Journal of Thermal Engineering
Yildiz Technical University Press, Istanbul, Turkey
Vol. 1, No. 2, pp. 116-130, April, 2015.
http://eds.yildiz.edu.tr/journal-of-thermal-engineering/Articles
Manuscript Received August 28, 2014; Accepted September 05, 2014

This paper was recommended for publication in revised form by Regional Editor Derya Burcu Özkan

CFD ANALYSIS OF SMOKE AND TEMPERATURE CONTROL SYSTEM OF AN INDOOR PARKING LOT WITH JET FANS

A. Senveli
Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University (YTU), Yildiz, Besiktas, Istanbul 34349, Turkey

T. Dizman
Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University (YTU), Yildiz, Besiktas, Istanbul 34349, Turkey

A. Celen
Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University (YTU), Yildiz, Besiktas, Istanbul 34349, Turkey

D. Bilge
Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University (YTU), Yildiz, Besiktas, Istanbul 34349, Turkey

A. S. Dalkilic*
Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University (YTU), Yildiz, Besiktas, Istanbul 34349, Turkey

S. Wongwises
Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi (KMUTT), Bangmod, Bangkok 10140, Thailand

Keywords: Normal Pollution Ventilation (NPV), Computational Fluid Dynamics (CFD), Jet Fans, Impulse Ventilation, Smoke and Heat Control (SHC)
** Corresponding author: A.S. Dalkilic, Phone: +90 212 3832819, Fax: +90 212 2616659*
E-mail address: dalkilic@yildiz.edu.tr

ABSTRACT

The aim of this study is to perform analysis and interpretation of daily emission ventilation and fire ventilation system design in indoor parking lots with jet fans with the help of CFD program. In the study, ventilation of 8-story parking lot of a major business center in Istanbul with jet fans have been carried out, and the analysis on one story of this parking lot is provided as a case study. The airflow in all the regions of parking lot has been examined, the most suitable jet fan placements have been determined for daily emission and fire ventilation, and accuracy of this placement has been proved through the analysis with this study. The performed CFD analysis has enabled us to see correct placement of the jet fans and to intervene and decide the best placement. And for the fire case, the optimal results have been obtained by creating various scenarios in accordance with the international standards. It has been observed that the use of CFD analysis in the solution of the problem has allowed significant savings in time and money.

1. INTRODUCTION

In the general overview of the ventilation systems of enclosed spaces in a building, it can be seen that the conditioned fresh air is fed to the atmosphere through many vents or diffusers by transferring via a channeling system from the ceiling level or a relatively higher level. This fresh air flown to the environment through vents and diffusers triggers the air movement in the atmosphere by the induction effect. A homogeneous air distribution and ambient comfort conditions are provided by mixing the fresh air with the ambient air in the area. As a basic principle, the exhausted polluted air is relatively unimportant since the amount of fresh air and supply points are kept under control.

However, the designs and applications of parking lots are based on obtaining a certain amount of circulation of the parking lot volume per hour, through an exhaust based channel system, for the last 20 years. The provision of fresh air to the parking lots is considered relatively less important than the exhaust system. The exhaust air volume is kept under control indirectly for the amount of fresh air, leading to an indirect control of fresh air. In general, the fresh air is taken from ramps, openings or air shafts naturally. And in extreme cases, if there aren't sufficient natural openings in the parking lot, fresh air is supplied to the parking lot via limited number of fresh air fans.

Ciro Caliendo et al.[1] studied the numerical simulation of different HGV fire scenarios in curved bi-directional road tunnels and safety evaluation. In their papers, the effects of position of the HGV fire in the tunnel, tunnel geometry, longitudinal ventilation of jet-fans, and the presence of traffic flow on hot gas temperatures, air flow velocity, visibility distance, toxic gases concentrations and the people evacuation process are investigated deeply.

Nele Tilley et al.[2] executed the CFD study of relation between ventilation velocity and smoke backlayering distance in large closed car parks. They used a large set of (more than 350) CFD simulation as 'numerical experiments' from parameter variation in the simulations. Three formulae have been recommended on these topics as the critical inlet velocity,the difference between inlet and outlet velocity and the required ventilation velocity in the car park.

Yuan Jian-ping et al.[3] studied the numerical simulations on sprinkler system and impulse ventilation in an underground car park by using FDS code.Two cases with different combinations of ventilation and sprinkler system have been simulated to examine the effect of sprinkler system and impulse ventilation during car park fires. They found that the impulse ventilation system is very effective to control the smoke during an underground car park fire.

In B. Merci and M. Shipp's paper [4], they concentrated on car park fire safety, more especially on fire and smoke (and heat) dynamics. They investigated principally on the influence of horizontal mechanical ventilation, a popular technique, on the smoke and heat generated by the fire source. They emphasized that the air flow momentum must be strong enough to overcome the flow resistance caused by the fire-induced smoke flow and the air flow must be able to reach the fire source.

S. Lu et al. [5] investigated the smoke control capacity of impulse ventilation system (IVS) in an underground car park. They simulated 10 scenarios in a 80 m long, 40 m wide and 3.2 m height domain with a fire source simulating a car fire with an peak heat release rate of 4 MW by using Fire Dynamic Simulator version 5.30. Their results show that the smoke control capacity of impulse ventilation system is sensitive to jet fan numbers and increment in extract rate is conductive to relay jet flows. They emphasized that high jet fan velocity may cause severe smoke recirculation.

X.G. Zhang et al. [6] studied the fire spread and smoke movement in a large underground car park under various fire scenarios by using Fire Dynamic Simulator code. Their simulated results show that the development of car fire in the underground car park can be classified into four stages; (1) initial stage, (2) developed stage, (3) extinction and re-burning stage (4) fast-developed stage. They simulated the effect of ventilation on the fire spread and smoke movement in a large underground car park with 50 cars. They gave heat release rate, oxygen and soot concentrations as well as temperature distributions in their paper.

Joao Carlos Viegas [7] applied ventilation system for covered car parks and used Impulse ventilation systems (IVS) in order to control the smoke in the event of fire. He displayed an analytical model for the flow field near the ceiling and compared with CFD simulations.

X. Deckers et al. [8] benefitted from Computational Fluid Dynamics (CFD) simulation and performed full-scale car park fire experiments with smoke and heat control (SHC) by forced mechanical horizontal ventilation. They investigated the influence of the SHC system on the smoke movement in fire conditions. They found that improving the smoke extraction rate does not assist to take away the smoke if smoke is trapped inside a recirculation region.

Ran Gao et al. [9] focused on the suggestion of the spread of smoke in a huge transit terminal subway station in six different fire-source locations with heat-release rates (HRRs) of 4 and 7.5 MW. They examined the effects of the natural and mechanical ventilation in correlation with atrium height, roof window, rate of air change, and fire-source locations. They investigated the dispersion of fire-induced, buoyancy-driven smoke in a subway station by applying the Large Eddy Simulation (LES) simulation.

In case of a fire, the ventilation system has three different tasks, according to the intended design. These are, supporting firefighting teams in the evacuation of smoke in the parking lot during or after fire, creation of smoke-free zones to enable firefighting teams to step in the starting point of the fire, and to ensure the safety of escape routes from the parking lot. In general, natural ventilation, channeled mechanical ventilation and jet fan systems are widely used in parking lot ventilation. Although natural ventilation may be sufficient for open and semi-open parking lots, channeled mechanical or jet fan ventilation systems should be used in underground and closed parking lots.

In this study, the ventilation and fire analysis of an 8-story covered parking lot has been investigated with a CFD program (Autodesk Simulation CFD) by dividing the area into zones. For daily ventilation, optimum placements of the jet fans have been determined thanks to CFD analysis as not to leave any dead ventilation zones. Fire scenarios have been performed for different places in the area according to the standards, and comparisons have been made in terms of the smoke evacuation time under different working conditions of jet fans (full capacity/half capacity/off) for different zones. Change of speed, temperature, visibility and smoke emissions over time were given separately for analyzed zones according to different fire scenarios.

2. PRINCIPLES OF JET FAN VENTILATION SYSTEM

In parking lots, after natural ventilation and channeled mechanical ventilation the most commonly used third method is known as jet fan ventilation and it's based on change of speed phenomenon. The jet fan ventilation method is optimized through continuous testing and integrated into the parking lot security systems.

All air is drawn through fans and ducts in conventional ventilation systems. This is performed for both the provided fresh air and the discharged, used air. The airflow rate is kept as low as possible to prevent pressure drop. However this means that the channels should be relatively wide, and therefore requires a larger area. But a different approach is used in jet ventilation system. In this system, a small amount of air is sucked into a fan and then it is discarded at a higher speed. When the exhausted air hits the air in front of the fan, it pushes that air forward and pulls the surrounding air at the same time. In this way, all of the surrounding air is mobilized and moved by 20-40 meters without using channels. Whole parking lot acts as an air duct. The principle behind the jet ventilation system is the same as that used in rockets where a small amount of air (combustion products) are thrust at higher speeds to push the rocket upwards. Space-saving, flexible installation, uniform air distribution, effective ventilation in the parking lot, energy and cost savings are among the advantages of the fan ventilation system.

The performance of jet fans is expressed by using thrust. The thrust, that is the force created by jet fans, is expressed as Newton [N] and it's the product of change in speed and mass flow rate. In practical application, it is usually suggested that the distance between the closest beams should be 0.5 meters at the entrance and 2 meters at the exit for the highest yield, in jet fan ventilation systems. Height of the beams should not be more than 0.4 meters. Otherwise, the beam height must be compensated either by hanging down the fans from the ceiling or by increasing fan's distance between the nearest beam. All energy is transported in the form of speed from the discharged air to the environment, since the fan is fastened securely. Fan remains in place when air is delivered forward. And as a result of the drag, the amount of moving air will always be greater than the amount of air passing through the fan. The amount of moving air is the same in different sections of the facility. Different speeds can be achieved in particular sections depending on the size of the system. Not only the size and number of jet fans depend on the size of the parking lot and its plan but also the purpose of the system to be used for the CO evacuation or smoke control.

In case of a fire, jet fans don't start immediately in accordance with the required smoke evacuation speed, and will remain off for a certain period of time, and then the fresh air and exhaust fans run at maximum power. This allows people in the parking lot to escape. Jet fans start to work after evacuation of the building or arrival of firefighting, and push the air to exhausters. This has two benefits. First, the smoke accumulates within a relatively restricted space, and this allows detecting fire's location and extinguishing. Second, it ensures that the temperature is low near the fire, so firefighters can come closer to the fire.

The smoke management refers to all measures that can be taken one by one or in combination in order to reduce the movement of smoke for the benefit of firefighting team and goods. Certain measures taken with the help of fans during the smoke evacuation: Control of smoke movements is provided by making zoning, smoke extraction ducts, gates, chimneys or using fire or smoke dampers and performing positive pressurization, in the jet fan systems

3. NUMERICAL SAMPLE APPLICATION ON INDOOR PARKING LOT OF A BUSINESS CENTER

Autodesk Simulation CFD [10] program is used to apply heat transfer and fluid flow analysis generally. Solution of the flow and heat transfer problems encountered in industry can take months via prototype, physical test, and trial-and-error methods or with traditional (academic) CFD software. The solution time of the flow and heat transfer problems can be reduced from months to days, with Autodesk Simulation CFD [10].

For this, the model or the environment to be analyzed should be modeled with one of the computer-aided design programs first. The created model is transferred and run in the CFD program, and all of the rest of the work is performed and completed with the CFD program. After opening the model in the CFD program, a material should be assigned to each fluid and volume, and there shouldn't be any volume left unassigned. After this process, the boundary conditions are determined, for example, all the necessary calculation values are entered such as air intake and exhaust flow rate values at the point where jet fans blowing air into the parking lot, zero pressure locations, temperature inputs and outputs, and so on.

The last step and most important step before the solution is the process called meshing. We can define mesh as the following: It's the process of dividing a volume or surface area into very small sections, to examine the movement of particles. Meshing has a critical importance. If the number of meshes isn't set well, we cannot obtain the desired results. For example, if too much mesh number is given, the computer will have difficulty in the solution and the solution time becomes longer unnecessarily. Or if we give too low mesh number, the result of the analysis may not be accurate as desired. Therefore, the fine-tune of the number of mesh is an important stage of the analysis process. Details of the mesh used in the analysis are shown in Figure 1.

The last step is the solution stage. Values to be entered and wanted results are selected and the number of iterations is determined. Determining the correct number of iterations is also critical in the analysis. Iteration is the process to use previous

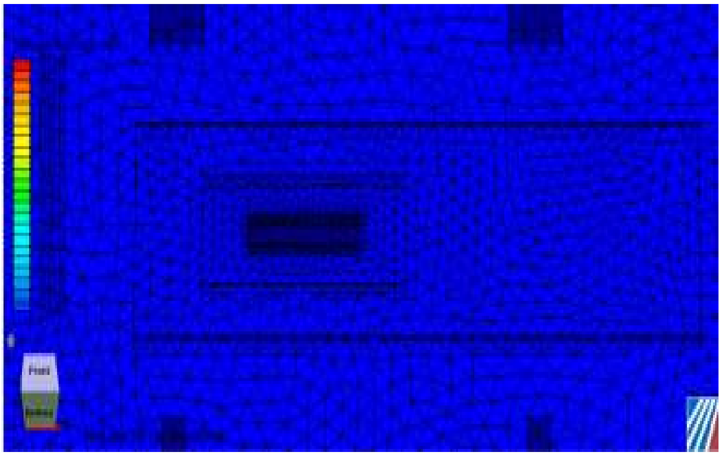


Figure 1 - Proper Mesh Transitions between different parts

output in the calculation as the input in the next calculation in order to successively converge to the result. Simply, accuracy of the result directly proportional to the number of iterations, but too much iteration will increase the analysis time unnecessarily.

The parking lot of the analyzed business center consists of 8 floors, including PB1, PB2A, PB2, PB3, PB4, PB5, PB6 and PB7. This parking lot has been divided into specific zones and the CFD simulation has been performed as if these zones are separate parking lots. The model of the analyzed parking lot is shown in Figure 2.

The smoke is spread other floors and regions from the fire location, then it can cause damage and death in these places. In recent years, zone (region) control methods have been developed to prevent movement of the smoke. The purpose here is to divide the building into pressure zones as shown in Figure 3, and control the movement of smoke. Smoke is sucked and exhausted from the fire area with the help of mechanical ventilation system; fresh air is pumped into the other areas. Zones should also be isolated well from air leaks and airflow, in order to obtain the desired result in the system. In this system, the exhaust of the smoke from the fire area also prevents the excessive pressure increases in the fire area caused by thermal expansion seen before.

Figures 4 and 5 show an example layout of a jet fan in the examined parking lot. The discharge unit consists of a screen, an aspirator and, if necessary, a noise-block in general. When the set limit is exceeded, the aspirator starts first, and then the jet fans start to work. In outdoor parking lots, where no ventilation system is required, it's possible to support natural ventilation with jet fans. This prevents the formation of "dead" areas. This is also valid for other parking lots, which cannot meet the outdoor parking lot requirements with a very small margin. The relevant conditions can be met by making use of jet fans only in this type of facilities. In such cases, the best solution is the use of 100% reversible axial fans. These fans can provide thrust in both directions. Thus, the direction of flow can be changed according to wind conditions.

As shown in Figure 6; the exhaust fans are working with a single exhaust shaft from PB2A to PB7, and this transfer discharges the exhaust gases. These fans are shown in red on the CFD model. As shown in Figure 7, the fresh air fans are working with 3 different fresh air shafts from PB2A to PB7, and these fans are shown in blue on the CFD model.

Fresh air will be sent to all floors with mechanical ventilation. However, the fresh air can pass naturally through gates where the parking lot floors are connected to other floors and entrance-exit ramps.

4. RESULTS AND DISCUSSION

The ventilation of parking lots are usually designed for the removal of smoke in case of fire, and limiting CO and other emissions of vehicles. The present ventilation system is used for both cases usually. The purpose of the ventilation is to discharge harmful gases as soon as possible or to rarefy their ratio in the environment.

CFD analysis has been performed for Daily Emission Ventilation (NPV - Negative Pressure Ventilation) and Emergency Mode (EM) for all of the floors. 5-air-exchanges has been used for the Daily Emission Ventilation and 10-air-exchanges has been used for the Emergency Mode, according to ASHRAE[11]'sheating, ventilation and air-conditioning standards. All jet fans are set to perform daily emission ventilation (NPV) and they start in all zones at the same time to perform 5 air exchanges per hour. The blowing distances of jet fans are chosen as to create airflow in the environment and to transfer the air coming from fresh air fans to the exhaust areas. A conversion factor is applied for the local pollution caused by movements of cars in their own area. The adjustment for air dampers for these local air exchange requirements can be performed periodically. This local exchange is an entirely optional decision based on analysis results.

The volumetric flow rates, NPV flow rates, EM volumetric flow rates, EM flow rates in the exhaust and fresh air shafts computed by considering the above mentioned air exchange coefficients for all floors according to usage areas of each floor is shown in Table 1 and Table 2. The floor height is accepted as 3 m on average in the calculation of these values.

The area has been divided into zones first, and the daily emission ventilation analysis has been performed for steady state and the fire analysis has been performed for transient cases, by designing different scenarios in the zones. The most appropriate results for analyzing the airflow are air velocities and LMA (Local Mean Age). In the LMA analysis no recirculation is desired in any of the zones. The desired parameter for LMA in Daily Emission Ventilation (NPV) analysis is 950 seconds. Again for the LMA, this parameter is 360 seconds (approximately 10 air changes) in the Emergency Mode (EM). The air speeds denote air distribution and air movement throughout the parking lot. In the analysis of airflow rate, areas without airflow are not desired in Daily Emission ventilation (NPV) and Emergency Mode (EM). Desired airflow

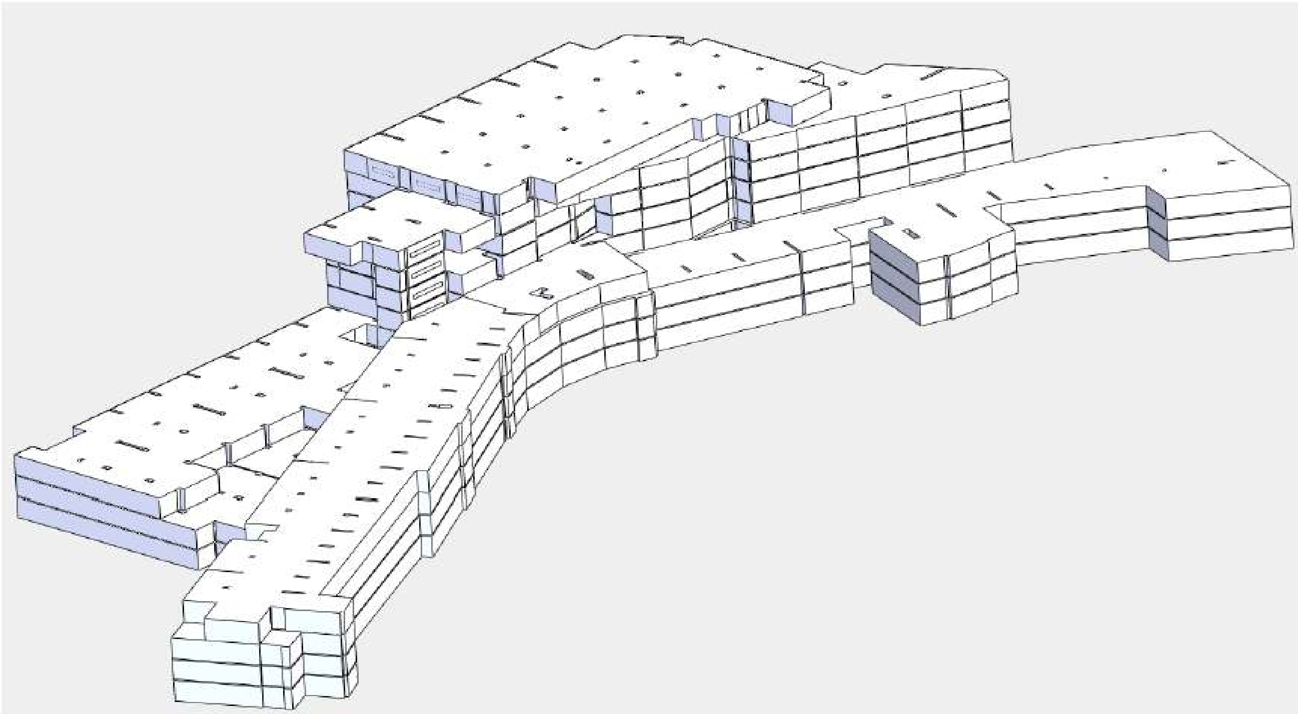


Figure 2 –The model of car-park

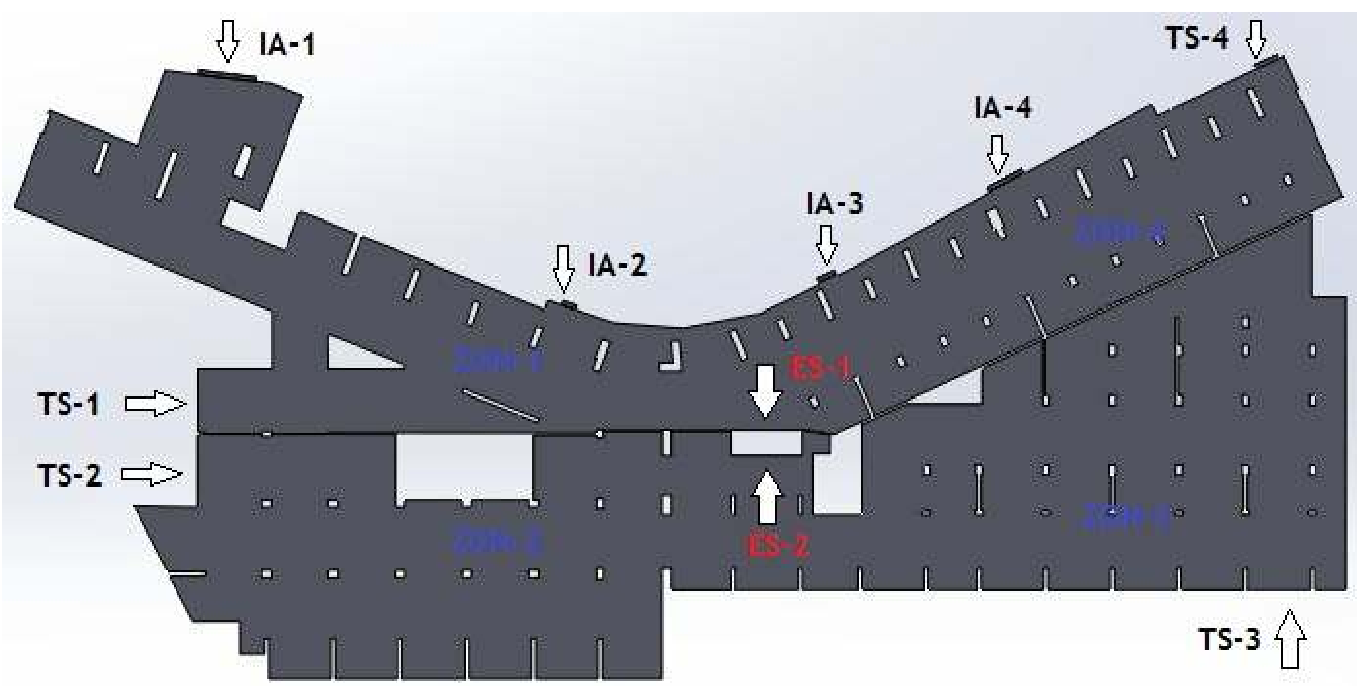


Figure 3 – Smoke control zones for PB7 floor

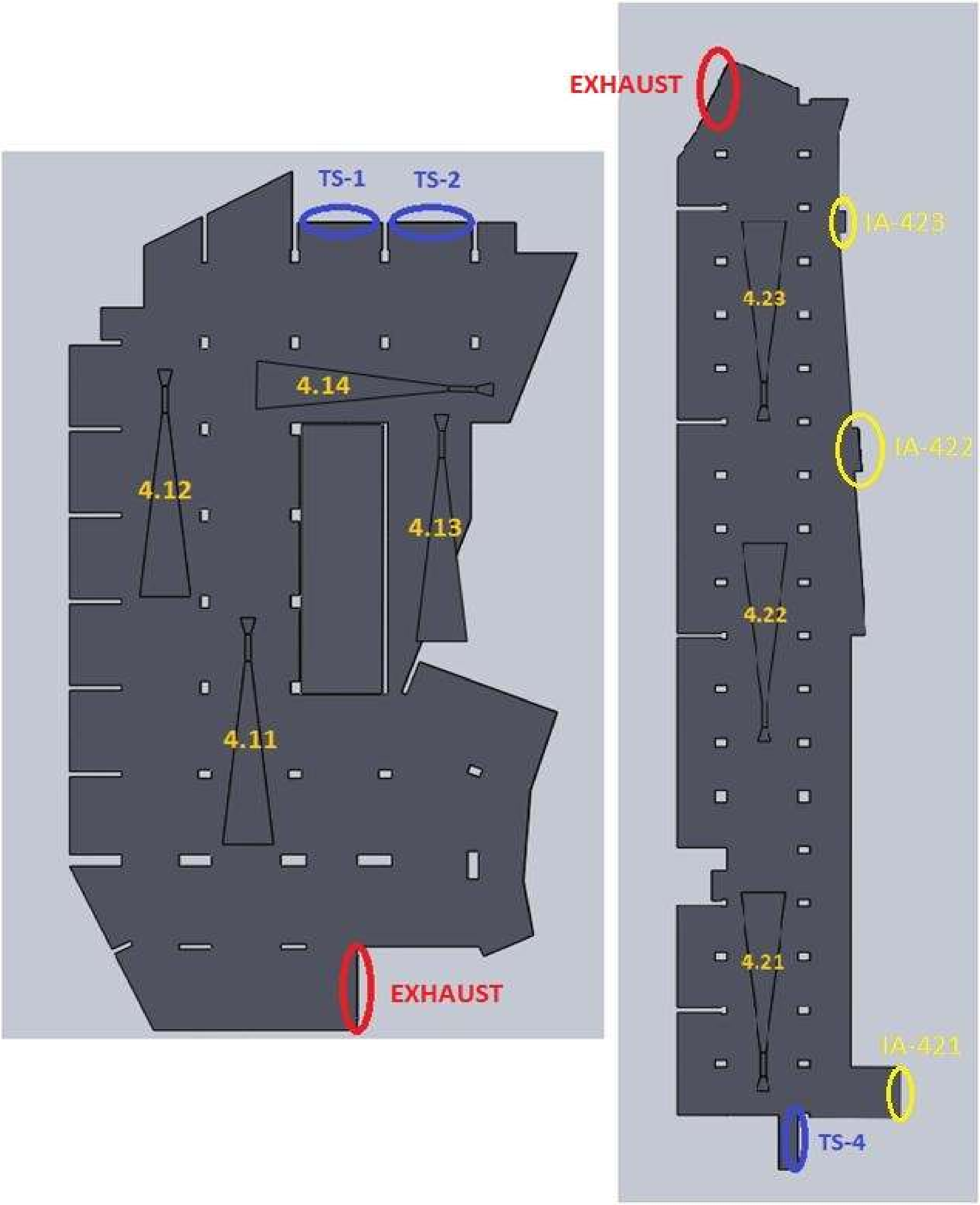


Figure 4 -Jet fan layout for PB4 Zone-1 and 2

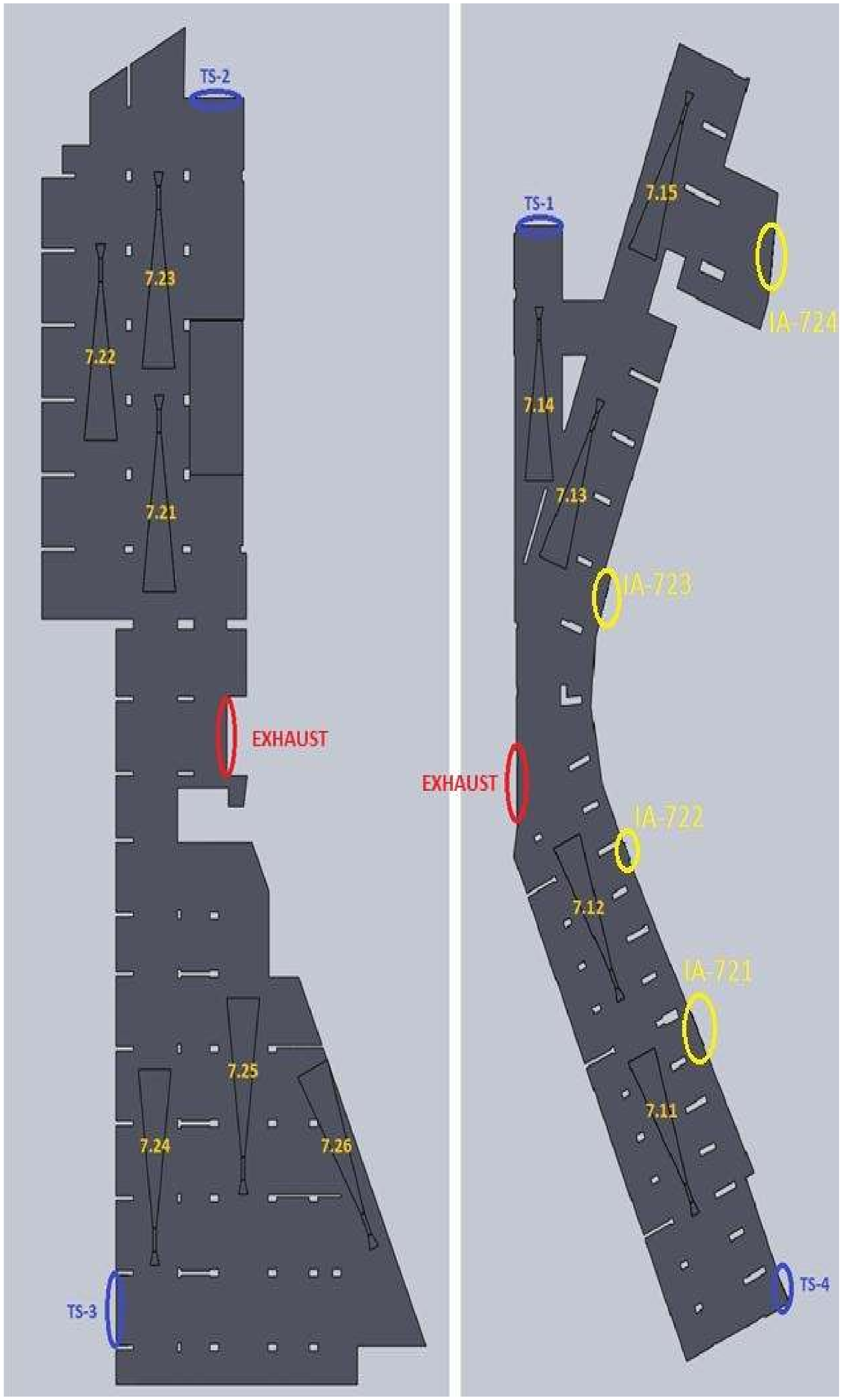


Figure 5 – The sample jet fan layout of PB7 floor

rate is 0.1 m/s or at higher speeds, and results suitable to this have been obtained.

As can be seen in Figure 8, no areas observed with stagnant airflow in parking lot and walking paths in the daily emission ventilation. One of the most important results to be examined in terms of NPV analysis, the LMA (Local Mean Age), that is the average period of time for air particles to stay in the environment. This is because; the purpose in NPV ventilation is to discharge CO particles at certain time intervals. As can be seen in Figure 9, there were no stationary air particles was observed, staying more than the optimum period, when looking at the LMA result of the analyzed floor. These are acceptable results, since the LMA values are in the desired range.

A fire scenario with a fire sprinkler system having 4MW of power, 14m length and 2x5 m dimension has been designed, in accordance with the standards in time-dependent fire analysis. The fire simulation has been performed for PB7 floor Zone 3. 3 pieces of jet fans is seen in Figure 10, and the size of fire exit is 5x2 m.

It's revealed by our analysis performed for fire cases in parking lot regions that analysis is required, where we have stopped the transfer of smoke to the other zones and confined and discharged the smoke in a single zone. It's very important for firefighting team to reach the fire area, in order to extinguish the fire and perform the evacuation of people without any hustle in those areas, by preventing smoke to spread to other zones during fire ignition, its growth, flares, fully developed fire, and extinguishing, in the fire zone. The type of solution to prevent spreading of smoke to other zones is revealed in the analysis results.

In order to prevent the spread of fire from the floor in fire to other floors, the shafts are closed and isolated from this floor, to block spread of smoke in exhaust shafts and fresh air shafts. And in the floor, working capacities of jet fans, fresh air and exhaust fans becomes two times the daily emission ventilation capacity, and all the power is used for the evacuation of smoke, during the fire. In addition to this, operation of jet fans in the adjacent zone prevents spread of smoke. In the time-dependent analysis we've performed at this point we can obtain the ability to make comments on whether the jet fans and fresh air fans at the other zones should operate or not. The analysis results we've obtained from various scenarios show us that we have obtained the desired results, and the jet fans in the other zones operate at half capacity and prevent the spread of smoke, by creating airflow and an air block in the exhaust area used by other zones in common. The results we've obtained in other scenarios have presented that smoke is spread to adjacent zone

Table 1 –The details of mechanical exhaust ventilation shafts

Floors	NPV flow rate (m^3/h)	Area (m^2)	NPV flow velocity (m/s)	EM flow rate (m^3/h)	EM flow velocity (m/s)
PB2A	44.928	4,20	2,97	212.000	14,02
PB2	42.912	4,20	2,84	212.000	14,02
PB3	43.584	4,20	2,88	212.000	14,02
PB4	50.285	4,20	3,33	212.000	14,02
	36.269	4,20	2,40	212.000	14,02
PB5	63.475	4,20	4,20	212.000	14,02
	65.971	4,20	4,36	212.000	14,02
PB6	72.307	4,20	4,78	212.000	14,02
	53.626	4,20	3,55	212.000	14,02
PB7	42.232	4,20	2,79	212.000	14,02
	32.532	4,20	2,15	212.000	14,02

Table 2 - The properties of fresh air shafts

Floors	Usage Area (m^2)	NPV flow rate (m^3/h)	NPV flow velocity (m/s)	EM flow rate (m3/h)	EM flow velocity (m/s)
PB2A	4,20	13.478	0,89	100.000	6,61
	4,20	13.478	0,89	100.000	6,61
PB2	4,20	12.874	0,85	100.000	6,61
	4,20	12.874	0,85	100.000	6,61
PB3	4,20	13.075	0,86	100.000	6,61
	4,20	13.075	0,86	100.000	6,61
PB4	4,20	15.085	1,00	100.000	6,61
	4,20	15.085	1,00	100.000	6,61
	2,80	21.761	2,16	67.000	6,65
PB5	4,20	18.156	1,20	100.000	6,61
	4,20	21.427	1,42	67.000	4,43
	4,20	21.496	1,42	100.000	6,61
	2,80	16.589	1,65	58.000	5,75
PB6	4,20	17.971	1,19	100.000	6,61
	4,20	14.204	0,94	67.000	4,43
	4,20	21.335	1,41	100.000	6,61
	2,10	22.049	2,92	58.000	7,67
PB7	4,20	18.236	1,21	100.000	6,61
	4,20	14.296	0,95	67.000	4,43
	4,20	20.229	1,34	100.000	6,61
	2,10	22.003	2,91	58.000	7,67

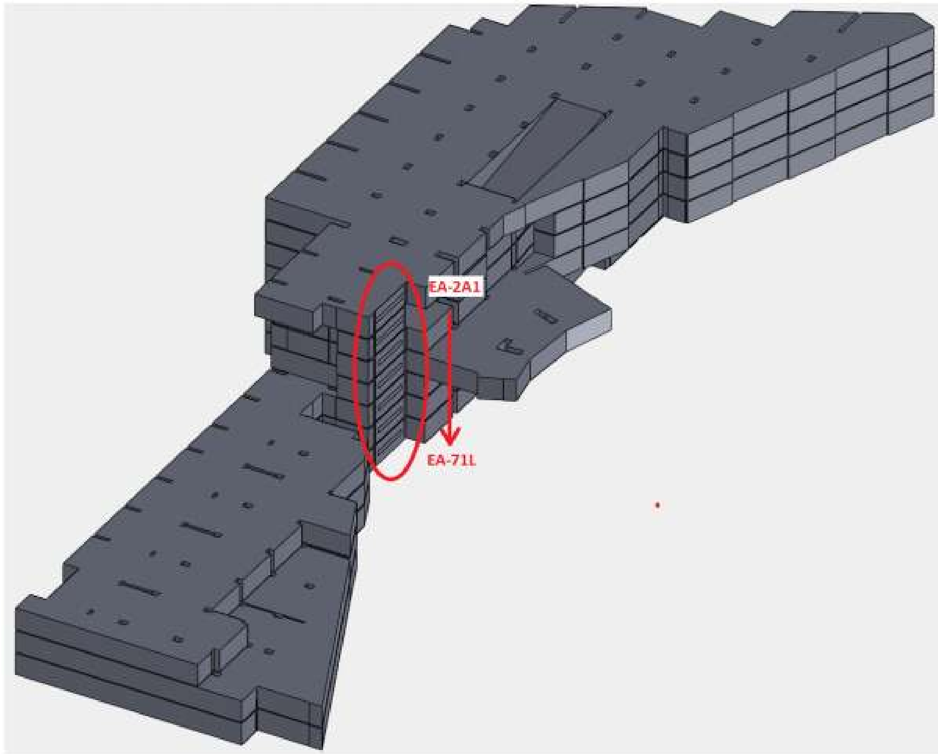


Figure 6 - Exhaust shaft

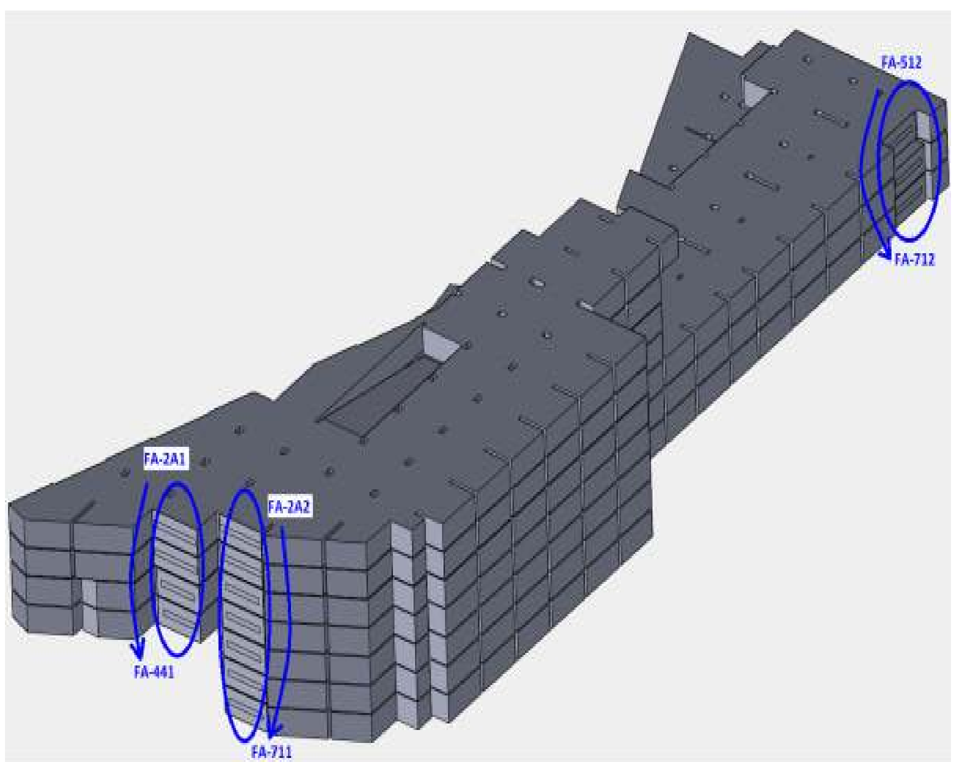


Figure 7–The zones of fresh air shaft

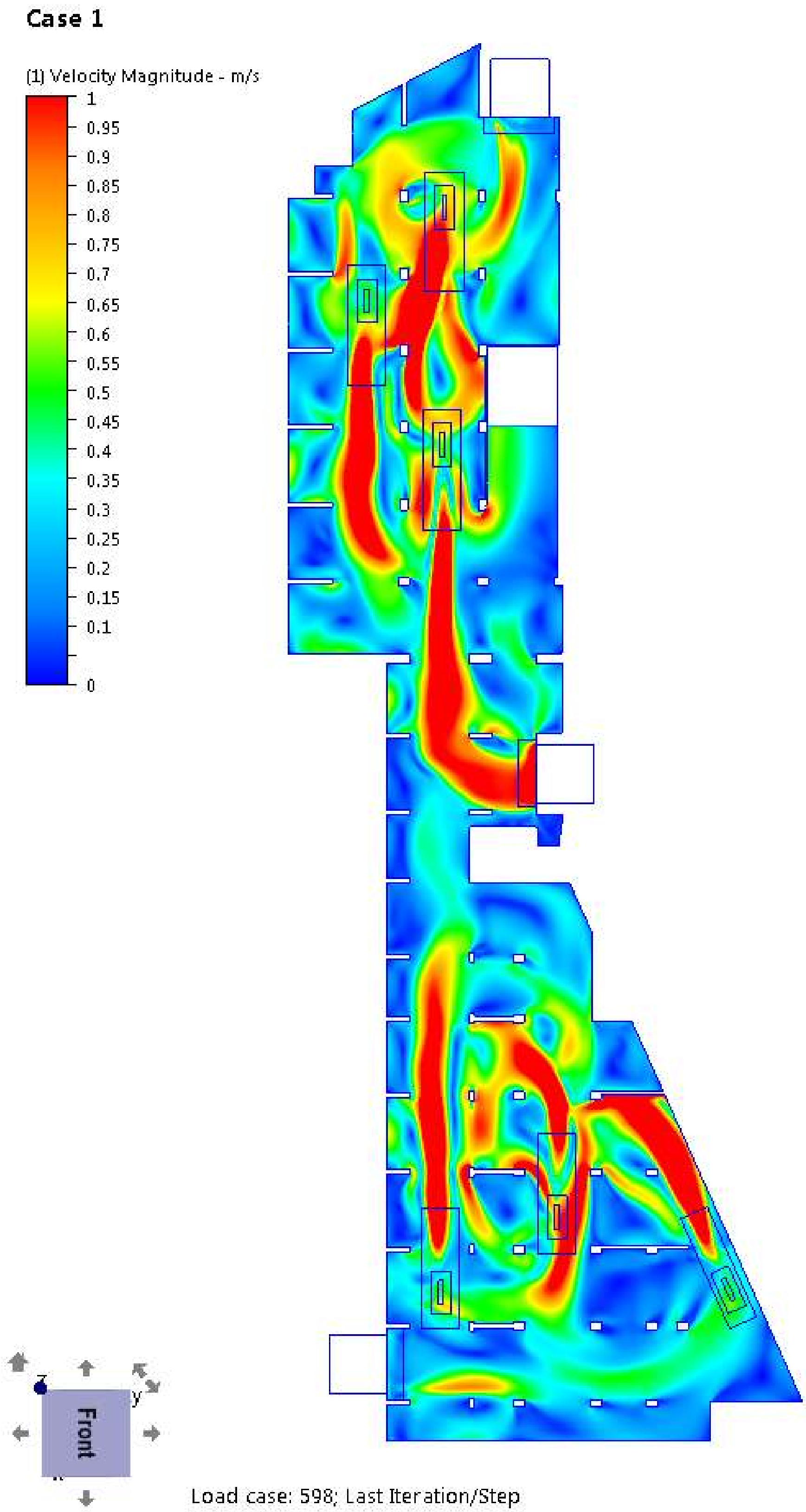


Figure 8–The velocity analysis at 1.7 m for PB7 Zone-2 and 3

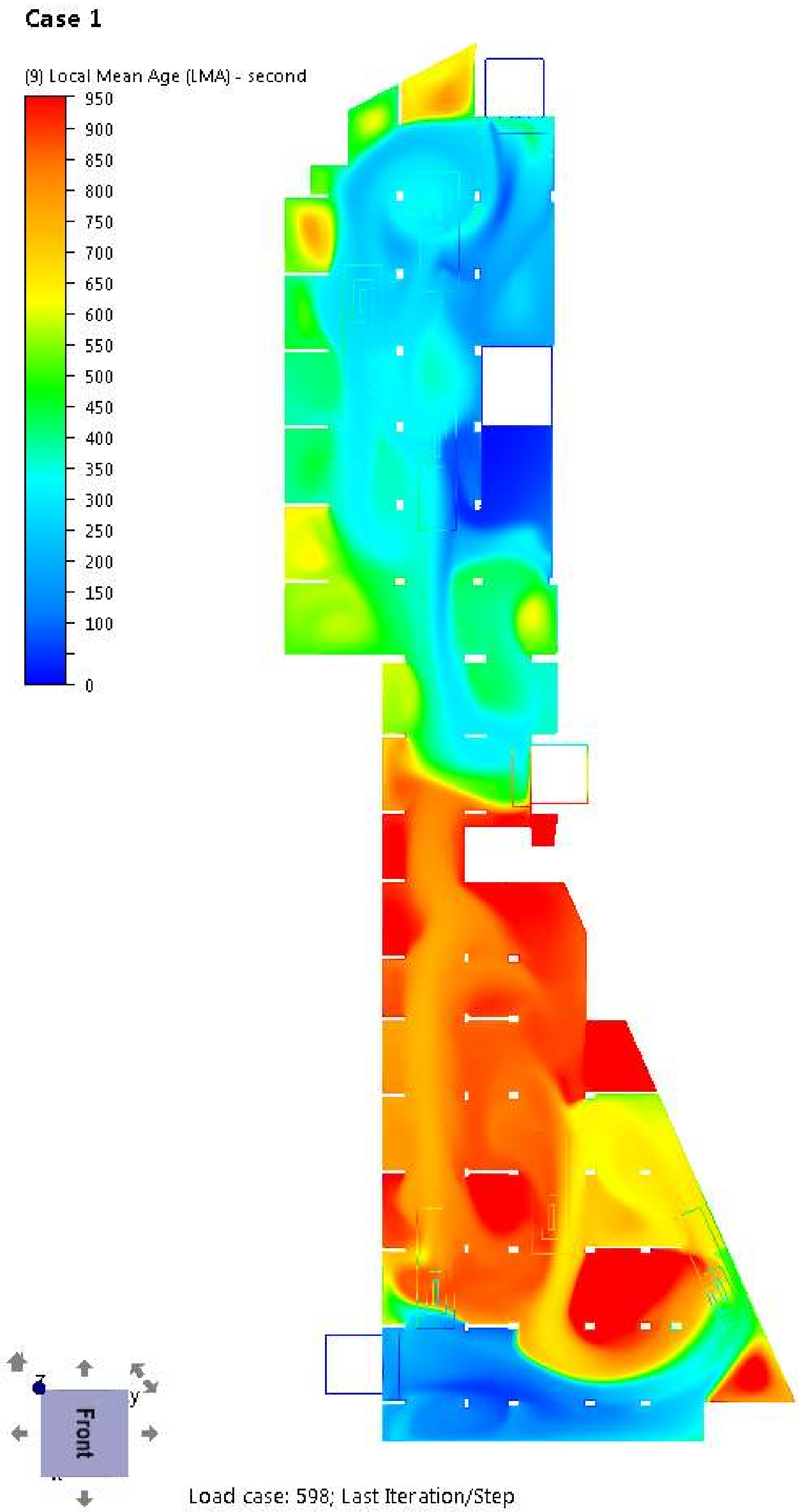


Figure 9 – Local Mean Age (LMA) analysis at 1.7 m for PB7 Zone-2 and 3

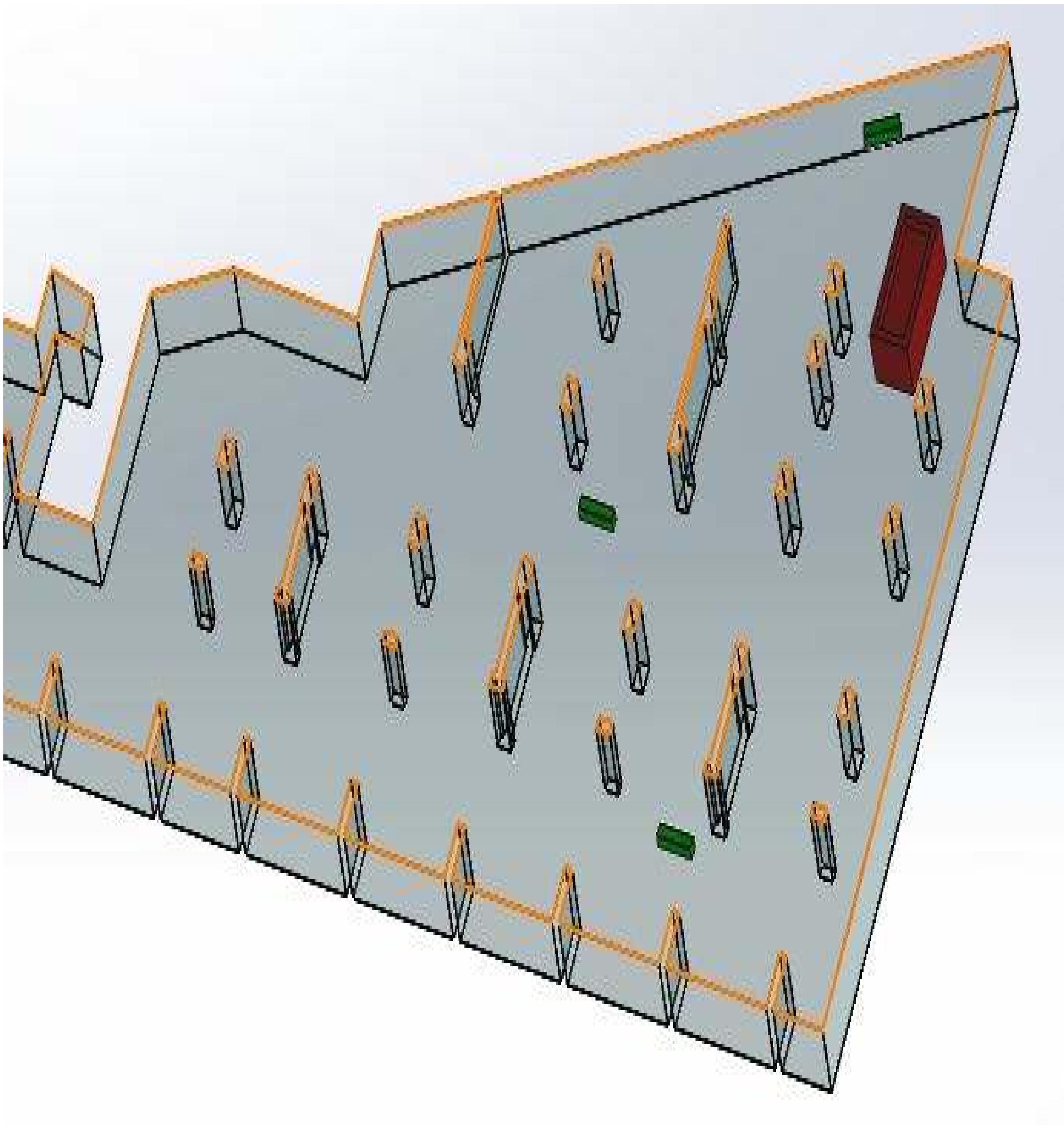


Figure 10 –The position of fire source

in this case (for example, in cases of nonoperational jet fans in the other zone or the operation of fresh air fan only, not the jet fans). Therefore, jet fans and fresh air fans at the neighboring zones should be operated for the discharge of smoke quickly, before spreading to the neighboring zones.

4.1 Main Scenario - Evaluations

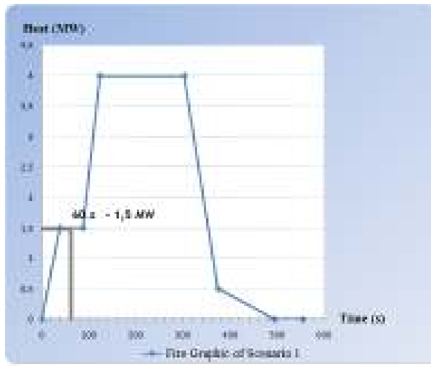
In this scenario, it's assumed that all the fans are working at full capacity, since the fire has been started in Zone 1. Fresh air fans and jet fans are not working in the adjacent zone Zone 2, and the analysis has been continued by assumed that the fire will continue for 3 minutes at 4MW of power, and will stop gradually.

In this scenario, the power has been reached to 4MW in 123 seconds, and continued for 3 minutes at the same power, in the fire started at Zone 1. Later, the fire has been stopped gradually, and the analysis results have been obtained.

Development of the fire, temperature, velocity and smoke distributions is given in the Figure 12 for 60 second after the start of fire, and in Figure 12 for after 550 seconds.

As a result of this analysis it's seen that smoke and heat energy of a 3 minutes of fire started in Zone 3 will escape to Zone 2, at 4MW.

It should be noted that detailed analyses on this subject can be seen from authors' publication [12].



60 s – 1,5MW

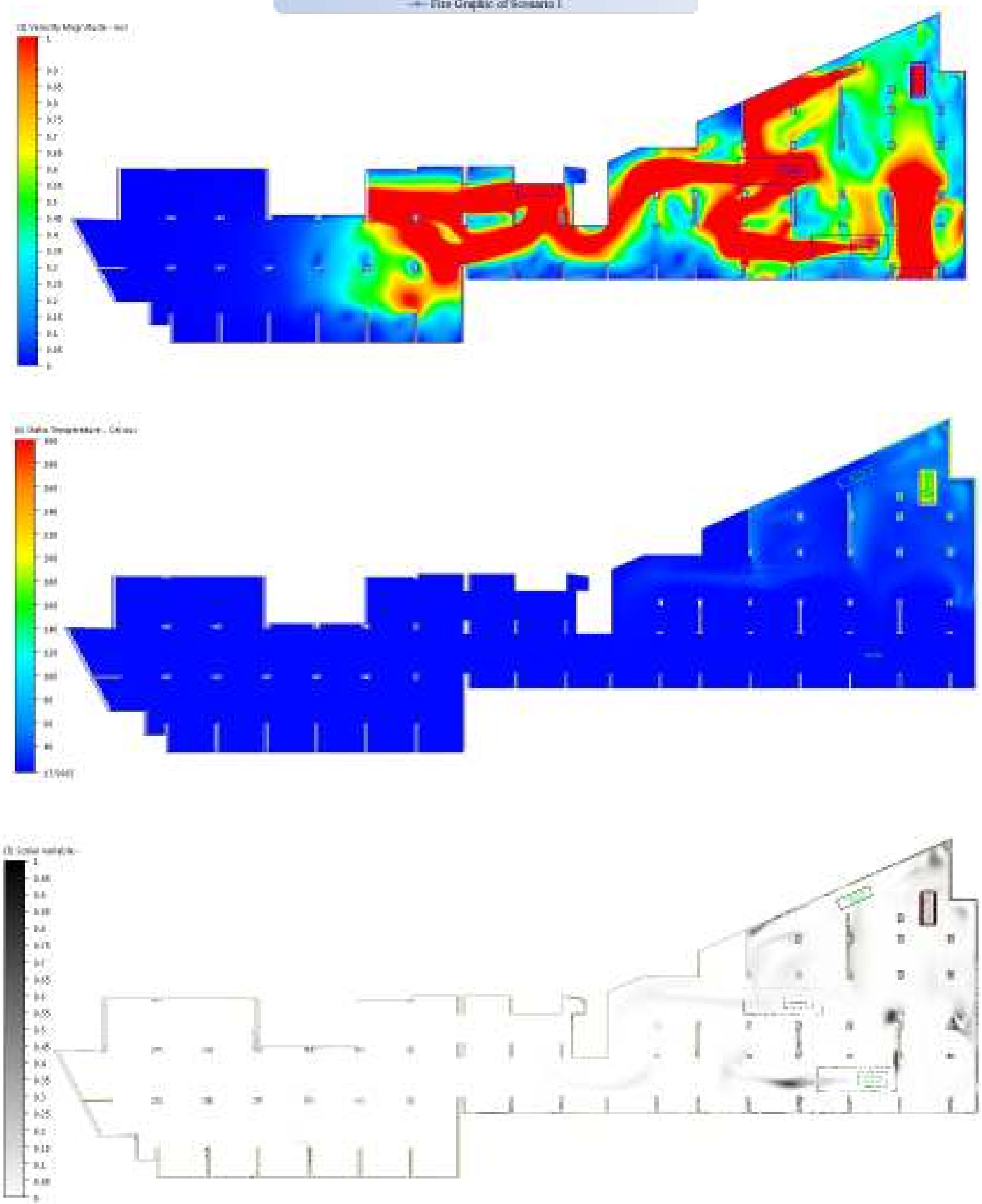
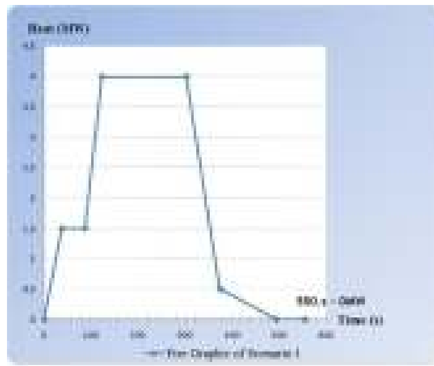


Figure 11–Analysis results at 1.75 m (Velocity, Temperature distributions, Smoke spread) – 60s



550 s – 0 MW

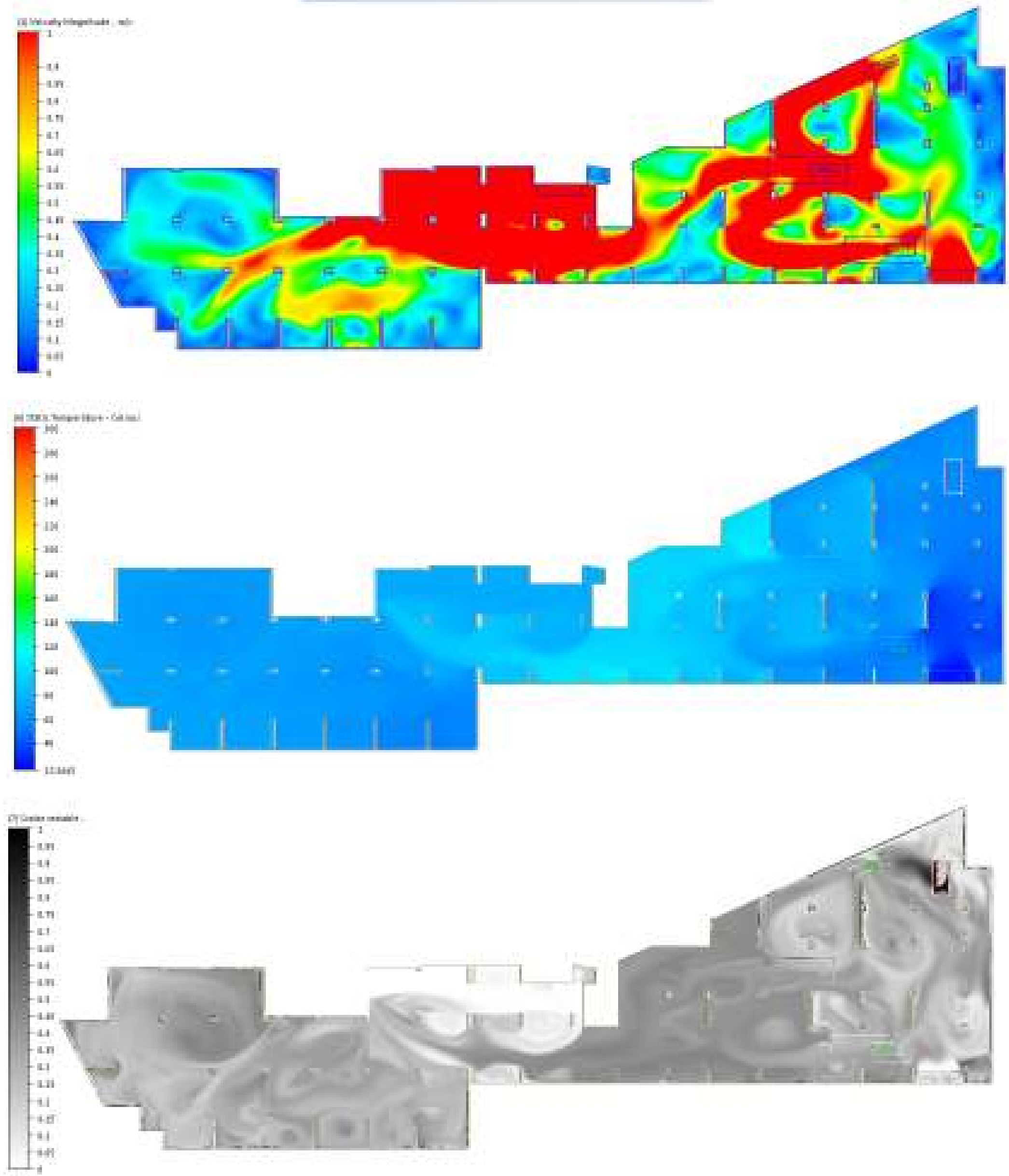


Figure 12 –Analysis results at 1.75 m (Velocity, Temperature distributions, Smoke spread)–550s

5. CONCLUSION

The CFD analysis performed in the study has a great importance in terms of daily emission ventilation, fire and smoke control ventilation, in indoor parking lots. It enables to create safe areas by evacuating smoke as soon as possible and control of hazards in advance. In addition, temperature, air flow rate, smoke density and visibility can be checked separately in any section of the parking lot for any time interval, and it's also seen that this gives us results in comply with the standards.

According to the analysis results, it's observed that the airflow has been created effectively by the placement of jet fans at optimum locations correctly. The dead zones have been eliminated, where there were no air exchanges, in the daily emission ventilation including the indented areas in the parking lot, with the formation of airflow. It is obvious that there will be economically large returns when the desired results have been obtained in the analysis by using models of jet fan placements in tight spaces at different air flow rates.

NOMENCLATURE

LMA Local Mean Age

IVS Impulse Ventilation System

SHEVS Smoke and Heat Exhaust Ventilation System

SHC Smoke and Heat Control

HRR Heat Release Rate

LES Large Eddy Simulation

FDS Fire Dynamics Simulator (Software)

NPV Normal Pollution Ventilation

EM Emergency Mode

ACKNOWLEDGMENTS

The authors are grateful to the company of Afs Boru Sanayi A.Ş. for their contributions to this study.

REFERENCES

1. Caliendo C., Ciambelli P., Guglielmo M., Meo M. and Russo P., "Numerical simulation of different HGV fire scenarios in curved bi-directional road tunnels and safety evaluation'', Tunnelling and Underground Space Technology, Vol. 31, pp. 33–50, 2012.
2. Tilley N., Deckers X.. and Merci B., ''CFD study of relation between ventilation velocity and smoke backlayering distance in large closed car parks,'' Fire Safety Journal, Vol. 48, pp. 11–20, (2012).
3. Jian-ping Y., Zheng F., Zhi T. and Jia-yun S., ''Numerical Simulations on Sprinkler System and Impulse Ventilation in an Underground Car Park'', Procedia Engineering, Vol. 11, pp. 634–639, 2011.
4. Merci B. and Shipp M., ''Smoke and heat control for fires in large car parks: Lessons learnt from research'', Fire Safety Journal, "Article in Press".
5. 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, Vol. 11, pp. 369–378, 2011.
6. Zhang X.G., Guo Y.C., Chan C.K. and Lin W.Y., ''Numerical simulations on fire spread and smoke movement in an underground car park'', Building and Environment, Vol. 42, pp. 3466–3475, 2007.
7. Viegas J., ''The use of impulse ventilation for smoke control in underground car parks'', Tunnelling and Underground Space Technology, Vol. 25, pp. 42–53, 2010.
8. Deckers X., Haga S., Tilley N. and Merci B., ''Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations'', Fire Safety Journal, "Article in Press".
9. Gao R., Li A., Hao X., Lei W. and Deng B., ''Prediction of the spread of smoke in a huge transit terminal subway station under six different fire scenarios'', Tunnelling and Underground Space Technology, Vol. 31, pp. 128–138, 2012.
10. Autodesk Simulation CFD, Autodesk, Inc., 111 McInnis Parkway San Rafael, CA 94903 Available from: http://usa.autodesk.com/
11. ASHRAE Handbook: HVAC Application 1995. Atlanta, Ga.,U.S.A.: American Society of Heating, Refrigeration and Airconditioning Engineers.
12. Dalkilic A.S., Kundu B., Celen A., Atayilmaz S.O., Kayaci N. and Wongwises S., ''Smoke control of a car park by means of CFD analyses using jet fans, ASME 2014 Fluids Engineering Summer Meeting August 3-7, USA, 2014.