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

IMPULSE VENTILATION IN UNDERGROUND CAR PARKS: THE INFLUENCE OF PARKED CARS IN SMOKE CONTROL

João C. Viegas*

*Laboratório Nacional de Engenharia Civil
Av. do Brasil 101, P-1700-099 Lisboa
e-mail: jviegas@lnec.pt

Key words: Impulse ventilation, Underground car park, Jet fan, Smoke control, CFD

Abstract. *Impulse ventilation systems (IVS) are used to provide ventilation for underground car parks and to control the smoke in the event of fire. In this paper, the influence of the parked cars in the smoke control is studied using CFD simulations. A sensitivity analysis, considering important parameters as the number of parked cars and the height of the car park, is carried out. The results from a simplified analytical model for the flow field near the ceiling are compared with CFD simulations. It is shown that the obstruction caused by parked cars reduces the smoke control flow rate and the average velocity of the flow; therefore, it reduces the efficacy of the smoke control. It is also shown that the simplified model is able to reproduce approximately the boundary of the smoke. This simplified model is intended to support a first approach of the design of IVS.*

1 INTRODUCTION

The use of underground car parks is more and more often in cities. This type of spaces raises some problems to smoke control systems due to their big dimension but usual small height. Recently, the use of IVS was proposed to restrict the smoke spread in underground car parks allowing wider compartments without any smoke barrier. These systems generates the momentum necessary to retain the smoke (ceiling jet) flowing from the fire plume and are able to drive the smoke to the exhaust fans. The basic concept of the application of impulse ventilation to smoke control is coming from tunnel longitudinal ventilation. However, while in tunnels in a simple approach the flow may be considered one-dimensional, in underground car parks such approach is not possible because this space may be very wide. Therefore, very often CFD simulations are used in order to support the design of a ventilation and smoke control system.

A fire may occur with the car park full of cars or with just a small number of parked cars. Therefore, it is necessary to assume, for design, a simple fire scenario that should correspond to the worst case. Usually the fire scenario considers that there are no other parked cars than the car containing the fire source [2]. When parked cars are considered there is no sensibility study for the influence of the parked cars [3]. Although parked cars increase the friction losses of the flow, they restrict the cross section of the flow thus increasing the velocity above cars. This is the most important zone for smoke control because corresponds to the hot layer (close to the ceiling) that contain the smoke. Therefore *a priori* is not possible to predict the impact of the parked cars in the smoke control.

This study is included in a research on the performance of IVS in underground car parks. Previous work was done on the domain of ventilation [4] and smoke control [5]. In the domain of car park ventilation, it was concluded that jet fans may have a pumping effect entraining the surrounding contaminated air into the jet. In this way jet fans are able to restrict the dispersion of the contaminants, provided that the exhaust flow rate from the car park is not lower than the jet flow rate. Impulse ventilation may also provide fast dilution of the local pollution peaks, but in this case the removal efficiency will be low. In the domain of smoke control, the interaction between the fire ceiling jet and the flow driven by jet fans was studied using CFD simulations. A sensitivity analysis considering important parameters as position and intensity of fire source, transversal distance between jet fans, restriction of exhaust flow rate and dimension of car park exhaust opening was carried out and rules for the design of 50 N thrust jet fans were deduced. An analytical model for the flow field near the ceiling was developed and compared with CFD simulations. This model is intended to support a first approach of the design of IVS and is also used in the work included in this paper.

In both cases the work done was the first approach to the problem; therefore, the simulations and the calculations were carried out just with the car park without any car. At this stage, the problem of the effect of the parked cars on the flow shall be considered. In order to assess the influence of the parked cars on the smoke control, when an impulse ventilation system is used, a set of CFD simulations (using the FDS computer code [1]) with different patterns of parked cars were made and compared with a simulation without cars. As the area of the cross section of the flow above the cars is also important, the influence of the height of the car park was also studied.

2 METHODOLOGY

Being the purpose of this work to assess the impact of parked cars in the performance of an IVS with a low computational cost, a relatively coarse grid was adopted. The same calculation domain adopted in this study and described hereafter was used in previous works [4 and 5]. It was shown that the validity of the simulations was acceptable and CFD simulations were used to show the influence of several parameters in the performance of an IVS. Having in mind that the validity of simulations in relatively coarse grids is questionable, the methodology was based on a sensitivity analysis where just one relevant parameter was changed from simulation to simulation. Therefore, the conclusions emphasized mainly the differences obtained from simulation to simulation as tendencies related with the changing parameters.

The same approach is followed here. In the first part of this study, one parameter is changed in each simulation, allowing a sensitivity analysis for that parameter. The calculation domain is large and is supposed to reproduce part of a big underground car park. It includes fifteen jet fans suspended under the ceiling and disposed according to a grid commonly used by designers in car parks. In these simulations only the impact of the more important parameters is assessed and conclusions to application of IVS to underground car parks are presented.

In the second part of the study, analytical expressions, previously established [5], are used to describe the limit imposed by IVS to the smoke flow. In a fire scenario, the mixing between the smoke layer and the lower cold layer due to IVS is expressed in this analytical model as a variable, so called "dilution parameter". It was shown that the adequate choice of its value is important to obtain an acceptable agreement between analytical model results and CFD simulations results. As no experimental data are available, the value of this parameter was previously obtained as the best fit between analytical and CFD simulations results and its value is $D = 0.001$ [5]. A short description of this model is in annex.

The computer code Fire Dynamics Simulator [1] was used to carry out CFD simulations. This CFD code uses Large Eddy Simulation turbulence model (LES). The dispersive character of the swirling jet from jet fans was introduced in the simulation using FDS [1] by the increment of the diffusion parameter to $C_S = 0.40$ in LES model. As the calculation domain is very large and the jet fans cause the dilution of the smoke, the temperature and smoke density in the beginning of the fire (the phase studied here, that is relevant to smoke control) are low. Therefore, is acceptable to perform the CFD simulations without using a radiation heat transfer model. As the radiation heat losses from fire source in the initial phase of fire are about 1/3 of the heat release rate, the simulated 4 MW convective heat release rate fire source corresponds approximately to a 6 MW total heat release rate fire source. The computer code is transient and the simulations were running until steady state was reached. In general, steady conditions were obtained before the time t = 300 s was reached. Nevertheless, the simulations were carried out up to t = 600 s. The results presented in this paper correspond to steady conditions (t = 600 s).

The validation of the simulation of jet fans in isothermal flow for the calculation grid used here and with the simulation parameters adopted was presented in [4] and the validation of the heat source simulation (under the same conditions) was presented in [5].

In the sensitivity analysis the following parameters are studied:

- Difference between IVS used for smoke control in a low park (2.30 m high) and in a higher car park (3.45 m high) (simulations I107 and I124);
- Influence of the parked cars in the smoke control of a low car park (2.30 m high) (simulations I107 and I122);
- Influence of the parked cars in the smoke control of a high car park (3.45 m high) (simulations I124 and I125);

- Influence of the number of parked cars in the smoke control of a low car park (2.30 m high) (simulations I107, I122 and I133).

The basic geometry simulated (simulation I107) is a space 95.0 m long (the centrelines of the jets are longitudinal), 75.0 m wide and 2.3 m height (figure 1). Fifteen 45 N thrust jet fans are placed close to the ceiling. The calculation domain contains 285*225*8 cells (x*y*z). One lateral boundary (parallel to the centrelines of the jets – the right lower boundary in the figure 1 containing the heat source) is a symmetry plan. The other lateral boundary, the upstream boundary and the downstream boundary are openings to outside. The horizontal ones (ceiling and floor) are impermeable boundaries, as before. The heat source was placed in the symmetry plan (centred at x = 41.0 and with an area of 2 m²), therefore only half part (2 MW) of the power previously considered was adopted. Due to the adoption of a symmetry plan, this calculation domain represents a car park zone 150 m wide, with thirty jet fans and with the total convective power of 4 MW.

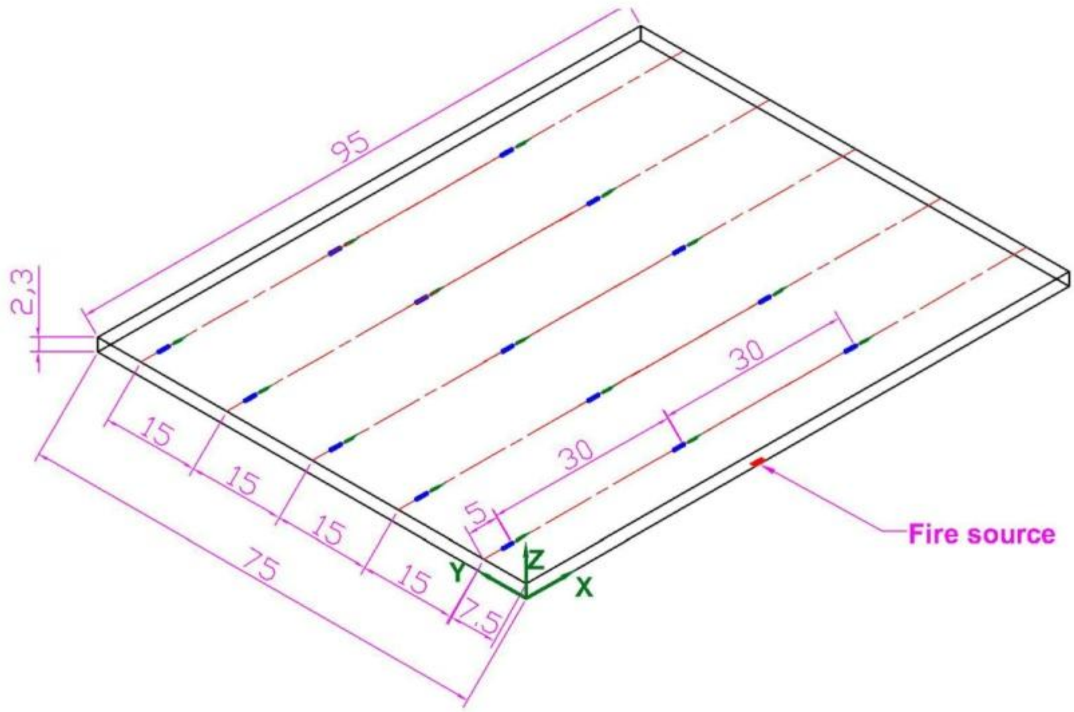


Figure 1: Calculation domain and placement of the jet fans [dimensions in m].

In the case of the simulation I124, the calculation domain is exactly the same as for the simulation I107 except that the height of the car park is now 3.45 m. This change obliged to use a higher number of cells along z axis in order that the spatial discretization was kept. Therefore, the calculation domain contains 285*225*12 cells (x*y*z).

In the case of the simulation I122 (figure 2), the car park of simulation I107 was full of cars. Ten rows of thirty vehicles were added, that corresponds to an occupation efficiency of 23.75 m²/vehicle, which is even lower than the best occupancy ratio for big car parks (that stands about 25 m²/vehicle). Every car is 4.00 m long, 1.50 m high and 1.70 m wide. The lateral distance between cars is 0.80 m and the longitudinal distance between cars is 1.00 m. The routes for cars are 11.00 m wide.

In the case of the simulation I125, the car park of simulation I124 was simulated full of cars, having the same pattern as described for simulation I122.

In the case of the simulation I133, the occupancy of the car park is just half part of the maximum. The calculation domain is equal to the simulation I122 calculation domain but in every row alternately a parked car was taken out (see figure 3).

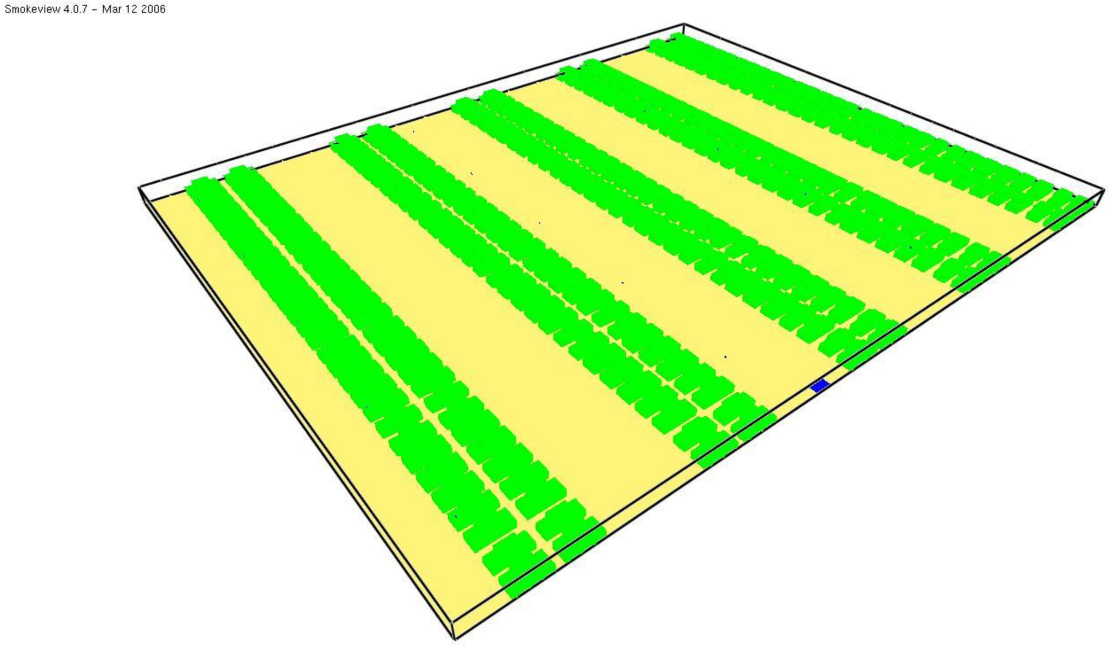


Figure 2: Position of the parked cars in simulation I122

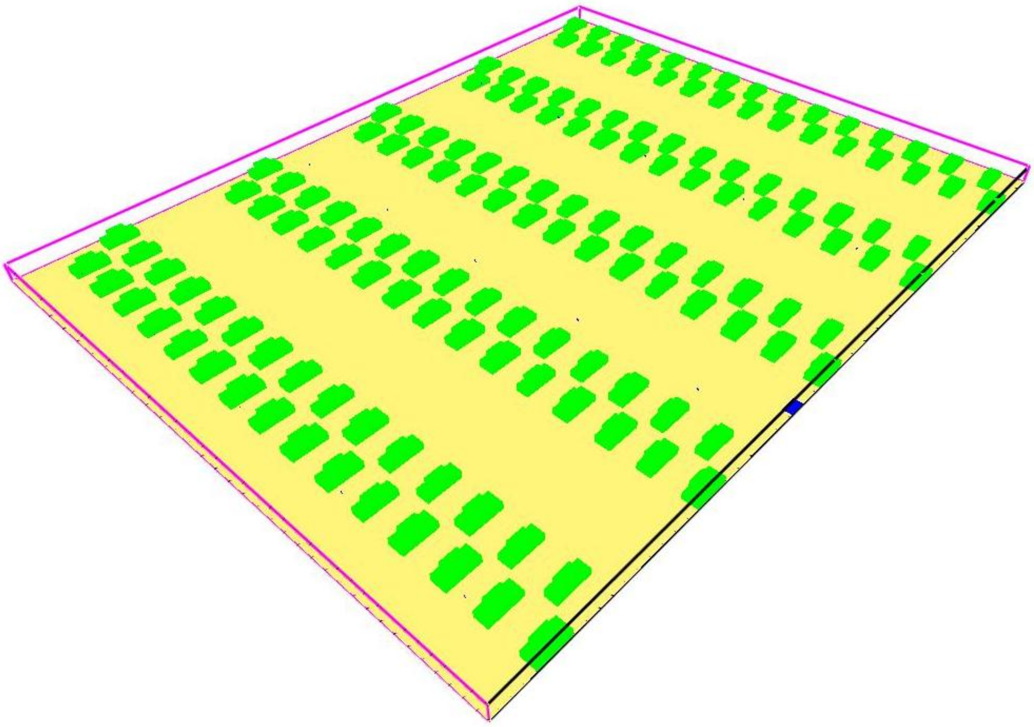


Figure 3: Position of the parked cars in simulation I133

In the work presented in this paper, the flow velocity opposing to the fire plume ceiling jet is due to the thrust of jet fans. This represents a big car park were the envelope walls are very far or a car park opened to outside. For the most common car parks the internal flow rate is imposed by exhaust fans. As it is expected that in this last case the exhaust flow rate is not so affected by parked cars obstructions, the simulations presented hereafter are supposed to correspond to the worst case.

3 ANALITICAL FORMULATION OF PHISICAL PROCESS

Two physical processes are relevant in this study: the submerged turbulent jet produced by a jet fan and the turbulent ceiling jet formed from the impinging point of a turbulent fire plume to the ceiling.

The analytical solution for the velocity field of submerged turbulent axisymmetric jets [6 and 7] considers that the flux of axial momentum of a jet is conserved; therefore, the flows at various axial distances from the nozzle are dynamically similar, if adequately nondimensionalized. The following equation results from that hypothesis and expresses the velocity field u(x,r) of a single submerged and not constrained jet, being x the axial coordinate, r the radial coordinate, u_c the centreline velocity, u_0 the velocity at the nozzle, r_0 the nozzle radius and k_0 and k the model constants:

$$u(x,r) = u_c e^{-\frac{1}{2}\left(\frac{r}{kx}\right)^2} = \frac{k_0 u_0 2r_0}{x} e^{-\frac{1}{2}\left(\frac{r}{kx}\right)^2} \quad (1)$$

Experimental results for single unconfined jet produced by a commercial jet fan used in underground car parks show that $k = 0.090$ and $k_0 = 5.41$ in the direction parallel to the nozzle guide vanes and that $k = 0.091$ and $k_0 = 5.60$ in the direction perpendicular to these guide vanes, showing that, for this particular case, there is no strong influence of the nozzle guide vanes in jet flow field. Moreover, the k values for this particular case are close to the expected k values for the unconfined jet without swirl ($0.072 < k < 0.085$), showing in this case limited influence of the rotation in the expansion of the jet.

In co-flow, the behaviour of the jet may be considerably modified, depending on the ratio u_0/u_∞ (being u_∞ the velocity of the co-flowing stream). Hinze et al. [9] showed that only when $u_\infty >> u_0$ and $u_\infty << u_0$ the similarity for sections of the jet flow at different distances from the nozzle is kept, thus enabling to use the general equation:

$$u^2 = u_\infty^2 + \left(2k_0 \frac{r_0}{x}\right)^2 \left[u_0(u_0 - u_\infty)\right] e^{-E_3\left(\frac{r}{b}\right)^2} \quad (2)$$

E_3 is a coefficient and b is the radius of the jet.

In the application to underground car parks, usually $u_0 > 20\ \mathrm{m/s}$ and $u_\infty < 1.0\ \mathrm{m/s}$; therefore, the conditions referred by Hinze et al. [8] are respected.

It was showed also [6] that, in the case of multiple jets, the flow field may be expressed by:

$$u^2(x,r,\theta) = \sum_{i=1}^{n} \left(\frac{k_{0\,i} u_{0\,i} 2r_{0\,i}}{x - x_i}\right)^2 e^{-\left(\frac{r_i}{k_i(x-x_i)}\right)^2} \quad (3)$$

The summation index i refers to every jet, r_i is the local radial coordinate and θ is the polar coordinate. Combining the previous equations, the following general solution may be obtained. It expresses the velocity field of n combined jets of nozzle velocity

u_{0i} in a co-flow of uniform velocity u_∞, nozzle radius r_{0i}, axial coordinate x, jet origin x_i, and model constants k_{0i} and k_i:

$$u_{impulse}^2(x,r,\theta)=u_\infty^2+\sum_{i=1}^{n}\left(\frac{k_{0i}2r_{0i}}{x-x_i}\right)^2\left[u_{0i}\left(u_{0i}-u_\infty\right)\right]e^{-\left(\frac{r_i}{k_i(x-x_i)}\right)^2} \tag{4}$$

It is possible to obtain an approximate solution of the confined jet pattern (wall jet) in co-flow if another symmetric jet is used to establish a symmetry plan coincident with the wall. However, this approach is not able to reproduce accurately the effect of the wall friction in the flow or the effect of the restriction of the exhaust flow rate.

According to Alpert [9], the temperature and velocity field of an unconfined ceiling jet from a turbulent fire plume is given by the following equations:

$$\Delta T = T - T_\infty = \frac{16{,}9\dot{Q}^{2/3}}{H^{5/3}} \qquad \text{for } r/H \le 0{,}18 \tag{5}$$

$$\Delta T = T - T_\infty = \frac{5{,}38\left(\dot{Q}/r\right)^{2/3}}{H} \qquad \text{for } r/H > 0{,}18 \tag{6}$$

$$u_{cj} = 0{,}96\left(\dot{Q}/H\right)^{1/3} \qquad \text{for } r/H \le 0{,}15 \tag{7}$$

$$u_{cj} = \frac{0{,}195Q^{1/3}H^{1/2}}{r^{5/6}} \qquad \text{for } r/H > 0{,}15 \tag{8}$$

T is the ceiling jet temperature, T_∞ is the ambient temperature, $\dot{Q}$ is the heat release rate (including the radiated fraction and in kW), H is the plume height, r is the radial distance from the impinging point of the plume into the ceiling and u_{cj} is the ceiling jet velocity. These equations were obtained theoretically and were adjusted taking into account test results. They are not applicable when a smoke hot layer is stratified below the ceiling jet. These equations were derived for the case where the heat source is far from the walls (distance bigger than 1.8 times the plume height).

4 SIMULATIONS RESULTS

4.1 Influence of the height of the car park

The influence of the height of the car park may be assessed through the comparison of the simulations I107 (2.30 m high) with I124 (3.45 m high). In figures 4 and 6 is possible to compare the field of temperature 0.29 m below the ceiling for both simulations. It is possible to see that in the case of simulation I107 the zone with temperature higher than 20ºC (the initial condition of ambient temperature) is not reaching the upstream boundary (the left side border in the figures). This means that the smoke flow is limited by the flow imposed by jet fans. It is seen for simulation I124 that the zone with temperature higher than 20ºC is already reaching the upstream border. In this case the smoke flow is not limited in the calculation domain. The analysis of the u-velocity flow field, shown in figures 5 and 6, confirms this conclusion because the u-velocity close to upstream border is always positive in simulation I107 and in simulation I124 there are some zones were u-velocity is negative. Figure 7 show that the velocity of the fire plume ceiling jet is higher in the case of the simulation I124 than in the simulation I107. In figure 8 it is shown that, in consequence, the temperature close to the ceiling is also higher in the simulation I124 when compared with simulation I107.

Equation (8) shows that for a higher plume the radial velocity of the fire plume ceiling jet is also higher, that explains the behaviour observed. Moreover, table 1 shows that although the flow rate in the cross section corresponding to x = 0.33 is higher for simulation I124 than for simulation I107, the flow average velocity is smaller because the area of the cross section is higher. Thus, the flow average velocity of the flow entrained by jet fans is lower than the fire plume ceiling jet velocity and jet fans are not able to restrict the smoke.

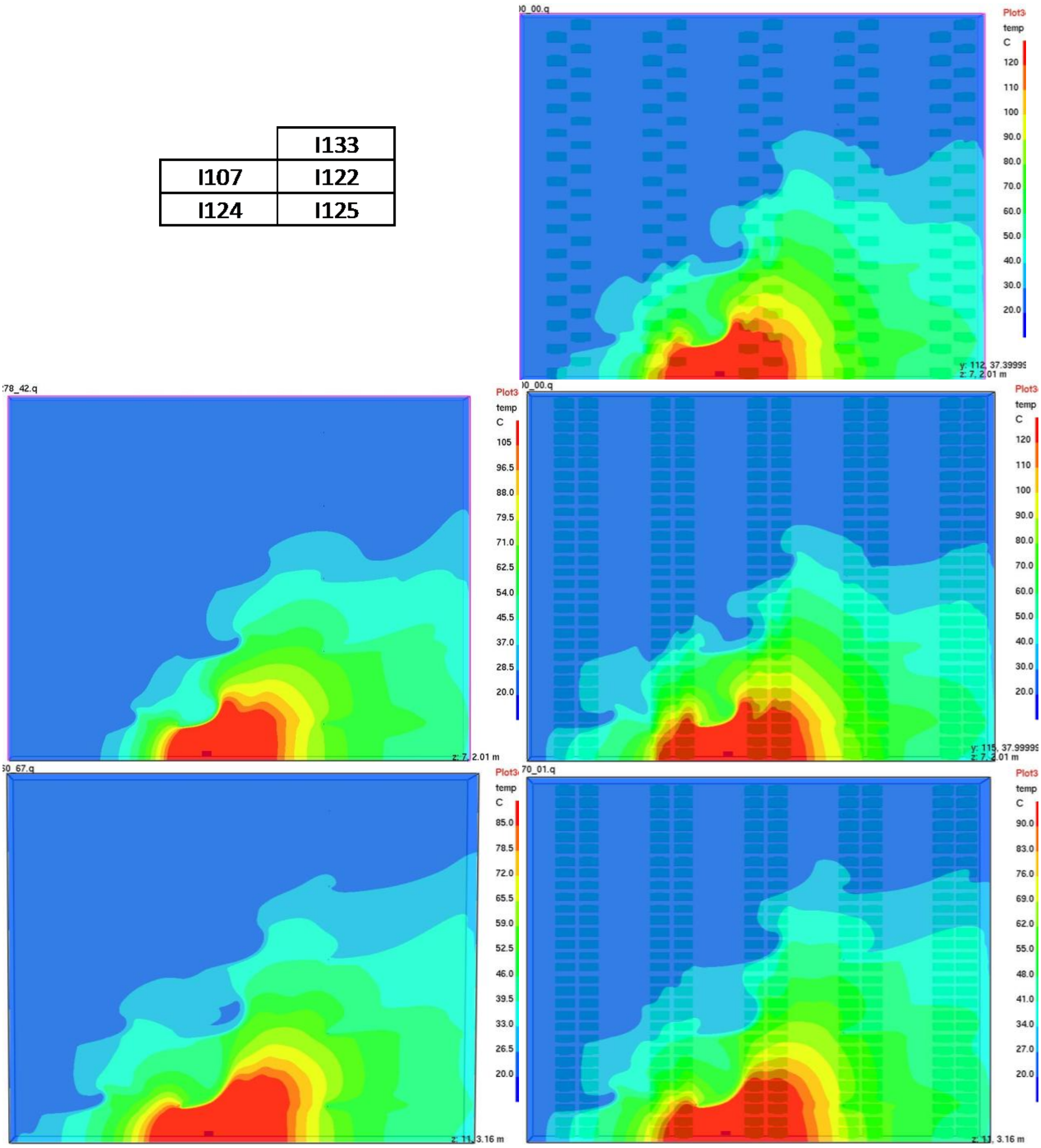


Figure 4: Horizontal cross section of temperature field at z = 2.01 m (simulations I107, I122 and I133) and z = 3.16 m (simulations I124 and I25)

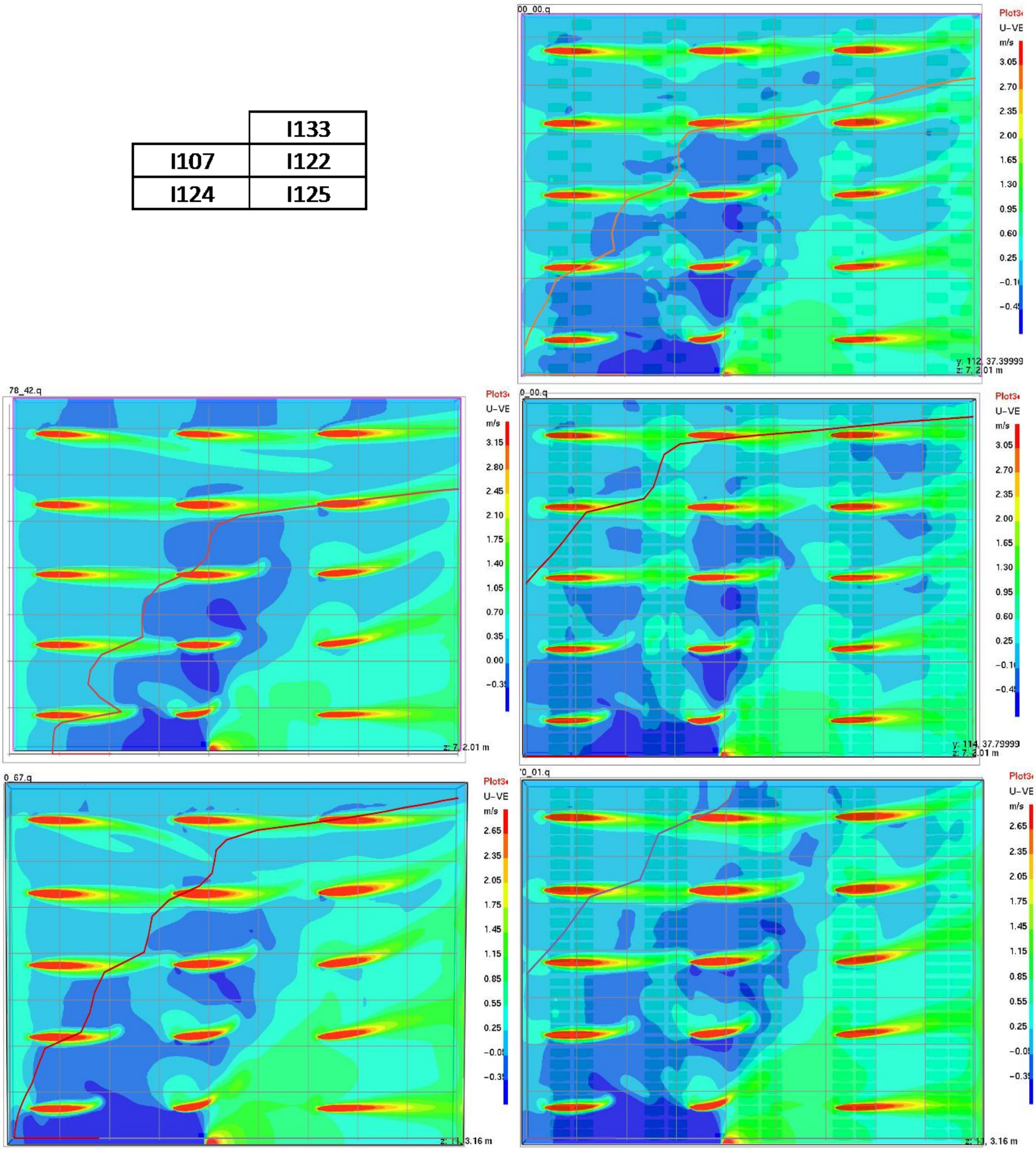


Figure 5: Horizontal cross section of U component of the velocity field at z = 2.01 m (simulations I107, I122 and I133) and z = 3.16 m (simulations I124 and I25)

4.2 Influence of the parked cars in the smoke control of a low car park

The influence of the parked cars in the smoke control of a car park with low height (2.30 m high) may be assessed through the comparison of the simulations I107 (no parked cars) with I122 (300 parked cars). Analyzing figures 4 and 6 it is seen for simulation I122 that the zone with temperature higher than 20ºC is almost reaching the upstream border. The analysis of the u-velocity flow field, shown in figures 5 and 6, shows that the u-velocity close to upstream border is negative in some zones of simulation I122. In this case the smoke flow is not limited in the calculation domain. Figure 7 and figure 8 confirm these observations.

Table 1 show that the flow rate in the cross section corresponding to x = 0.33 is much smaller for simulation I122 than for simulation I107. This reduction is due to friction losses developed by the parked cars into the flow. Thus, the flow average velocity is smaller than the fire plume ceiling jet velocity and the flow entrained by jet fans is not able to restrict the smoke.

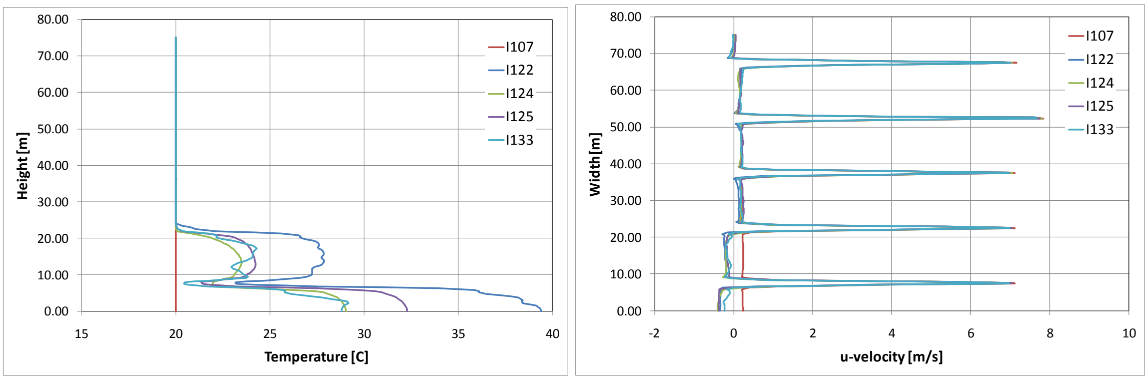


Figure 6: Profiles of temperature and u component of the velocity at x = 7.00 m and at z = 2.01 m (simulations I107, I122 and I133) and z = 3.16 m (simulations I124 and I25)

Simulation	Flow rate [m³/s]	Average velocity [m/s]	Cross section [m²]
I107	62.7	0.36	172.5
I122	31.2	0.18	172.5
I124	80.2	0.31	258.7
I125	57.9	0.22	258.7
I133	42.0	0.24	172.5

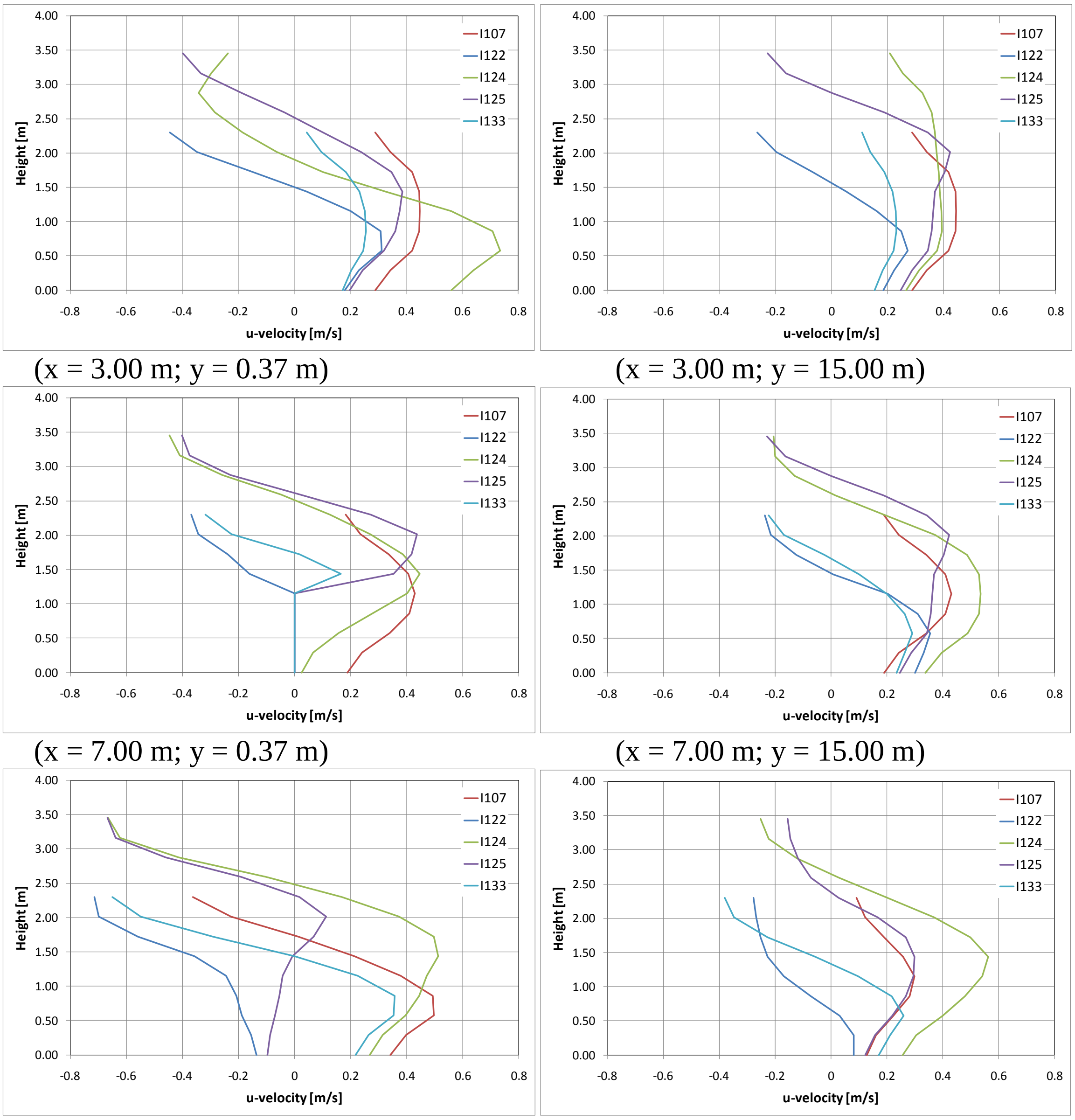
Table 1: Flow rate and average u-component of the velocity at the cross section x = 0.33.

4.3 Influence of the parked cars in the smoke control of a high car park

The influence of the parked cars in the smoke control of a higher car park (3.45 m high) may be assessed through the comparison of the simulations I124 (no parked cars) with I125 (300 parked cars). Analyzing figures 4 and 5 it is seen that there are no significant differences between simulations I124 and I125. The detailed analysis of the flow fields, shown in figure 6, confirms that there is no significant difference for the u-velocity field. In both cases the smoke flow is not limited in the calculation domain. Figure 7 and figure 8 confirm these observations.

Even so, the worst case corresponds to simulation I125 because, as shown in Table 1, the flow rate in the cross section corresponding to x = 0.33 is smaller for simulation I125 than for simulation I124. As before, this reduction is due to friction losses developed by the parked cars into the flow.

It is possible to conclude from these observations that when the gap between the top of the cars and the ceiling is bigger the influence of the obstruction to the flow due to parked cars is less significant.



(x = 3.00 m; y = 0.37 m) (x = 3.00 m; y = 15.00 m)

(x = 7.00 m; y = 0.37 m) (x = 7.00 m; y = 15.00 m)

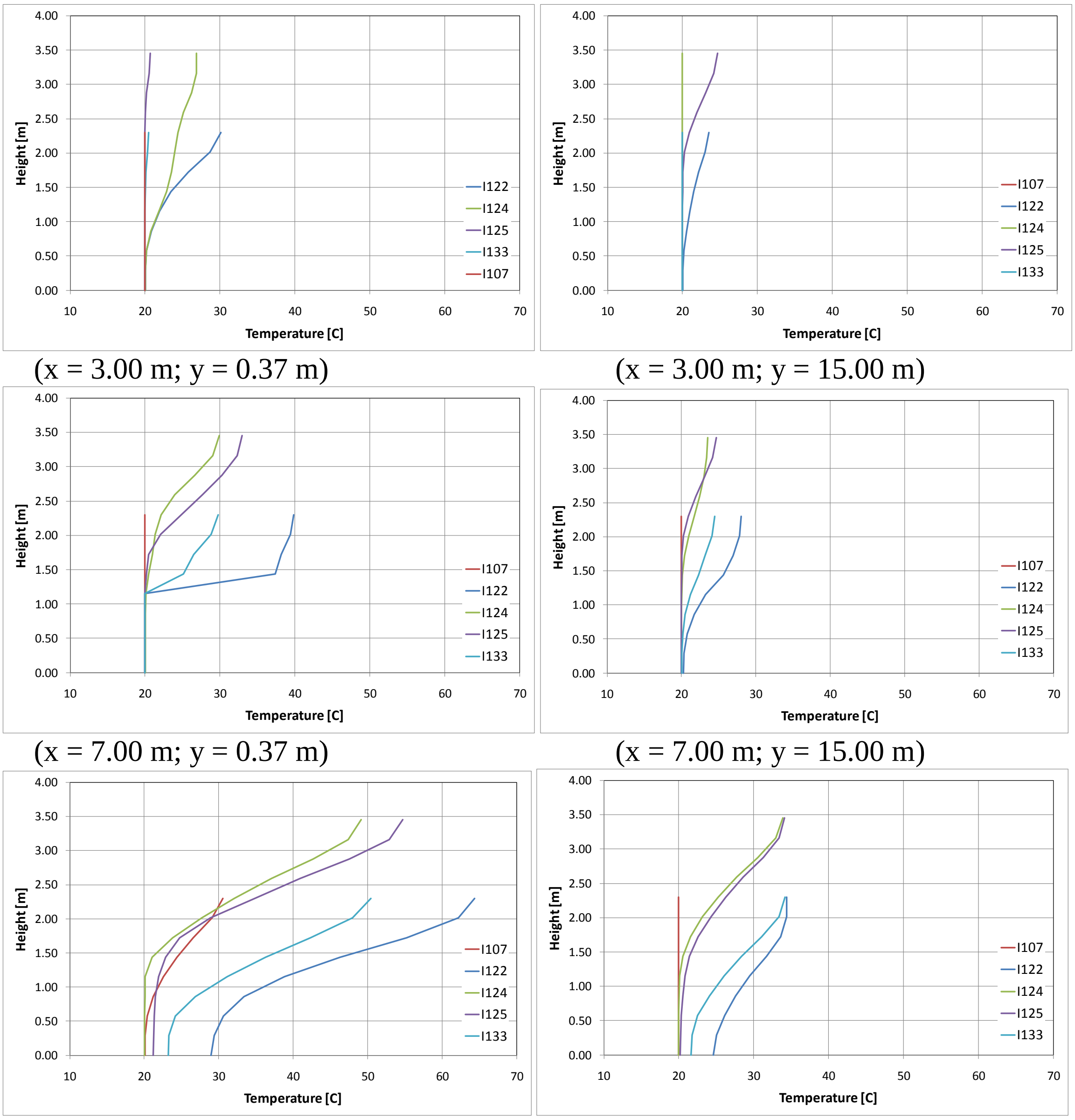
(x = 19.33 m; y = 0.37 m) (x = 19.33 m; y = 15.00 m)

Figure 7: Vertical profiles of u component of the velocity for simulations I107, I122, I124, I125 and I33

4.4 Influence of the number of parked cars in the smoke control

The influence of the number of parked cars in the smoke control of a car park may be assessed through the comparison of the simulations I107 (no parked cars) with I122 (300 parked cars) and I133 (150 parked cars). Analyzing figures 4 and 5 it is seen that the results of the simulation I133 are standing between the results obtained for simulations I107 and I122. For simulation I133 the backlayering is almost stopped near the upstream boundary. The detailed analysis of the flow fields, shown in figure 6, confirms that there is a zone at upstream border where the u component of the velocity is very close to zero for simulation I133. For this case, the smoke flow is limited in the calculation domain. Figure 7 and figure 8 confirm these observations.

Table 1 shows a significant reduction of the incoming flow rate due to parked cars from simulation I107 to simulation I133. This reduction is more important that the one observed from simulation I133 to I122. In fact, the cars added to the simulation I122 are in line and close to the parked cars considered in the simulation I133; therefore, the obstruction to the flow caused by the cars added in simulation I122 is smaller.



(x = 3.00 m; y = 0.37 m) (x = 3.00 m; y = 15.00 m)

(x = 7.00 m; y = 0.37 m) (x = 7.00 m; y = 15.00 m)

(x = 19.33 m; y = 0.37 m) (x = 19.33 m; y = 15.00 m)

Figure 8: Vertical profiles of temperature for simulations I107, I122, I124, I125 and I33

4.5 Results of the analytical model

To apply the analytical model the term u_∞ assumed the average velocity values presented in table 1.

Horizontal cross sections of u component of the velocity field, for simulations I107, I122; 124, I125 and I133, are presented in figure 5 and the curves representing the analytical less favourable streamline (starting at the symmetry plan upstream from fire plume) are included. The analytical curves represent the smoke flow limit. It may be seen that the analytical curves are close to the zero iso-velocity line. Usually the approximation obtained with the analytical model is conservative. It is possible to conclude that the simplified analytical model may be used as a first approach to the IVS engineering design.

5 CONCLUSIONS

- Due to the height of the fire plume, for a higher underground car park the velocity of the fire plume ceiling jet is higher. For the same set of jet fans the flow average velocity is smaller because the area of the flow cross section is higher. Thus, is more difficult to restrict the smoke for higher car parks.
- It is shown that the obstruction caused by parked cars reduces the smoke control flow rate and the average velocity of the flow; therefore, reduces the efficacy of the smoke control for the studied cases.
- When the gap between the top of the cars and the ceiling is bigger, the influence of the obstruction to the flow due to parked cars is less significant. Thus, the influence of the parked cars on the smoke control is also less significant.
- As expected, when the number of parked cars decreases, the friction losses due to the car obstruction to the flow decreases and the average velocity of the flow entrained by jet fans increases. Therefore, the restriction to the smoke flow is closer to the fire source.
- It is also shown that the simplified model is able to reproduce approximately the boundary of the smoke. This simplified model is intended to support a first approach of the design of IVS.
- In the work presented in this paper, the flow velocity opposing to the fire plume ceiling jet is due to the thrust of jet fans. Further work on this subject should analyze the effect of the parked cars when smoke exhaust is carried out by exhaust fans. In this case, depending on the characteristic curve of the exhaust fans, it is expected that the exhaust flow rate is not so affected by parked cars obstructions.

ACKNOWLEDGEMENTS

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

REFERENCES

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

[2] 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).

[3] X. G. Zhang et al., Numerical simulations on fire spread and smoke movement in an underground car park, Building and Environment, Elsevier Science Ltd. 42, pp. 3466–3475 (2007).

[4] 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)

[5] 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.

[6] D.X. Viegas, Uma técnica de erosão aplicada ao estudo da interacção de jactos turbulentos com uma superfície plana, Coimbra (1981). PhD Thesis.

[7] R.D. Blevins, Applied fluid dynamics handbook, Van Nostrand Reinhold Company (1984).

[8] J.O. Hinze and B.G. Zijnen, Transfer of heat and matter in the turbulent mixing zone of an axially symmetrical jet, *Journal Appl. Sci. Res.* A1 (1949).

[9] R. L. Alpert, Fire Technology, 8, 181 (1972).

ANNEX – DESCRIPTION OF THE ANALYTICAL MODEL

Combining eqs. (4) and (8) it is possible to obtain an analytical model to represent the velocity field near the ceiling. Due to buoyancy it is supposed that the smoke affected zone is wider near the ceiling. This analytical model includes the basic physics needed to predict approximately the maximum limit of the smoke layer. However, it does not include the physics needed to reproduce the recirculation zones that may be developed near jet fans, when exhaust rate is limited. Eqs. (9) to (12) represent this model, being (x;y) = (0;0) the coordinates of the impinging point of the fire plume to the ceiling:

$$r(x,y)=\sqrt{x^2+y^2} \tag{9}$$

$$u_{flow}=\left(u_{cj}\frac{x}{r}At_v\right)\left|u_{cj}\frac{x}{r}At_v\right|+u_\infty^2+\sum_{i=1}^{n}\alpha_i\left(\frac{k_{0i}u_{0i}2r_{0i}}{x-x_i}\right)^2 e^{-\left(\frac{r_i}{k_i(x-x_i)}\right)^2} \tag{10}$$

$$u(x,r)=\begin{cases}\sqrt{u_{flow}} & \text{if} \quad u_{flow}\geq 0\\ -\sqrt{-u_{flow}} & \text{if} \quad u_{flow}<0\end{cases} \tag{11}$$

$$v(y,r)=\frac{0{,}195Q^{1/3}H^{1/2}}{r^{5/6}}\frac{y}{r}At_v \tag{12}$$

The term u_∞ should be adjusted in order to satisfy the continuity equation. Thus, for every cross section normal to x axis, the following equations apply (for simplicity, the contribution of u_{cj} is ignored because it is much smaller than the contribution of the flow driven by jet fans):

$$\dot{V}(x) = \iint u_{impulse}(x, r, \theta)\, dr\, d\theta \qquad (13)$$

$$u_{\infty}^{2}(x) = \left[\frac{\dot{V}(L) - \dot{V}(x)}{A} \right]^{2} \qquad (14)$$

The variable At_v represents a ceiling jet velocity decrement factor due to the reduction of buoyancy caused by the dilution of ceiling jet by flow driven by jet fans (that reduces the ceiling jet temperature, as seen before). It is related with the mixture between the ceiling jet flow rate (due to fire plume) and the flow rate driven by jet fans and this mixture is expressed by a dilution parameter D, which should be experimentally assessed:

$$At_v = \frac{\dfrac{0{,}195 Q^{1/3} H^{1/2}}{r^{5/6}}}{\dfrac{0{,}195 Q^{1/3} H^{1/2}}{r^{5/6}} + D \sqrt{u_{\infty}^{2} + \sum_{i=1}^{n} \alpha_i \left(\dfrac{k_{0\,i} u_{0\,i} 2 r_{0\,i}}{x - x_i} \right)^{2} e^{-\left(\frac{r_i}{k_i (x - x_i)} \right)^{2}}}} \qquad (15)$$