Int. J. of Thermal & Environmental Engineering
Volume 14, No. 2 (2017) 95-101

Smoke Clearance in an Underground Car Park using the Jet Fan System

Essam E. Khalil* and Sherif M. Gomaa

Mechanical Engineering Department, Cairo University, Giza, Cairo11321, Egypt

Abstract

Fire propagation and control in underground car parks are of an important safety issue. This paper investigates the effect of the jet fan system on the smoke clearance in an underground car park using CFD simulations. Two fire locations were considered under a steady state fire source of 4 MW. The consideration of the fire zone was also studied. The underground car park used in this study is 5,290 m^2 in area with a height of 3.7 m. A comparison between CFD results and analytical correlations for the fire modeling was made. The ANSYS FLUENT 14.0 software was used for all simulations. The results showed that the temperature is limited to the zone, where the fire is detected, and it is within an accepted range. The CO_2 mass fraction was presented and showed how the jet fans contribute in reducing the smoke density and hence improve the visibility. It was found that dividing the car park into zones is highly recommended and should be taken in the design of the jet fan system.

Keywords: *CFD, Fire Simulation, Dynamic Simulation, Car Parks*

1. Introduction

Fires in underground car parks are an important issue. Besides firefighting systems, a proper ventilation system is required to reduce the smoke density and temperature. The car park fires affect humans, finances and the environment. Smoke flowing rapidly through the enclosed area expresses the dangers of fires in underground car parks. Fire sources may be very difficult to fight because smoke may fill up the whole car park and obstruct vision. The main cause of death is the inhalation of smoke not by injuries from the fire itself.

The most common way to fight smoke, during a fire and once the fire has been extinguished, is by using the concept of smoke clearance. This way is not intended to maintain any area of a car park clear of smoke, to limit smoke density or temperature to within any specific limits, or to assist means of escape [1]. The alternative approach is to control the smoke movement in order to provide clear access for fire-fighters to combat the source of the fire [2]. It is imperative to mention that smoke control can provide smoke clearance [3] due to the fact that it removes and controls smoke from the car park while smoke clearance only removes smoke without controlling it.

Smoke clearance was based upon a duct work ventilation system to exhaust smoke during and after the course of a fire. In the duct work system two extract points are installed, one at the ceiling level and one at the bottom level. Each one extracts 50% of the smoke [1]. When a fire occurs, smoke naturally becomes buoyant and thus it rises to the ceiling of the car park. Therefore, 50% of the smoke is exhausted at the high level point while the air is mostly exhausted at the low level one. This indicates that almost 50% of smoke is not extracted.

A more successful alternative to the duct work system has recently appeared – the jet fans system. This system eliminates the need of duct works by hanging jet fans on the ceiling of the car park. The jet fans keep the smoke at the high level and transfer it to the exhaust fans or louvers. This means, almost 100% of the smoke is extracted. However, both systems still require the use of fresh air and extraction fans. The jet fan system consists of four main components; jet fans, extract and supply fans, a detection system, and a control system [4].

Limited literature thus far on the smoke clearance or smoke control in underground car parks has been published for public use. Henriques et al. [5] studied how the jet fans restrict the smoke, with a fire source of 4 MW, in the zone of the fire which provides good visibility for the firefighters. Viegas [6] studied the influence of parked cars with different car park heights on the smoke control using CFD simulations. Lu et al. [4] examined smoke control capacity of the impulse ventilation system in an underground car park. The jet fan numbers, the jet fan velocity, the extraction rate, and the system robustness on

* Corresponding author.
E-mail: khalile1@asme.org

DOI: 10.5383/ijtee.14.02.002

fire position were analyzed in this study. WANG et al. [7] studied the fire safety in an underground car park by providing a set of CFD simulations. Ali and Khalil [8] studied the performance of impulse ventilation system for smoke control in an underground car park and how the jet fan system improves the visibility in case of a fire. Tilley et al. [9] examined the effect of the smoke and heat control in an underground car park. The effect of the fire HRR, the smoke extraction flow rate, the openings for incoming air, and the presence of beams were studied.

In a limited-ventilation space, a special fire phenomenon occurs, which is called back draft [10]. It is an explosion caused by a fire, resulting from sudden re-insertion of oxygen to the combustion when it becomes oxygen-starved, which could result in a fire ball or blast wave by the opening of doors or windows [11]. However, the amount of fresh air must be taken into consideration in the design of the smoke clearance or control systems.

The jet fan operation time is an important parameter in the design of the smoke clearance system. If the jet fans turned on once the fire occurred, it might affect the means of escape by circulating the smoke. For this reason it is preferable to delay operation of the jet fans until after the automatic detection of fire. In order to avoid preventing the escape of occupants, it should be taken into consideration that the velocity of air within escape routes and ramps should not exceed 5 m/s [1].

It is important to define the Heat release rate (HRR) curve or value as a first step to design the system of smoke clearance. HRR of cars is the rate at which fire releases energy, measured in units of megawatts (MW). According to Li [12] the value of 8 MW seems to be appropriate for a worst passenger vehicle fire scenario. Okamoto et al. [13] carried out sedan passenger car fire experiments, and the peak HRR was close to 3 MW. Viegas [14] used 4 MW heat source to simulate a car fire. BS7346-7:2006 [1] suggests values for steady state car fires are as shown in Table 1. Time-dependent design fires should be based on an experimental test fire. The value of 4 MW HRR was considered in this paper.

Table 1: Steady-state design fires [1]

Fire Parameters	Indoor Car Park Without Sprinkler System	Indoor Car Park With Sprinkler System
Dimensions	5 M X 5 M	2 M X 5 M
Perimeter	20 M	14 M
Heat Release Rate	8 Mw	4 Mw

In the design of smoke clearance system, the car park should be divided into zones [15] of not more than 2000 m^2 [1], with a fully addressable fire detection system to limit the smoke dispersion in the car park in case of a fire.

The extraction rate from a car park is different from country to country and defined by a parameter called the Air Changes per Hour or ACH. The ACH is a measurement of ventilation; defined by how many times the air within the car park is replaced in an hour. The British standard BS7346-7:2006 [1] requires a minimum of 10 ACH for smoke clearance in case of a fire. Some standards give attention to the smoke volume produced from a car fire so that the extraction rate is equal or greater than this volume to make sure that the smoke is extracted.

2. Numerical Description

The governing differential equations for continuity, momentum, energy, species, and turbulence transport were solved using the CFD package ANSYS FLUENT 14.0 [16] and can be expressed in a general form as:

$$\frac{\partial}{\partial x_i}(\rho U_i \phi) = \frac{\partial}{\partial x_i}\left(\Gamma_\phi \frac{\partial \phi}{\partial x_i}\right) + S_\phi \tag{1}$$

where ρ is the air density, U_i is the velocity vector, ϕ is a dependent variable (scalar or vector), Γ_ϕ is the diffusion coefficient, S_ϕ and is the source term of ϕ. Table 1 shows the typical values of ϕ, Γ_ϕ, and S_ϕ. The FLUENT software utilizes the finite volume method [17] to solve these governing equations.

Table 2: Values of ϕ, Γ_ϕ, and S_ϕ

Equation	ϕ	Γ_ϕ	S_ϕ
Continuity	1	0	0
U-Momentum	U	μ	$-\frac{\partial P}{\partial x} + \rho g_x$
V-Momentum	V	μ	$-\frac{\partial P}{\partial y} + \rho g_y$
W-Momentum	W	μ	$-\frac{\partial P}{\partial z} + \rho g_z + \rho g \beta \Delta t$
Enthalpy	h	$\frac{\mu}{\sigma_h}$	S_h
Species	Y_i	$\frac{\mu}{\sigma_i}$	S_i
Turbulence Energy	K	$\frac{\mu}{\sigma_k}$	$G - \rho\varepsilon$
Turbulence Dissipation	ε	$\frac{\mu}{\sigma_\varepsilon}$	$\frac{\varepsilon}{k}(C_1 G - C_2 \rho \varepsilon)$

$\mu = \mu_l + \mu_t$, $\mu_t = \rho C_\mu \frac{K^2}{\varepsilon}$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $\sigma_h = 1$,
$C_1 = 1.44$, $C_2 = 1.92$, $C_\mu = 0.09$

$$G = \mu_t\left\{2\left[\left(\frac{\partial U}{\partial x}\right)^2 + \left(\frac{\partial V}{\partial y}\right)^2 + \left(\frac{\partial W}{\partial z}\right)^2\right] + \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)^2 + \left(\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x}\right)^2 + \left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y}\right)^2\right\} \tag{1}$$

In Equation (1), the scalar ϕ is considered for the local mean age of air or LMA. The LMA is defined as the average time for air to reach the point P once it enters the room. The newest air is at the fresh air inlets whereas the oldest is at the exhaust points and stagnant air areas.
Abanto et al. [18] defined that the diffusion coefficient Γ_ϕ for the LMA can be numerically calculated from the effective viscosity of the air μ as:

$$\Gamma_\phi = 2.88 \times 10^{-5}\rho + \frac{\mu}{0.7} \tag{2}$$

A user-defined function, UDF, was programmed [19] to solve Equations (1) and (2) using a user-defined scalar to be dynamically loaded with the ANSYS FLUENT solver.
The solver used was a steady, segregated pressure-based solver. Pressure and velocity were coupled with the SIMPLE algorithm to solve the equations of the discretized model. The second-order upwind discretization scheme was applied for the

momentum conservation equations and energy equation and first-order upwind for the turbulent kinetic energy, dissipation rate, and species equations. Standard K-ε turbulence model was used to account for turbulence. Standard wall function was used to account for near wall treatment modeling. The P-1 model was used to account for radiation. The gravitational acceleration was considered.

Comparison of CFD Results against Analytical

A comparison between the CFD results and analytical correlations made by Alpert [20] to make sure that the results are reasonable in dealing with smoke in case of a fire. Alpert has developed easy-to-use correlations for the maximum gas temperature at different positions in a ceiling jet flow produced by a steady fire. In SI units, Alpert's correlations for maximum ceiling temperature are as follow:

$$\mathbf{T - T_\infty = 16.9\,\frac{\dot{Q}^{\frac{2}{3}}}{H^{\frac{5}{3}}} \quad for\ \frac{r}{H} \le 0.18} \tag{3}$$

$$\mathbf{T - T_\infty = 5.38\,\frac{\dot{Q}^{\frac{2}{3}} \Big/ H^{\frac{5}{3}}}{\left({}^{r}\!/_{H}\right)^{\frac{2}{3}}} \quad for\ \frac{r}{H}} > 0.18 \tag{4}$$

where the temperature, T, is in °C; the total heat release rate, Q°, is in KW; and the radial position & ceiling height (r/H) are in m. Figure 1 shows r and H.

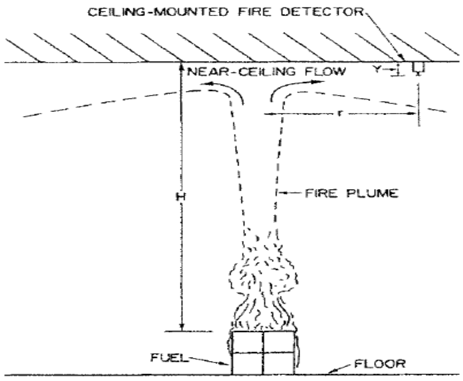


Figure 1: Ceiling jet flow beneath an unconfined ceiling [20]

In the validation process of the present work, the geometry of the domain was chosen to be a room with dimensions of 50 m x 50 m x 3.7 m. A fire was put in the center of the domain as a box of 5 m long and 2 m wide and 1.5 m high, from the floor, which is the fire dimensions for steady state calculations [1]. The domain boundaries were defined as being atmospheric pressures. The fire was defined as a surface that generates the mass rate of smoke at the smoke temperature. The mass flow rate of the smoke was estimated from Equation (5) [3] as:

$$\mathbf{m^o = C_e\, P\, Y^{1.5}} \tag{5}$$

where m^o is the mass flow of smoke rising past height Y, 2.5 m, C_e is a constant took a value 0.21 and P is the fire perimeter, 14 m. The temperature of the smoke was calculated using Equation (6) [3] as:

$$\mathbf{\theta = \frac{Q_s}{m^o C_p}} \tag{6}$$

where θ is the temperature of the smoke in °C above ambient, C_p is the specific heat of air at constant pressure, and Q_s is the heat carried by the smoke in kW, 4 MW.

As it is known, complete combustion yields carbon dioxide CO_2 and water H_2O. Combustion equation was considered for methane:

$CH_4 + 2O_2 \longrightarrow CO_2 + 2H_2O$

A simplification was made by defining the products of combustion and ignoring the combustion process during simulation; the mass fraction of CO_2 is 0.55 while the mass fraction of H_2O is 0.45.

The unstructured tetrahedral mesh was used. Small grid size was used to cover the fire domain with maximum element size of 0.15 m and 0.3 m to test the grid sensitivity, increasing by a growth factor of 1.2 to cover the entire domain, various grid sizes were used and the final results at particular location did not change by more than 1%. The mesh quality was respected with value above 0.58 for 0.15 m and 0.3 m grid sizes, where values close to 1 correspond to high quality. Table 3 gives the general characteristics adopted for the construction of the computational mesh for the two different mesh sizes. Figures 2 shows the mesh generation on x-z plane at y = 25 m. As shown, fine mesh was made around the fire domain.

Table 3: Computational mesh characteristics

Item	0.15 m	0.3 m
Minimum cell volume, m^3	7.21 x 10^{-5}	5.39 x 10^{-4}
Maximum cell volume, m^3	1.97 x 10^{-2}	2.04 x 10^{-2}
Total number of cells	1586055	1442563
Number of nodes	279693	274241

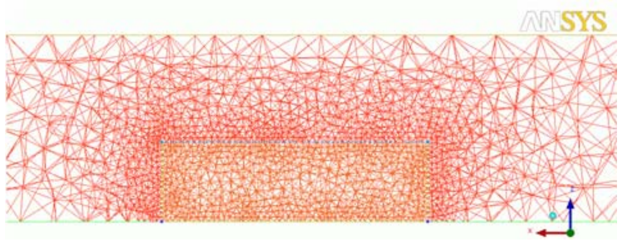


Figure 2: Volume mesh on x-z plane at y = 25 m

The temperature obtained numerically was compared to the temperature obtained analytically using Equations (4) at T_∞ = 20 °C (293.15 K), H = 2.2 m and at different distances from x-axis at r = 2 m from the center of fire, 4, 6, 8, 10, 12, 14, 16, 18 and 20 m, where the analytical data were made. Figure 3 presents this comparison as well as grid sensitivity with maximum element size of 0.15 m and 0.3 m on the fire domain increasing by the same growth factor of 1.2 to cover the entire domain.

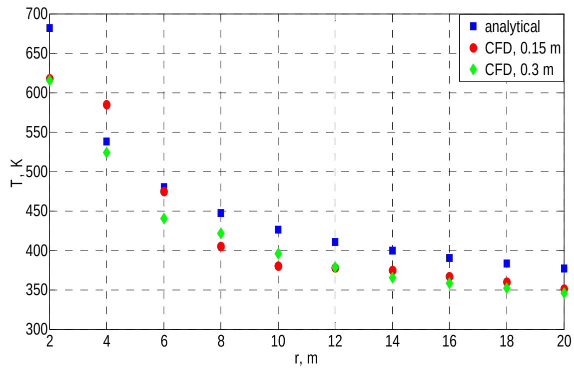


Figure 3: Ceiling temperature at different distances from fire center

It is apparent that there is a good agreement between the analytical and the predicted values. From Figure 3 it can be seen that there is almost no significant change in results when the mesh size on fire domain is 0.15 m and 0.3 m. Figure 4 shows the temperature contour at x = 25 m, the center of the fire along the y-axis.

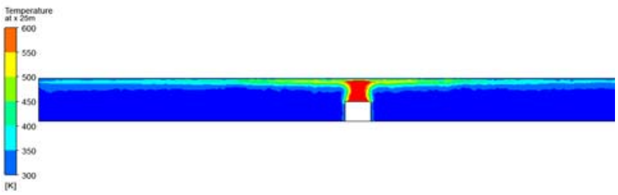


Figure 4: Temperature contour at y = 25 m

An Underground Car Park

The underground car park, used in this study, has 163 parking spaces on one single level with a gross area of 6,160.55 m^2. The net area of the car park is 5,290 m^2 which the computational work was carried out on. The floor to ceiling height of the car park is 3.7 m, the ceiling is quite flat. The car park has two exhaust points, EF1 and EF2, and one supply point via an entry/exit ramp, which remains permanently open to the exterior. The area in the middle is a permanently closed area used for machines; it was not considered in this study. Figure 5 shows the computational domain of the car park in 3D.

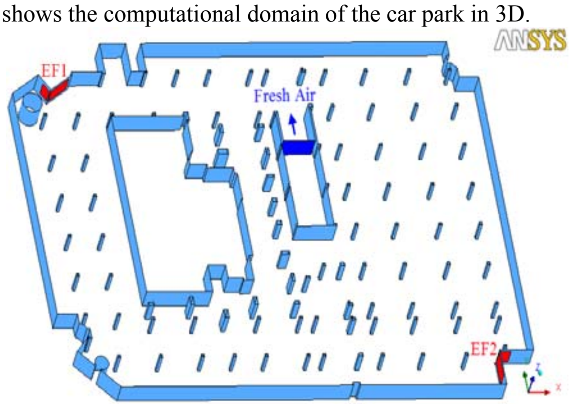


Figure 5: The computational domain of the car park in 3D

Exhaust fans connected to two louvers were installed at two points, one on the top left while the other is on the bottom right. Therefore, there is a louver at each point, louver 1 corresponding with EF1 and louver 2 corresponding with EF2. The area of louver 1, equal to area of louver 2, is 16.625 m2. EF1 conveys approximately 53% of the total air flow rate, while EF2 conveys approximately 47% of the total air flow rate. Inlet fresh air will be drawn into the car park via the ramp. It is assumed that air from the exterior can permanently circulate through the parking area with minimal resistance. The ramp width is 6.5 m. The jet fans configuration is as shown in Figure 6. 23 jet fans were installed at height 3 m from the floor.

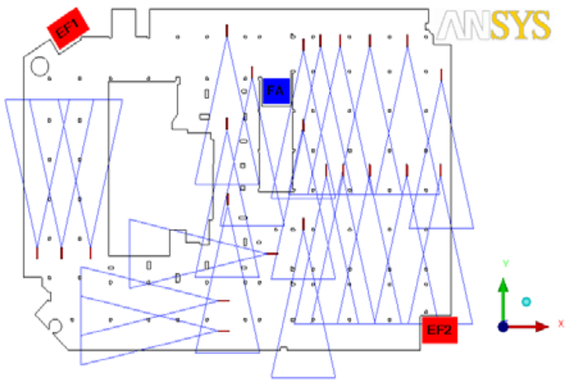


Figure 6: Configuration of the jet fans

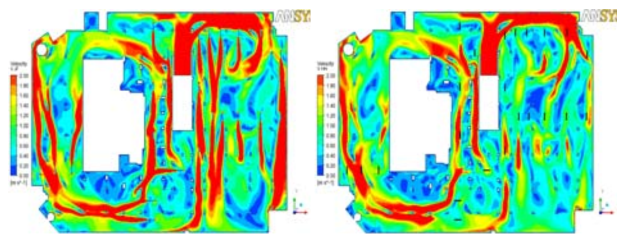
Steady state calculations were performed for all cases to achieve the 10 ACH [1]. The unstructured tetrahedral mesh was used. Small grid size was used to cover the jet fan domain with maximum element size of 0.08 increasing by a growth factor of 1.2 to cover the entire domain with maximum element size of 0.25 m at the inlet and outlet and 0.37 m on the walls, ceiling and floor. The 0.15 m size was chosen for the fire so that a gradual mesh is obtained all over the domain. Table 4 shows the simulation characteristics.

Table 4: Simulation characteristics

Jet Fan Number	Total number of cells	Flow Rate JF, m^3/s	Thrust N	Flow Rate EF1, m3/s	Flow Rate EF2, m3/s
23	3,739,942	2.3	54	34.6	30.7

3. Results and Discussions

Simulation results presented at two different heights; the height from the floor to the center of the jet fan (JF) at 3 m and the average adult human height (HH) at 1.7 m from the floor. The color-map range was set from 0 – 2 m/s for the velocity and set in the range of concern in all other results as in LMA with 10 ACH, for example, the value of 360 seconds was within the range. Figure 7 shows the velocity contours at 3 m and 1.7 m. Figure 8 shows the LMA contours at the same heights.



at 3 m at 1.7 m

Figure 7: Velocity contour at 3 m and 1.7 m

As shown in Figure 7, the velocity of air at 1.7 m did not exceed 5 m/s [1] in the entire car park in order to avoid preventing the escape of occupants.

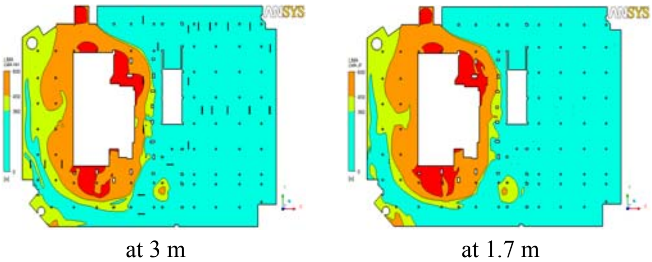


Figure 8: LMA contour at 3 m from the floor

From Figure 8, the performance of the ventilation system for the smoke clearance has been achieved in the right side of the car park and gave an acceptable range in the left side.

Consideration of Ventilation Zones

To avoid smoke spreading into the entire car park, the concept of fire zones were studied. Usually, this approach is used in large car parks when supply and exhaust points in each zone are available. For the smoke clearance mode, the system is operated according to the detected zone of fire within the car park.

Due to the geometry of the car park, it was split into two ventilation zones and has 23 jet fans installed at 3 m from the floor, controlled by the detection device. Every jet fan is assigned a number as shown in Figure 9; 13 jet fans in Zone A and 10 jet fans in Zone B. As the ramp is a common source of fresh air for both zones and the geometry is somewhat complex, the EF1 and EF2 are active regardless of which fire zone is detected. Also, EF1 and EF2 both together are just sufficient for smoke produced from a small car.

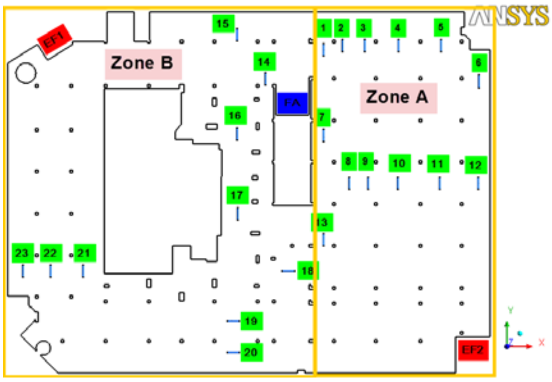


Figure 9: Car park ventilation zones

In case of a fire, Figure 10 shows the velocity contours at 3 m and 1.7 m for both zones A and B, respectively. The velocity of air at 1.7 m did not exceed 5 m/s as same as the case with no zones.

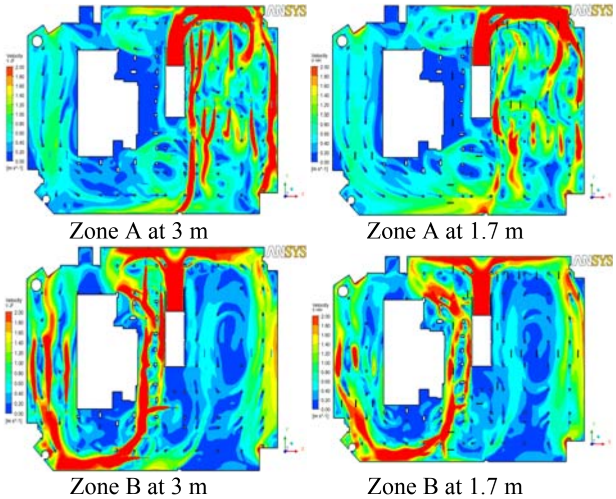


Figure 10: Velocity contours at 3 m and 1.7 m for zones A and B

Figure 11 shows the LMA contours at 3 m and 1.7 m for zones A and B, respectively.

It is apparent that the 10 ACH (360 seconds) was almost achieved in both zones which gives good insight about using such an approach, zoning the car park. However, sometimes it is required to activate some jet fans in the inactive zone to steer smoke masses toward the exhaust fans or to pressurize smoke by fresh air.

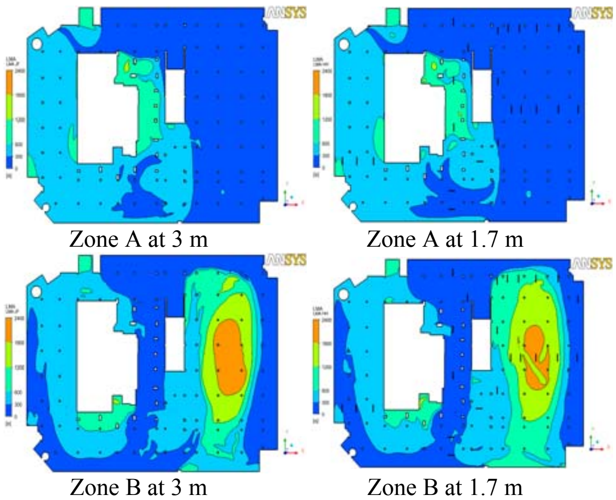


Figure 11: LMA contours at 3 m and 1.7 m for zones A and B

Consideration of Fire Location

Smoke clearance is not intended to maintain any area of the car park clear of smoke, to limit smoke density or temperature to within any specific limits, or to improve the visibility to assist means of escape, unlike the smoke control. However, it is good practice to examine the system to make sure that the ventilation system is effective in case of a fire and to fully understand it. In this section, two fire locations were considered, one in each zone as shown in Figure 12. The fire was defined using Equations (5) and (6).

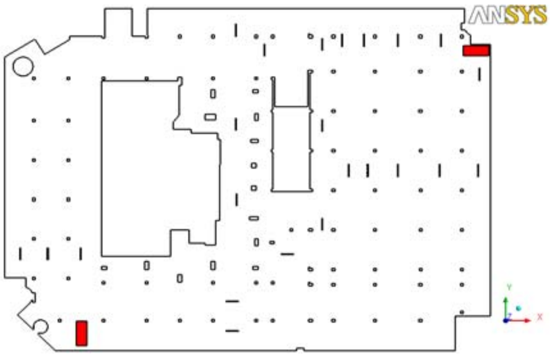


Figure 12: Fire location inside the car park

Figure 13 shows the velocity contours at 3 m and 1.7 m for zone A and B, respectively, when the jet fans is activated in each zone. Figures 14 shows temperature contours at 3 m and 1.7 m for zone A and B, respectively.

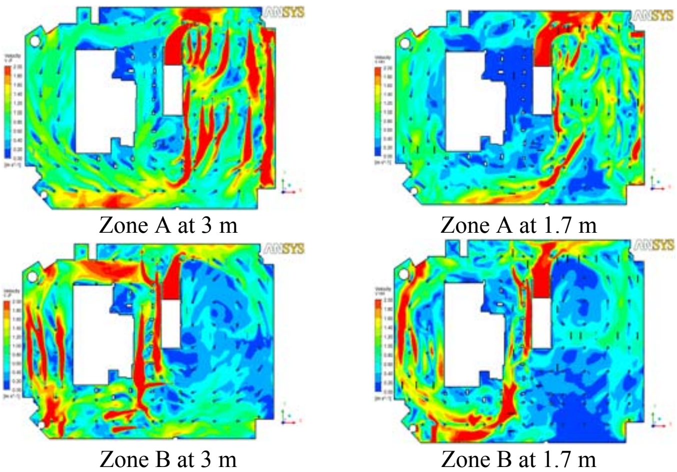


Figure 13: Velocity contours at 3 m and 1.7 m for zones A and B

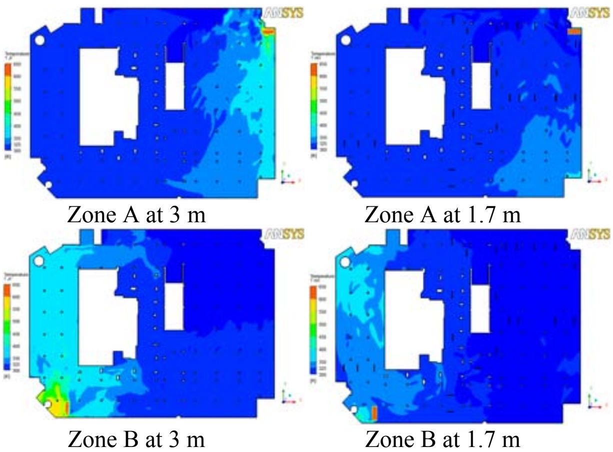


Figure 14: Temperature contours at 3 m and 1.7 m for zones A and B

It is clearly seen that the temperature is limited in the zone, where the fire is detected, and it is within the range of 350 – 450 K which is accepted. Figure 15, 16 and 17 show how the jet fan steer the smoke towards the exhaust point for zone A and B.

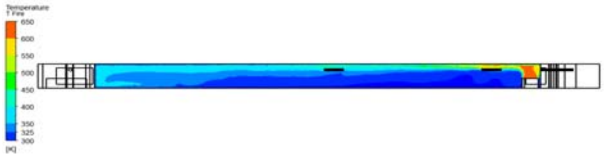


Figure 15: Temperature contour along y-axis, zone A

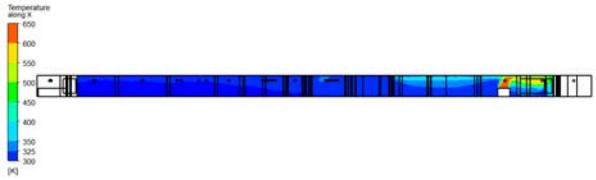


Figure 16: Temperature contour along x-axis, zone B

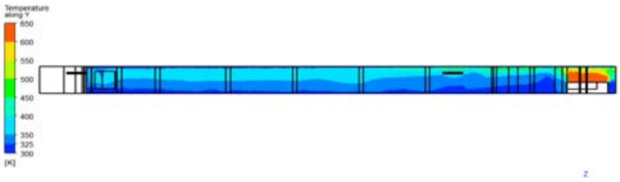


Figure 17: Temperature contour along y-axis, zone B

For further analysis of the flow field, it was important to look at the CO_2 pattern which is index for the visibility. Figures 18 shows the CO_2 mass fraction at the two concerned heights for zone A and B. It can be seen that the jet fans reduced the smoke density and restricted it where the fire is detected.

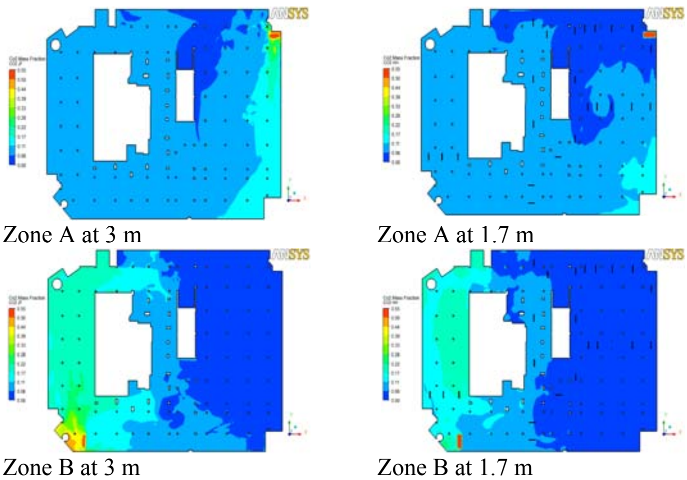


Figure 18: CO_2 mass fraction contours for zones A and B

4. Conclusions

This paper explained the concept of smoke clearance in an underground car park using jet fans through CFD simulations. The aim was to achieve 10 ACH in case of a fire. The concept of fire zones was studied to avoid smoke spreading into the entire car park. The car park was split into two zones. It was shown that dividing the car park into zones is highly recommended and should be taken into account in the design of the jet fan system. Two fire locations were considered to make

sure that the ventilation system is effective in case of a fire and to fully understand it. The results showed that the temperature is limited to the zone, where the fire is detected, and it is within an accepted range. The CO_2 mass fraction was presented and showed how the jet fans contribute in reducing the concentration of smoke. In all cases the velocity of air at 1.7 m did not exceed 5 m/s [1] in the entire car park in order to avoid preventing the escape of occupants.
From the results, it can be concluded that 10 ACH was not sufficient for zone B to clear all smoke produced from a small car fire. Therefore further works should be made to study the effect of ACH in the design of smoke clearance in underground car parks. Steady state calculations were considered in this study, a comparison between the steady and unsteady state in the design of such system should be investigated.

References

[1] British Standards Institute. "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." London, British Standards Institute, 2006.

[2] Colt. "Ventilation Systems for Car Parks and Service Areas. "Colt International Ltd, 2011.

[3] Morgan, H. P., Vanhove, B., and De Smedt, J. C. "Extending the Principles of Impulse Ventilation in Tunnels to Apply to Smoke Control in Car Parks." International Journal on Engineering Performance-Based Fire Codes 6, no. 2 (2004): 53-71.

[4] Lu, S., Wang, Y. H., Zhang, R. F., and Zhang, H. P. "Numerical Study on Impulse Ventilation for Smoke Control in an Underground Car Park." Procedia Engineering 11 (2011): 369-378.

[5] Henriques, B., Didier, E., and Brás, R. "Calibration of a Numerical Jet Fan Model for Simulating Smoke Control in Underground Car Park." In Fifth European Conference on Computational Fluid Dynamics (ECCOMAS CFD 2010), Lisbon, Portugal, June, 2010.

[6] Viegas, João Carlos. "Impulse Ventilation in Underground Car Parks: The Influence of Parked Cars in Smoke Control." In Fifth European Conference on Computational Fluid Dynamics (ECCOMAS CFD 2010), Lisbon, Portugal, June, 2010.

[7] Wang, Ying, Shi, Bi Ming and Mu, Chao Min. "Numerical Simulation on Fire Smoke Flow with Jet Fans in an Underground Garage." Advanced Materials Research 374 (2012): 534-537.

[8] Ali, Shadi, and Khalil, Essam E. "Numerical Investigation of Impulse Ventilation for Smoke Control in an Underground Car Park." Proceedings, IECEC paper IECEC-2012-1314859, 2012.

[9] Tilley, Nele, Deckers, Xavier, and Merci, Bart "CFD Study of Relation Between Ventilation Velocity and Smoke Backlayering Distance in Large Closed Car Parks." Fire Safety Journal 48 (2012): 11-20.

[10] Mao, J., Xi, Y. H., Bai, G., Fan, H. M., and Ji, H. Z. "A Model Experimental Study on Backdraught in Tunnel Fires." Fire Safety Journal 46, no. 4 (2011): 164-177.

[11] Wu, Jinxiang, Zhang, Yingting, Gou, Xiang, Yan, Meijuan, Wang, Enyu, and Liu, Liansheng. "Experimental Research on Gas Fire Backdraft Phenomenon."Procedia Environmental Sciences 11 (2011): 1542-1549.

[12] Li, Yuguang, and Spearpoint, Michael. "Analysis of Vehicle Fire Statistics in New Zealand Parking Buildings." Fire Technology 43, no. 2 (2007): 93-106.

[13] Okamoto, Katsuhiro, Watanabe, Norimichi, Hagimoto, Yasuaki, Chigira, Tadaomi, Masano, Ryoji, Miura, Hitoshi, and Ochiai, Satoshi, et al. "Burning Behavior of Sedan Passenger Cars." Fire Safety Journal 44, no. 3 (2009): 301-310.

[14] Viegas, João Carlos. "The Use of Impulse Ventilation for Smoke Control in Underground Car Parks." Tunnelling and Underground Space Technology 25, no. 1 (2010): 42-53.

[15] Aveiro, João L., and Viegas, João C. "Smoke Control in an Underground Car Park with Impulse Ventilation." In Fifth European Conference on Computational Fluid Dynamics (ECCOMAS CFD 2010), Lisbon, Portugal, June, 2010.

[16] Ansys Fluent. "14.0 Theory Guide." Ansys Inc., 2011.

[17] Versteeg, Henk Kaarle and Malalasekera, Weeratunge. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education, 2007.

[18] Abanto, Juan, Barrero, Daniel, Reggio, Marcelo, and Ozell, Benoît. "Airflow Modelling in a Computer Room." Building and Environment 39, no. 12 (2004): 1393-1402.

[19] Ansys Fluent. "14.0 UDF Manual." Ansys Inc., 2011.

[20] Alpert, Ronald L. "Calculation of Response Time of Ceiling-Mounted Fire Detectors." Fire Technology 8, no. 3 (1972): 181-195..