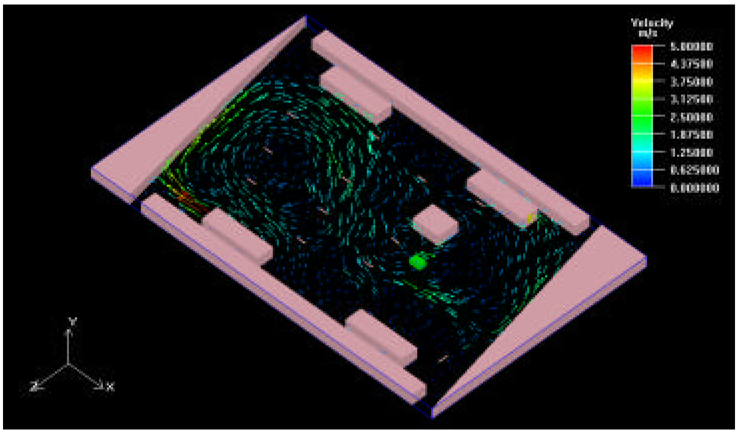

Figure 14 Velocity pattern of 1.8m on Y axe.

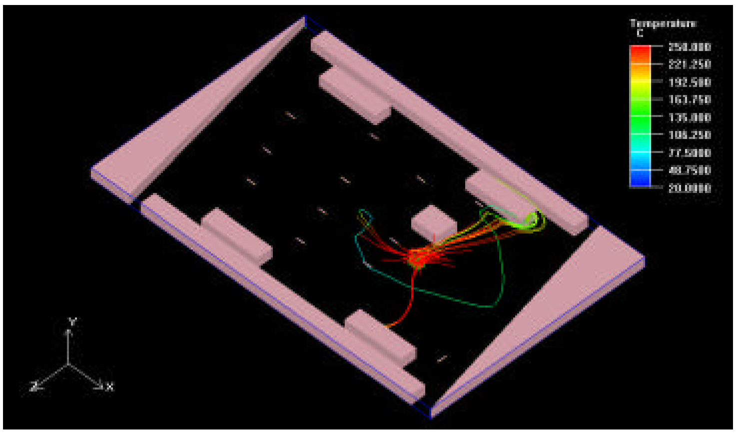

Figure 15 - Path of particles source

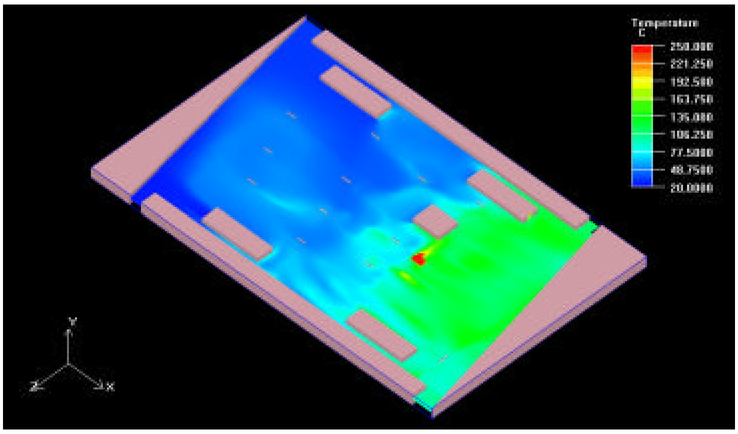

Figure 16 - Temperature in plane z=1.8m using steady numerical model.

The smoke are contained downstream of the local fire which allows the evacuation of occupants and ensures the appropriate environmental conditions for fire-fighters intervention with good visibility.

Control of CO concentrations was efficiency also like we can see on figure 17.

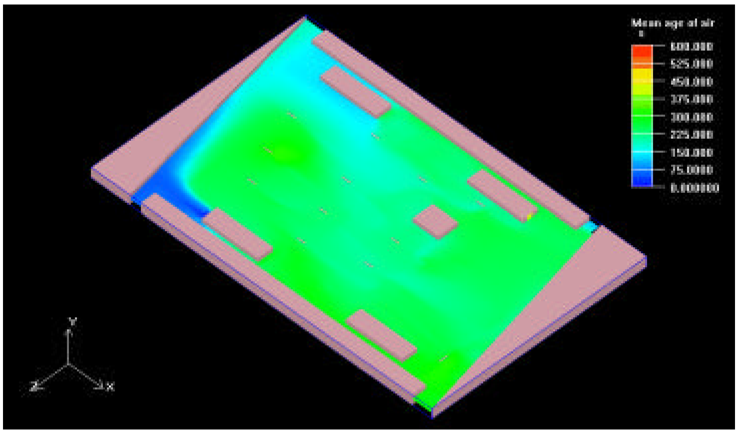

Figure 17 - Mean age of air in plane Y=1.8

4 VENTILATION SYSTEM WITH THE WIND EFFECT

It's very difficult to predict wind effect on system ventilation. When the project has mechanical extraction and natural air compensation, wind effect can change completely the smoke extraction efficiency. In this specific project, in normal conditions, the existing mesh of unidirectional jet fans should be enough to forward all the smoke to the extraction zone, but the wind effect was significant, not allowing the perfect ventilation.

The wind passing tangentially to the air vents, provokes Venturi effect, causing depression on openings and where we consider only for supply pass to have exhaust trough this openings.

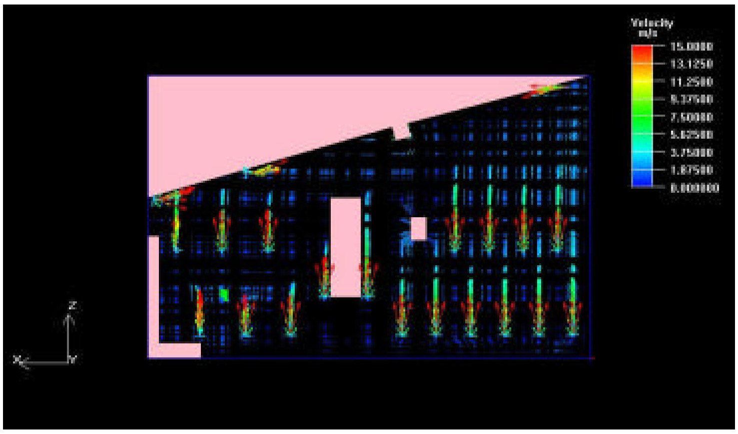

Figure 18 - Velocity vectors of jet fans and axial fans.

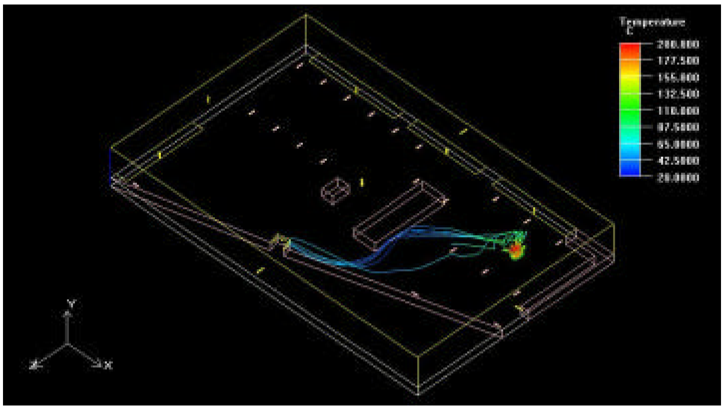

Figure 19 - Path of particles source without wind

As can be seen, fire though close to an opening, not having the effect of wind, all the particles are directed to the extraction.

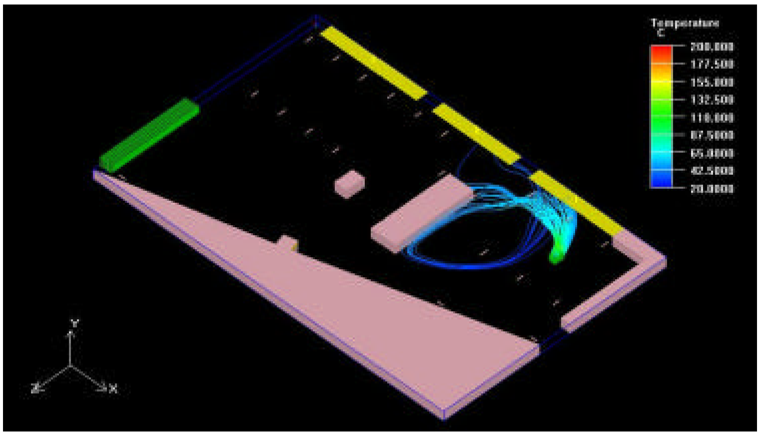

Figure 20 - Path of particles source with wind

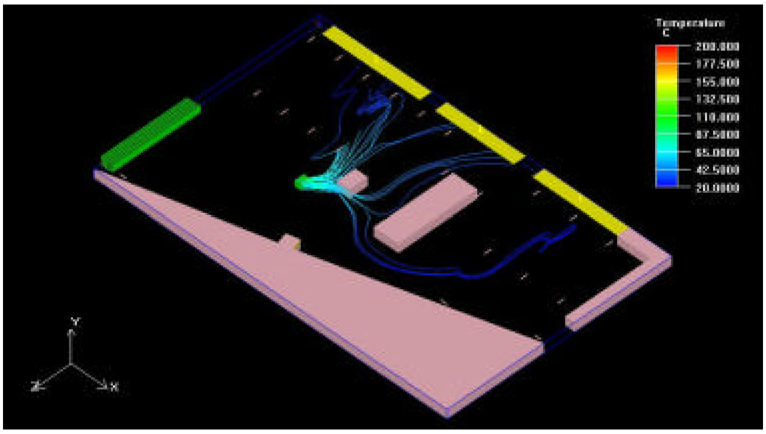

Figure 21 - Path of particles source with wind

As can be seen, fire though close to an opening or distant, having the effect of wind (1m/s on tree opening, only one to inlet air trough the car park), all source particles go to the openings even with jet fans working in the opposite way.

The solution passes to install anemometers to measure the velocity at the opening as well as its direction. With this statement could change the matrix to control the smoke extraction and act accordingly anemometer signal. If speed in the opening is greater than 1m / s with the upward direction, then the flow should work towards the natural openings, if the values are lower or if the wind direction is downward, we have been mechanically extract and compensation trough all openings.

Some images of the tests occurred in work with all entities responsible for passing the license to open the car park. It was proved that if it had underestimated the effect of wind, would not have made the array with the anemometer (figure 22) and therefore the park would not open.

Figure 22 - Anemometer

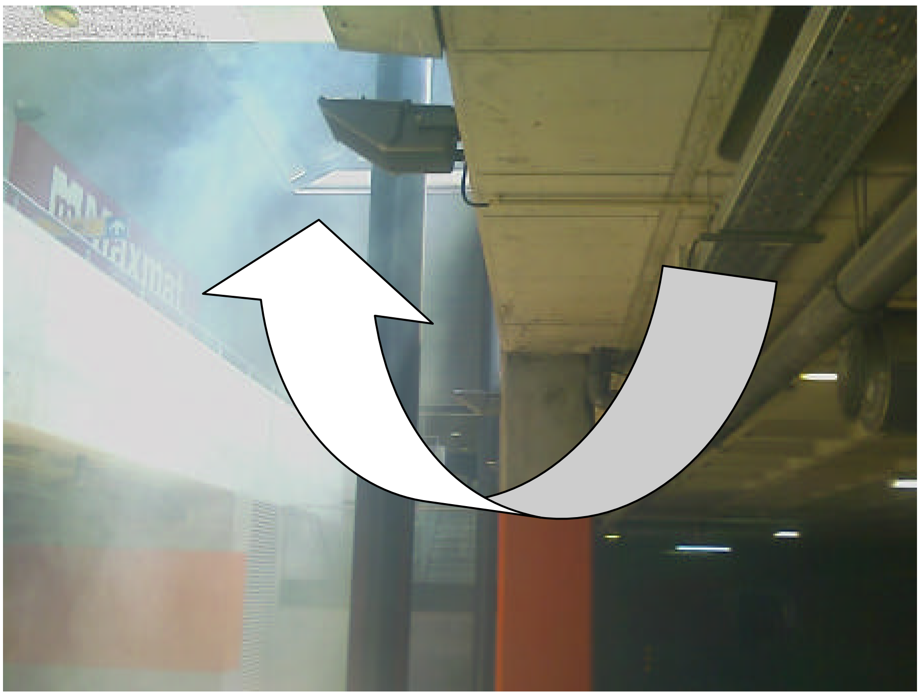

Figure 23 – Smoke through opening by wind effect

5 CONCLUSIONS

Computational Fluid Dynamics is a great tool to help engineers to predict the flow patterns and to verify the efficiency of ventilation system and smoke extraction. With this powerful tool, we can detect if spread of smoke occurs and if exist some death zones very critical to control CO concentration.

CFD ensures system optimization and, more importantly, that the occupants safety isn't compromised. This avoids making a project unnecessarily expensive by using too many fans, or an under performing system by specifying too few.

A solution consists developing a simple numerical jet fan model that represented the typical flow pattern produce by the fan.

With systems that makes possible extract smoke near fire source, 100% reversible systems help on a better control reducing the danger of spread smoke to other zones.

Numerical simulation CFD helped identify the interaction of the ventilation system with the wind effect in the openings of the inlet air. When the project has mechanical extraction and natural air compensation, wind effect can change completely the smoke extraction efficiency. In this specific project, in normal conditions, the existing mesh of unidirectional jet fans should be enough to forward all the smoke to the extraction zone, but the wind effect was significant, not allowing the perfect ventilation.

With the use of numerical simulation it was possible to obtain one array command able to react effectively whatever the effect of wind. It was concluded that it should be installed anemometers in the natural openings measuring speed and direction of air. This way, it was possible revert the direction of extraction. It could be made through the "original" extraction fans or through the natural openings.

Validation tests made in the job proved that the ventilation system wouldn't work safely for people and for the fire department if the effect of wind had not been considered.

REFERENCES

[1] Reulamento Geral Segurança contra incêndio em Edifício, Decreto Lei nº220/2008 de 12 de Novembro e Portaria nº 1532/2008 de 29 de Dezembro

[2] J.C. Viegas, Utilização de ventiladores de impulso em parques de estacionamento cobertos, In Proceedings of the *Congresso de Métodos Numericos en Ingeniería*, Barcelona, Spain (2009)

[3] J.P. Reeves, Car Parking Ventilation by Jet Thrust System, a comparison between ducted and impulse ventilation systems, Interne communication, Flak Woods, (2006)

[4] J.H. Ferziger and M. Peric, Computational Methods for Fluid Dynamics, New York, Barcelona, Hong Kong, London, Milan, Paris, Singapore, pp. 277-289 (1999)

[5] BS 7346-7 – Components for smoke and heat control systems –
Part 7 : Code of practice on functional recommendations and calculation methods for smoke and heat control systems for covered car parks.

[6] FLUENT 6.2 User´s Guide, Fluent Incorporated, (2005)

[7] GUIA FRANCE-AIR PORTUGAL 2009/2010

[8] France-Air, Ventilação de parques de Estacionamento Jet Thrust System, Internal report, France Air (2007)

[9] LNEC, Ensaios e análise comparativa de ventiladores de impulso, Laboratório Nacional de Engenharia Civil, Relatório 108/2009-NAICI (2009)

[10] Ministère de l' Equipement des Transports du Logement du Tourisme et de la Mer. Guide des Dossiers de sécurité des tunnels routiers, fascicule 4, Les Etudes Spécifiques des Dangers (ESD), Centre d'Etudes des Tunnels (2003)

[11] Precise Air Management – Car Park Ventilation - FLAKTWOODS