Journal of Engineering Research and Reports

13(3): 27-40, 2020; Article no.JERR.57470
ISSN: 2582-2926

CFD Analysis of Smoke and Temperature Control System of Car Park Area with Jet Fans

M. T. Çakir[1*] and Ç. Ün[2]

[1]Sivas University of Science and Technology, Mechanical Engineering, 58000, Sivas, Turkey.
[2]Department of Mechanical Engineering, Çukurova University, 01000, Adana, Turkey.

Authors' contributions

This work was carried out in collaboration between both authors. Author MTC designed the study, performed the statistical analysis, wrote the protocol and wrote the first draft of the manuscript. Authors MTC and CUN managed the analyses of the study and managed the literature searches. Both authors read and approved the final manuscript.

Article Information

DOI: 10.9734/JERR/2020/v13i317102
Editor(s):
(1) Dr. Anuj Kr. Goel, Maharishi Markandeshwar University, Haryana, India.
Reviewers:
(1) Ryohji Ohba, The University of Tokyo, Tokyo.
(2) Zhongjian Pan, China.
Complete Peer review History: http://www.sdiarticle4.com/review-history/57470

Original Research Article

Received 25 March 2020
Accepted 03 June 2020
Published 13 June 2020

ABSTRACT

Basements of hospitals are generally used as indoor parking area. When underground car parks are examined physically, it is not possible to ventilate them by natural roads. Therefore, in order to keep the density of harmful gases below certain limits; ventilation systems must be used. In this study, for a parking lot with a closed area of 19438.59 m²; jet fan system is designed and the performance of the system is simulated with Computational Fluid Dynamics (CFD). The covered parking area examined is located in Isparta; It is located on the 2nd basement of Isparta City Hospital. The parking area is divided into 5 zones and CFD smoke analysis is made for the jet fan ventilation system designed for the parking area. The purpose of this study in Isparta City Hospital (basement-2) to examine the ventilation system is designed for parking areas using CFD. The CFD analysis made allowed us to see the correct placement of the jet fans and to decide the best placement. In case of a fire, various scenarios were prepared in accordance with international standards and the best results were obtained.

Keywords: Isparta; city hospital; computational fluid dynamics; axial jet fan.

**Corresponding author: Email: mutlutarik.cakir@yahoo.com;*

1. INTRODUCTION

In the study by Immonen [1], two important design difficulties were investigated in a parking lot via CFD regarding the optimal placement of jet fans. In the first of these difficulties, jet fans, which do not affect the air quality, have been identified in a jet fan layout that is available in a closed area. These jet fans have been identified by creating a simulation based on classical fractional multi-factor design. In the design, reducing the required number of CFD's to a manageable level has become even easier.

In the study by Viegas [2], analytical and CFD simulation studies on the use of jet fan system for ventilation, fume control in indoor car parks were conducted. In studies carried out, the flow created by jet fans and jet fans was examined through CFD simulations. Analyzes, the location and size of the fire source, it has been made considering the restriction of the distance between the jet flow and exhaust fans. In this analysis, the difference between jet fan ventilation and smoke control, the effect of fire heat release, the effect of the location of the fire source, the distance between the jet fans and the effect of the limitations in the exhaust flow were investigated. In this study, it is informed that jet fans can only be used to add momentum to the flow in tunnels, while it can prevent smoke distribution for car parks.

In the study by Lu et al. [3], the capacity of the jet fan smoke control systems have been studied. For the analysis, a car park with dimensions of 80 x 40 x 3.2 m has been used and 10 different fire scenarios have been prepared according to 4 MW magnitude of fire. In the system; jet fans, fresh air fan and exhaust fan are used. Fresh air and exhaust fans are east-west direction. Jet fans have been chosen bidirectionally. In the analysis, the number of jet fans, jet fan speed and exhaust ratio were examined and 4 different scenarios were prepared for the analysis of the jet fan numbers. In these scenarios, 0, 8, 10 and 11 jet fans were activated respectively, and all remaining conditions remained the same. Thanks to these scenarios; It is concluded that the number of jet fans directly affects the viewing distance and clarity.

In the study by Deckers et al. [4], mechanical horizontal ventilation and smoke and heat control were examined with a full-scale parking simulation. The fire heat radiation rate is defined as the input parameter and its effect on smoke and fire smoke movement in terms of the temperature control system was examined. Fire heat release rate, smoke exhaust flow, openings with air inlet and the presence of beams on the car park ceiling were also examined as parameters, thus different flow models were created. In the study, the effect of Jet fans is dealt with a series of simulations.

In the study by Li et al. [5], the effect of the amount of fresh air on the smoke exhaust efficiency was investigated. In the study, a fire simulation that lasted for 400 seconds in a 50 m x 25 m x 24 m (length x width x height) dimensions at the center of the atrium with a magnitude of 5 MW was performed. The sensors was located symmetrically on both sides of the fire zone and measurements was taken at vertical intervals of 0.5 meters. For the room with 30000 m^3 volume, 30%, 50% and 70% of the exhaust smoke amount based on 4 ACH air changes; Fresh air was given at flow rates of 36000, 60000 and 84000 m^3 / h, respectively, and the effect of different rates of fresh air on smoke exhaust efficiency was examined. According to the results, it was observed that the efficiency of the exhaust smoke was higher compared to 50% and 70% when using fresh air as much as 30% of the exhaust smoke.

In the study by Spiljar et al. [6], it was shown that the ventilation system installed in the car parks not only depends on the design rules but also on the car park architecture. In the study, CFD analysis was made only on daily ventilation scenario for jet fan system in a closed parking lot with a total area of 2651.8 m^2. Ramps are considered a parking space and the remaining space will be ignored. In the analysis, impact on the air flow compartment walls of a car parking architecture were examined and it is concluded that it affects the air flow to a large extent.

In the study by Dalkılıç et al. [7], the fire ventilation system provided with a jet fan for an indoor parking lot located in Istanbul was analyzed with the CFD program in accordance with international standards. As a result of the study, CFD analysis of daily ventilation and fire / smoke control has been reported that in terms of great importance. In the study; Temperature, air flow, and the smoke density according to standard results obtained are indicated in terms of visibility. At the same time, it has been observed that jet fans are properly placed in optimum locations and the non-use areas in the parking lot are minimized by directing the air flow.

In the study by Arinia et al. [8], 1: 8 scale basement fire and smoke movement were experimentally observed; at the same time, 7-minute fire and 12 ml gas fire were modeled in the experiment. The results obtained were compared with the 1200-second real-scale fire simulation using the FDS program. It was reported that the reason for the incompatible match of the results was the use of a parking garage in complex geometry.

2. PRINCIPLES OF JET FAN VENTILATION SYSTEM

2.1 Jet Fan System and its Basic Components

There are basic components that must be used in order to operate the jet fan system correctly and effectively. These key components are listed below.

- ✓ Jet fans
- ✓ Axial fans
- ✓ Dampers
- ✓ System control board

Jet fans are designed for ventilation of underground and semi-open type car parks and tunnels and smoke evacuation in case of fire. These fans are produced in two different types, axial and radial propellers. Axial jet fans are the jet fans in tunnel ceilings adapted to indoor parking lots. These jet fans consist of axial fan, engine with fire resistance, silencer, deflector and wire cage at the suction-discharge nozzles. In Fig. 1, axial jet fan is shown.

In order to work during the fire; the engines of these jet fans were classified according to European norms. Temperature resistance classes of fans according to EN 12101-3 standard are given in Table 1.

Axial jet fans get their names from the axial throw. With the axial impeller structure, air is sucked parallel to the shaft rotating around an axis and the air is expelled from the same axis.

The silencers located on the jet fans are placed in the suction and discharge nozzles of the fans to reduce the noise. The manufacturer catalogs contain sound pressure levels of jet fans at certain distances.

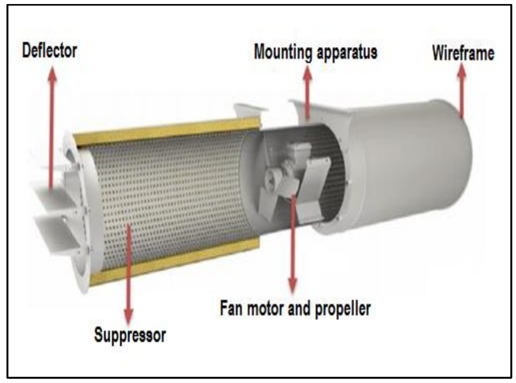


Fig. 1. Axial jet fan [9]

According to these data, the sound pressure level of the jet fans with silencer at a distance of 3 meters; It ranges from 60 dB (A) to 80 dB (A) in proportion to the blowing power of the jet fan. Jet fan systems, in case of fire by jet fans mounted on the ceiling, used for ventilation in the car park without laying any channel.

The most important feature that distinguishes traditional ducted smoke exhaust systems from jet fans; There is no need for ducts in the jet fan system. In ducted systems, ducts made of sheet metal are used to transport the smoke to the outside atmosphere. Resistance to air in these ducts directly affects fresh and exhaust air fans. As pressure losses increase, the power consumed by the fan and its size should be enlarged. Since there are no such losses in jet fan systems, the selected fans consume less energy.

Table 1. Temperature resistance of fans according to EN 12101-3 standard

Class	Temperature (°C)	Duration (minutes)
F 200	200	120
F 300	300	60
F 400	400	120
F 600	600	60

3. COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

CFD analysis principle; for each cell within the numerical solution network, by dividing the flow area into small control volumes; It is based on the solution of Navier-Stokes equations by repeating them until a converged solution appears. 3D drawing of the car park is required before starting CFD analysis. In this study, the parking lot was drawn using the Blender program. Blender 20 modeling, motion tracking, video editing, creating games is a software that supports the whole of the three-dimensional works such as animation and simulation creation [10].

A software is required for the CFD analysis of the 3D designed model. Ansys 18.2 software program was used in this study. For parking lots; It use to 3D modeling with CFD analysis to see the temperature, gas, smoke concentration and air velocities at the design stage without the need to experiment. In this way, the accuracy of the positions of the jet fans placed in the parking lot, the way the smoke will emerge in the event of a fire, determine the adequacy of the fan capacities; At the same time, it helps to have foresight in many issues such as the effect of obstacles in the car park on the air speed, distribution of air coming from the ramps and openings [11].

CFD Simulation of fire spreading & spread of smoke in a basement, to model fired heat generation temperature dependent floatation effects of smoke concentrations of species, resulting from the exhaust velocity and pressure changes, fresh air and Jet fans in operation. CFD analysis provides a detailed picture of the smoke path inside the basement and different parameters like smoke concentrations, smoke temperatures, visibility levels and velocity positions at different times and different velocity profiles inside the basement.

When applying a CFD package to undertake a flow and thermal analysis, there are number of steps that involved for completing the CFD process.

- ✓ Defining the geometry and domain
- ✓ Selecting physical sub-models
- ✓ Specifying boundary conditions at the frontiers of the domains including walls, inlets, openings, supply fans, exhaust fans, Jet fans, fire location and burning material properties.
- ✓ Discretizing the mathematical equations, which includes creating a mesh (which sub divides the space into small volumes), setting time steps (which divides the time into discrete steps) and selecting numerical sub-models
- ✓ Monitoring the iterative solution process
- ✓ Analyzing the solution obtained
- ✓ Uncertainties that may arise at each of these steps are highlighted
- ✓ Visualize the obtained solution

3.1 Governing Equations

Governing equations are two types based on compressible and incompressible fluid. To build a mathematical representation of fluid zone and solve numerically; The general preservation of the mass equation indicates that the mass storage rate within a given control volume is compensated by the net mass flow rate by convection due to density changes. In the case of a constant flow, the conservation of the mass equation indicates that what flows in it must occur (Cox 1995) [12].

$$\frac{\partial \rho}{\partial t} + \nabla . \rho u = 0 \quad (1)$$

Intensity specific density changes with time as the first term and second term defines the location of the mass transfer. Where ‘u’ is the vector describing the velocity in the u, v and w directions.

Preservation of the mass fraction "Y" of a chemical specific "i" in the presence of a vector ,

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla . \rho Y_i u = \nabla . \rho D_i \nabla Y_i + \dot{m}_i''' \quad (2)$$

The first term represents the accumulation of cases depending on the kind of change in intensity specific density, the second term is the input and the output from the control kind of volume due to convection. Right side, the data rate of production of certain types of terms in the volume control that enter via the volume control of the species or depending on the diffusion and chemical reaction caused by flowing out. The ratio of the momentum equation for the conservation of momentum of a fluid element (Cox 1995) upon movement of the Newton effect is equivalent to stating that the sum of forces is obtained by a second rule [12]. The equation is written as follows:

$$\rho\left(\frac{\partial u}{\partial t}+(u.\nabla)u\right)=-\nabla p+\nabla.\tau+\rho g+f \quad (3)$$

Here, the increase in left side momentum and inertial forces includes the forces acting on it. These forces include pressure p, gravity g, an external force vector f, and a measure of the viscous stress tensor that acts on the liquid in the control volume.

$$\left(\frac{\partial}{\partial t}(\rho E)+\nabla.(\vec{v}(\rho E+p))\right)=\nabla.\left(k_{eff}\nabla T-\sum_j h_j\vec{J}_j+(\overline{\overline{\tau}}_{eff}.\vec{v})\right)+S_h \quad (4)$$

k_{eff}, effective thermal conductivity and $\vec{J}_j$ species "'j" is the diffusion current. The first three terms on the right side, respectively, transmission of the above equation, represents the type of energy transfer due to diffusion and viscous dissipation, heat and other chemical reactions include S_h in-defined volumetric heat sources.

In the equation above

$$E=h-\frac{p}{\rho}+\frac{v^2}{2} \quad (5)$$

Where sensible enthalpy 'h' is efined for ideal gases as

$$h=\sum_j Y_j h_j \quad (6)$$

And for incompressible flow as

$$h=\sum_j Y_j h_j+\frac{p}{\rho} \qquad h_j=\int_{T_{ref}}^{T} c_{pj}\,dT \quad (7)$$

Y_j is the mass fraction of species j

3.2 Turbulence Modeling

Turbulence is characterized by a seemingly random and chaotic three-dimensional eddy fluid motion state called Turbulence. The most important result was the development of turbulence physical handling. Momentum, energy and particle transport rates in a turbulent flow greatly exceed molecular transport rates. Turbulent flows, exhibits a smaller scale compared to non-turbulent structures. When there is turbulence, it affects flow events, resulting in high energy, heat transfer and drift. If there is no three-dimensional vortex, it is not true turbulence. Because the former is the ability to produce new vortex swirl is necessary to turbulence. Three-dimensional flow, swirl flow and the required stress by itself can be rotated. As per the details received in the form of 2D layout drawings & information of ventilation system; the 3d geometrical model for the Basement-2 car park area of "Isparta Şehir Hastanesi" is prepared with rectilinear mesh dimension of 0.1 to 0.35 m is formed and overall mesh cells of 5 000000 is provide. Covered parking in fire safety is important because accidental fires can be dangerous. When fire broke out in a parking garage, which leads to greater visibility and difficulty in high temperature discharge a large amount of smoke. In most cases, victims are poisoned by smoke or starve. Also, smoke causes most of the damage to vehicles.

The first priority in case of fire, smoke and throw the parking lot and installation of ventilation systems must work efficiently.

This is a fast and secure way to provide relief to people in the parking lot and will also provide smoke-free areas for the firefighters to control the fire.

In this study, fire, octane fire in the car (petrol) total 4.0 MW heat dissipation rate of the fuel (according to BS 7346-7 standard) is modeled as convective combustion and radiation effects.

Fire simulation, with 4.0 MW peak rate of heat dissipation is carried out according to the following fire curve.

Fire source, 5 m long in the parking space as shown in the received input detail is represented as a car 2 m wide and 1.25 m high.

During fire simulation, according to the specifications, the exhaust fan and the jet fans will run at speeds specified in the venting section of this report.

The Jet fans will activate after 2.5 minutes of sufficient delay as per BS 7346-7 standard.

Any fire can have three stages for fire growth when material is being burned:

a) The stage of growth or flare, where average temperatures are relatively low and the fire is localized near its origin. In this stage, fire slowly grows depending on entrainment of air and availability of fuel. In case of fire in enclosed area, the growth period is very less, within short time the fire reaches post flash-over stage.
b) Where all material is fully developed in the afterglow of the fire and flames filled the entire volume of fire near the fire. During fully developed fire phase, reaches a maximum rate of heat release and smoke release rate and temperature are also maximized at this stage.
c) After the average temperature is usually defined as the degradation time of the fire was reduced to 85% of peak values. During this period, exhausted from the fuel combustion rate decreases when volatility and decay during high local temperatures will keep the fire for a while.

Table 2. Emergency ventilation/Unsteady state Fire simulation was done according to the following inputs and assumptions

Parameter	Reference Used	Value
Surface area of a horizontal plane in concerned geometry (**A**)	As per the input data	19438.59 m^2
Height of the roof (**H**)	As per the input data	4.8 m
Total Volume to be concerned (**V**)	**--**	88227.77 m^3
Ambient Temperatures used inside Car park	As per the input data	27°C
Fire location	As per the input data	As shown in the Fig. 4
Fire Size (Peak Heat Release Rate) used	As per BS standards	4 MW
Fire Material Used	--	Octane (Petrol) Fuel in the car
Soot Yield (Fraction) of fire material	BS 7346-7	0.038
Fire Heat Release Rate Curve	Experimental Fire Curve	Fig.2
Total Simulation Time	-	Till smoke clearance

Table 3. Jet fan specifications

Type	Quantity of jet fans working for the present fire point	Volume Flow Rate in CMH (Each Fan) & Thrust in Newton
AJR 400-2/4 (B)-TR-L	15	9450 CMH (66N)

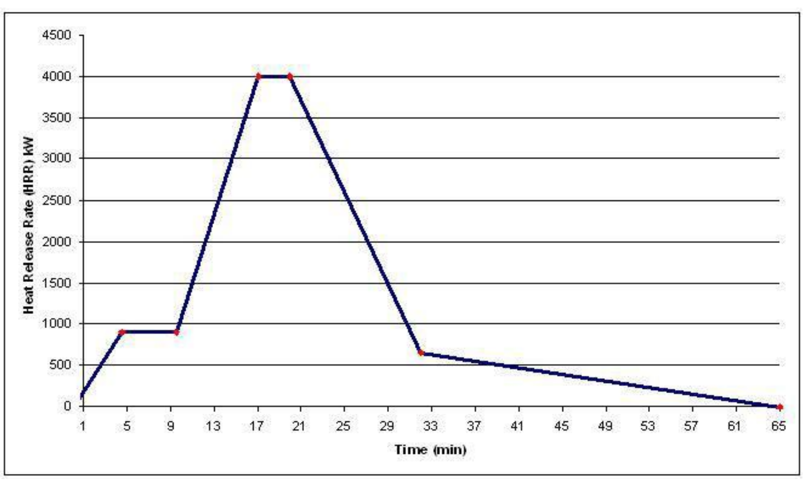


Fig. 2. Experimental fire growth rate graph considered to represent the realistic fire scenario in the CFD simulation

Table 4. Ventilation general system details

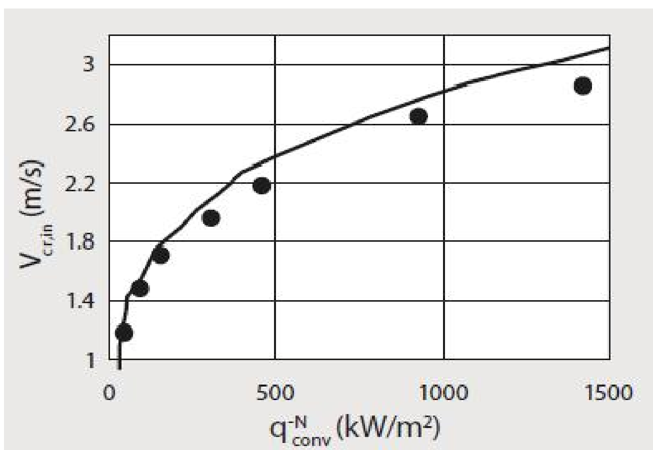
Parameter	For Fire Simulation
No. of Exhaust Fans working for Fire Point	02
No. of Supply Fans working for Fire Point	03
Positions of Exhaust fans & Fresh air Openings	As shown in Fig. 4
No. of Jet fans operating for Fire Location	15
Type of working Jet fans	AJR 400-2/4 (B)-TR-L
Positions of the Jet fans	As shown in Fig. 4
Mode of Supply Air Inlet	Natural & Mechanical
Mode of Exhaust Air Outlet	Mechanical

Table 5. Supply air shafts openings specifications

Supply Air Shafts Openings	No. of Fans	Total Volume Flow Rate in CMH
Shaft-7 Opening	01	Fan in shaft-7 is working at 40% capacity 110000 x 40% = 44000 CMH
Shaft-9 Opening	02	Fans in shaft-9 is working at 50% capacity 2 x 105000 x 50% = 105000 CMH

Table 6. Exhaust air shafts openings specifications

Exhaust Air Shafts Openings	No. of Fans	Total Volume Flow Rate in CMH
Shaft-10 Opening	01	105000 CMH
Shaft-11 Opening	01	105000 CMH



Graph 1. Critical velocity change by heat flux

4. FIRE SİMULATION CFD RESULTS

Dead spot formation can be prevented by supporting natural air conditioning with jet fans at parking lots. In case of fire, rapid smoke diffusion in the entire floor must be taken into consideration due to the low ceiling height (approximately 2.5 m) in underground parking lots. Pressure drop occurring in the entire system from the clean air inlet until the bleeder outlet must be considered when the fan group's dimensions are estimated. Ceiling beams or any obstacle must be considered when jet fans are installed. These obstacles create resistance to air flow and cause turbulence.

After the smoke is directed from the first jet fan, the momentum effect of the fan on smoke will decrease as the replacement amount increases due to several environmental factors. A turbulent flow will start and begin to disperse as the smoke gets away from this jet effect. Therefore, optimal adjustment of jet fan diffusion is a critical factor. Detecting the returns during the designing phase depends on experience in CFD programs more than having extensive fluid dynamics knowledge. In this respect, designer intervention will be more possible as CFD tools are used. The horizontal and vertical distance between the jet fans must be adjusted accurately in order to prevent the returns. As mentioned, this situation changes depending on the thermal power of heat source and parking lot height and width. graph 1 shows critical velocity change by heat flux.

(A) Following figurers shows the smoke generated from burning car inside the car park at different timings

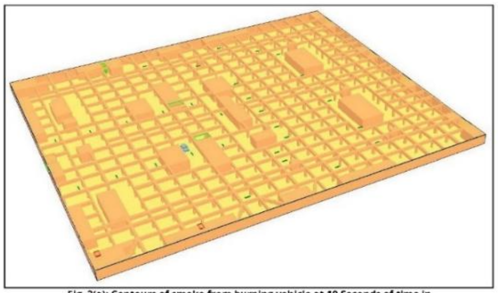


Fig. 3(a): Contours of smoke from burning vehicle at 10 Seconds of time in "Isparta Şehir Hastanesi (Basement-2)"

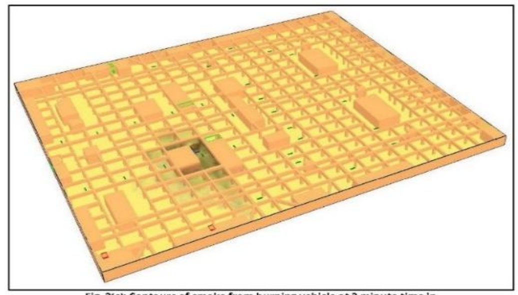


Fig. 3(c): Contours of smoke from burning vehicle at 3 minute time in "Isparta Şehir Hastanesi (Basement-2)"

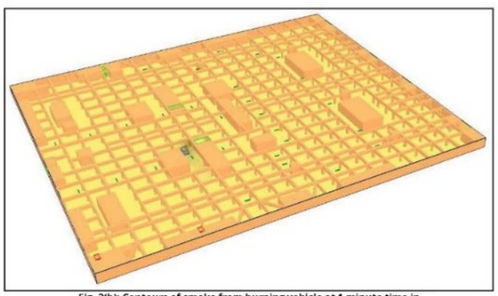


Fig. 3(b): Contours of smoke from burning vehicle at 1 minute time in "Isparta Şehir Hastanesi (Basement-2)"

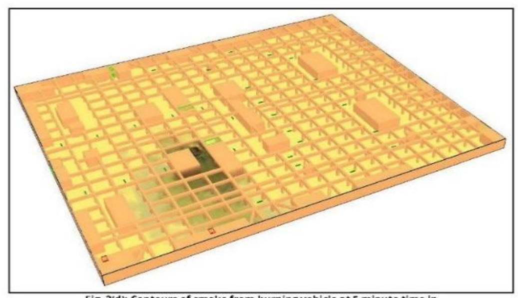


Fig. 3(d): Contours of smoke from burning vehicle at 5 minute time in "Isparta Şehir Hastanesi (Basement-2)"

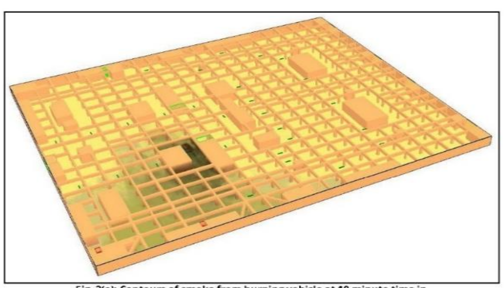


Fig. 3(e): Contours of smoke from burning vehicle at 10 minute time in "Isparta Şehir Hastanesi (Basement-2)"

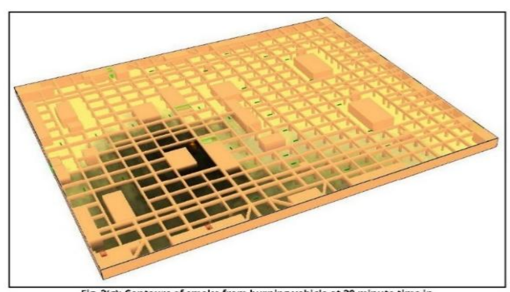


Fig. 3(g): Contours of smoke from burning vehicle at 20 minute time in "Isparta Şehir Hastanesi (Basement-2)"

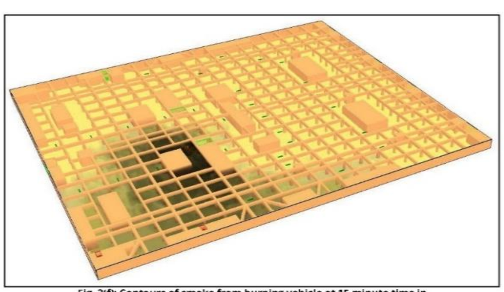


Fig. 3(f): Contours of smoke from burning vehicle at 15 minute time in "Isparta Şehir Hastanesi (Basement-2)"

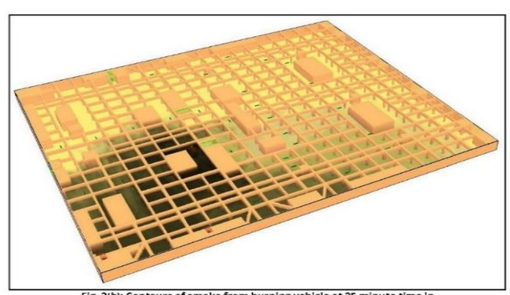


Fig. 3(h): Contours of smoke from burning vehicle at 25 minute time in "Isparta Şehir Hastanesi (Basement-2)"

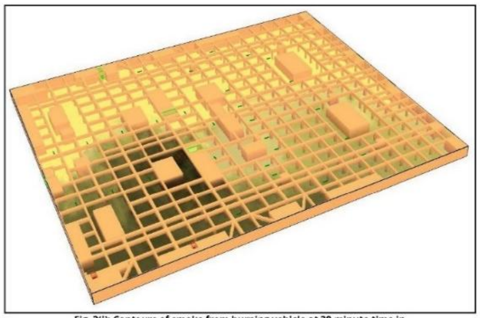


Fig. 3(i): Contours of smoke from burning vehicle at 30 minute time in "Isparta Şehir Hastanesi (Basement-2)"

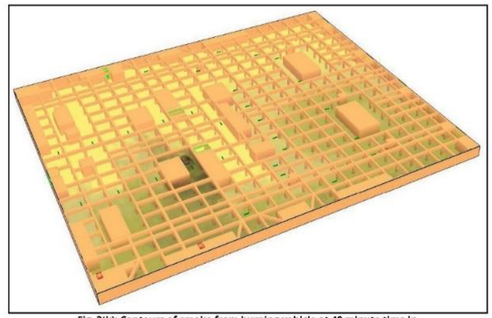


Fig. 3(k): Contours of smoke from burning vehicle at 40 minute time in "Isparta Şehir Hastanesi (Basement-2)"

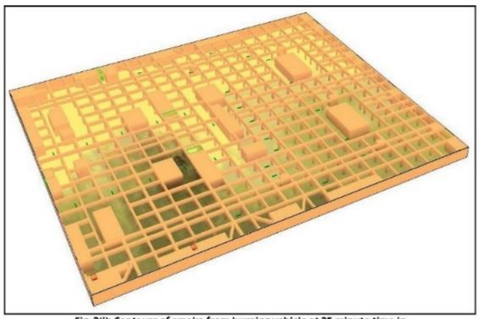


Fig. 3(j): Contours of smoke from burning vehicle at 35 minute time in "Isparta Şehir Hastanesi (Basement-2)"

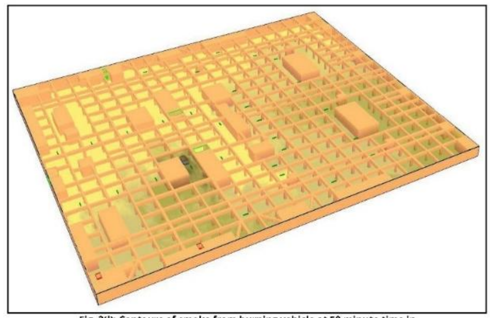


Fig. 3(l): Contours of smoke from burning vehicle at 50 minute time in "Isparta Şehir Hastanesi (Basement-2)"

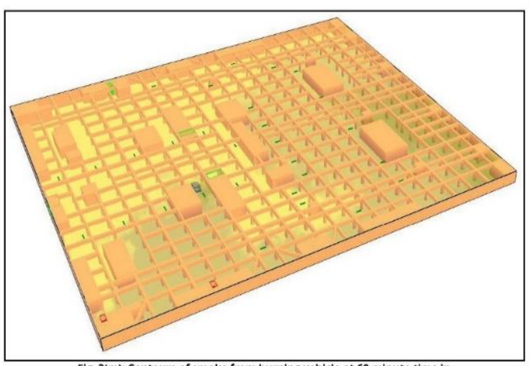


Fig. 3(m): Contours of smoke from burning vehicle at 60 minute time in "Isparta Şehir Hastanesi (Basement-2)"

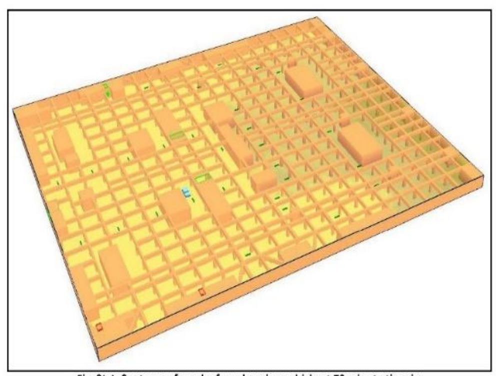


Fig. 3(o): Contours of smoke from burning vehicle at 70 minute time in "Isparta Şehir Hastanesi (Basement-2)"

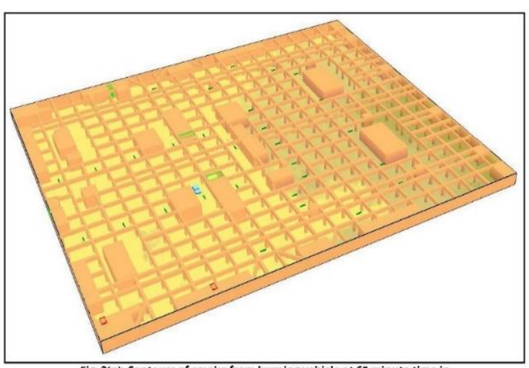


Fig. 3(n): Contours of smoke from burning vehicle at 65 minute time in "Isparta Şehir Hastanesi (Basement-2)"

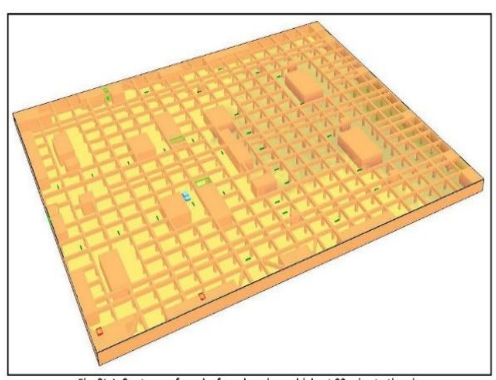


Fig. 3(p): Contours of smoke from burning vehicle at 80 minute time in "Isparta Şehir Hastanesi (Basement-2)"

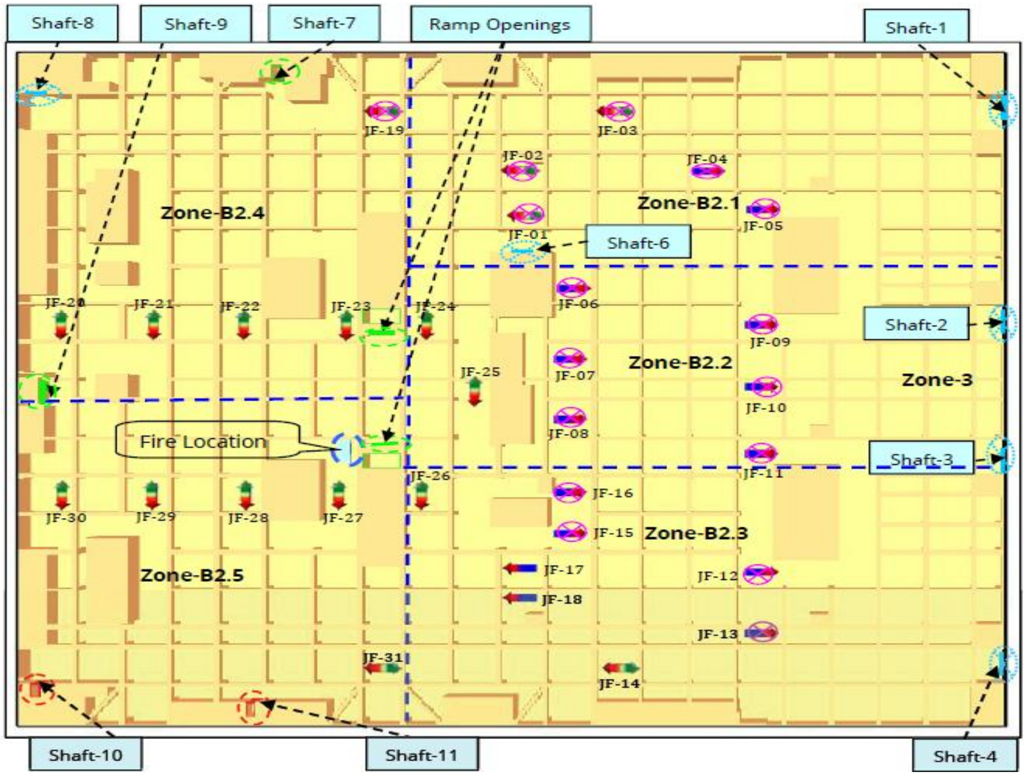


Fig. 4. Location of jet fans, ramp openings, exhaust air shaft openings and fire location in "Isparta City Hospital (basement-2)"

1. Jet fans JF-14, JF-17, JF-18 & JF-31 are working towards left side direction and JF-20, JF-21, JF-22, JF-23, JF-24, JF-25, JF-26, JF-27, JF-28,JF-29, & JF-30 are working towards downside direction and remaining Jet fans are not operating for the fire located in Zone-B2.5.

2. Shaft-7 is working at 40% capacity and shaft-9 is working at 50% capacity as Supply air shafts and shaft-10 & 11 are working at full capacities as Exhaust air shafts for the fire in Zone-B2.5 and remaining shafts are not working for the fire located in Zone-B2.5

(B) Following Simulation Results shows the Velocity Contours of the air in the car park

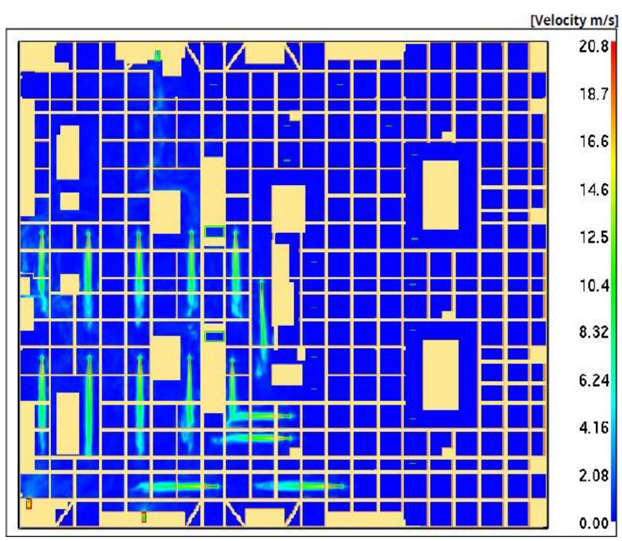
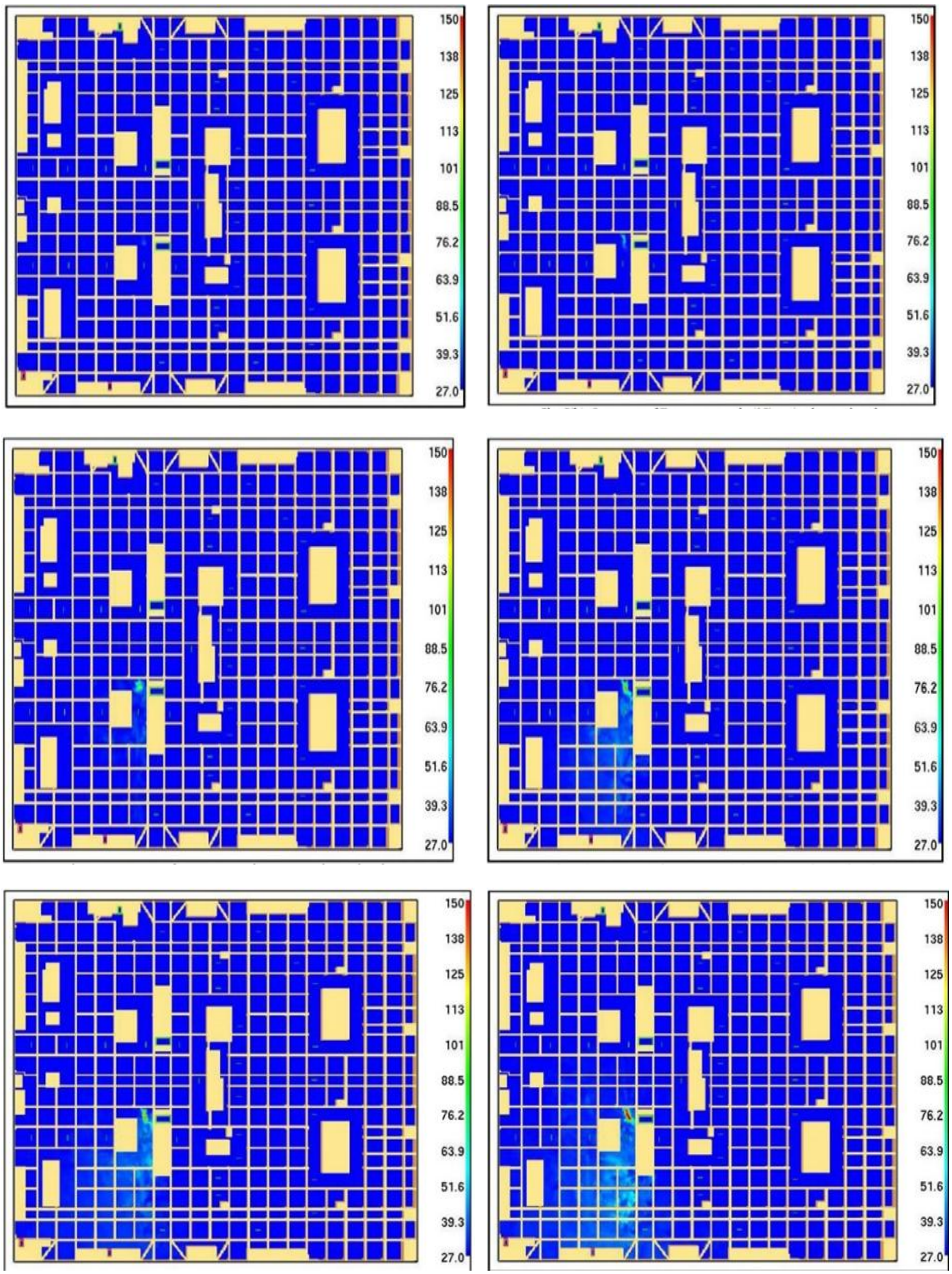
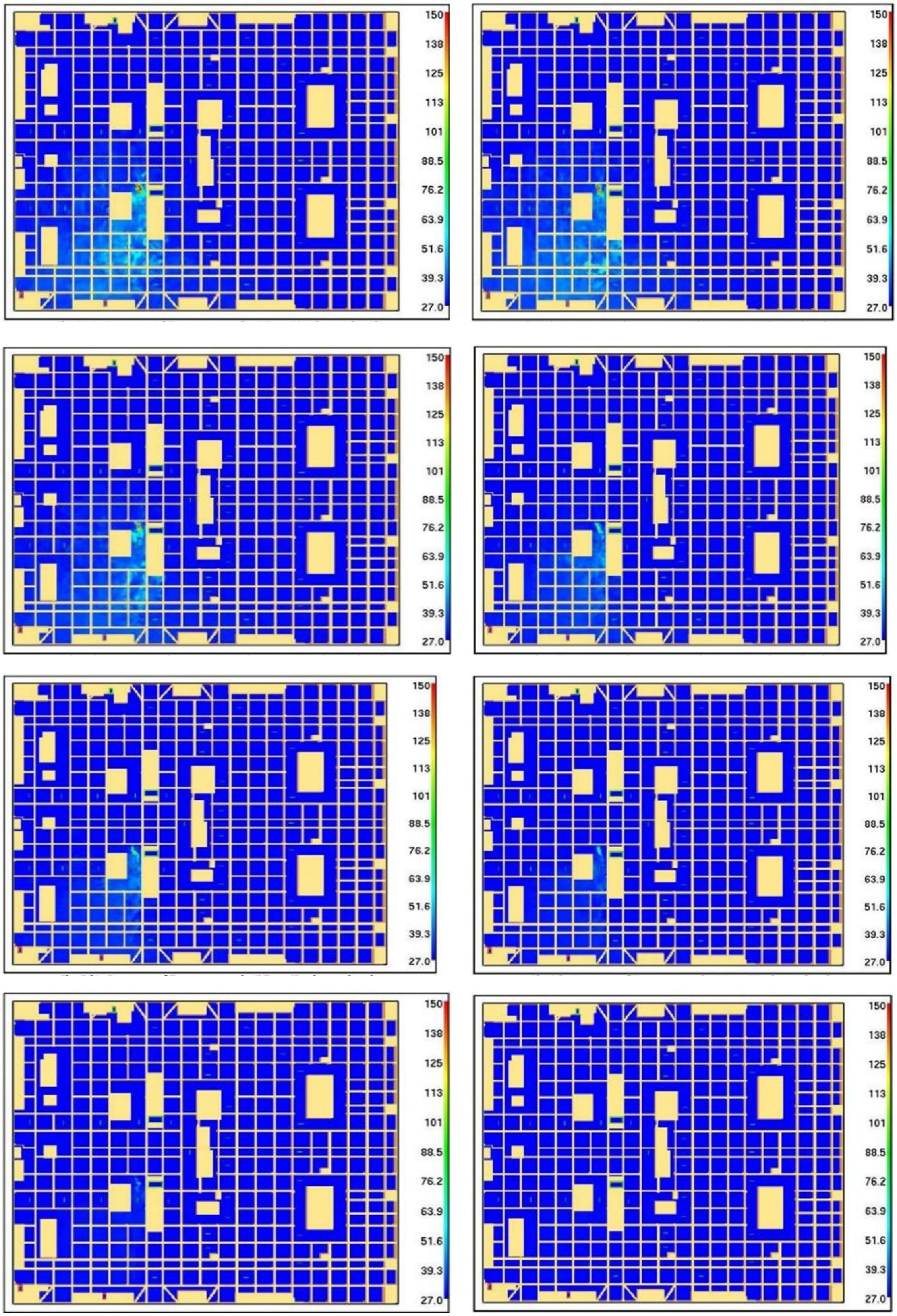


Fig. 5. Simulation results shows the velocity contours of the air in the car park

(C) Following figures shows the temperature generated from burning car inside the car park in horizontal place at 1.7 m height & at different things




150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0
150
138
125
113
101
88.5
76.2
63.9
51.6
39.3
27.0

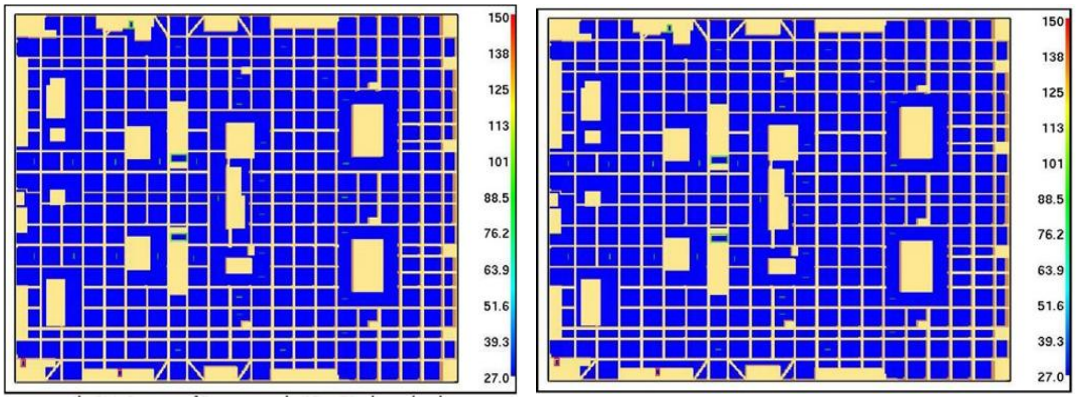


Fig. 6. Contour plot of velocity magnitude at Jet fan height (3.15 m) from ground of "Isparta City Hospital (Basement-2)"

In this study, it is aimed to be dried the red pepper which is Sena type (*Capsicum Annuum* L.) with a greenhouse type solar dryer within the province of Kahramanmaraş. Sera TARTES (www.tartes.com.tr) is a high plastic tunnel type dryer module that is designed and manufactured for the purpose of in drying process . The dimension of this structure is 8 x 6 x 2.86 m and it is conncected to solar collector on the northern frontage. The heated air in this collector is sucked from the bottom to the greenhouse. A fan of Alfan brand (2007) with 0.98 x 0.95 m dimensions of 0.3675 kW and a maximum air flow of 8500 m^3 / h was installed in order to ensure this forced convection just above the high plastic tunnel entrance door. A fan was installed in Alfan brand (2007) with a maximum air flow rate of 8500 m^3 / h and a mass of 0.98 x 0.95 m on the high plastic tunnel entrance door (Fig. 1).

4. CONCLUSIONS

"Isparta City Hospital" CFD simulations of the jet, jet ventilation system can be reached the following conclusions:

1. The fire simulation of the car park has been simulated for 80 minutes of duration with CFD; this work is carried out to examine the proposed jet fan ventilation for the 4 MW of fire located in Zone-B2.5 of Basement-2 car park.
2. As per the above CFD results for smoke distribution (refer to section – A, Fig. – 3(a) to 3(p)):

   ✓ The smoke is distributed in most of the car park area and it took less than 5 minute time to clear the smoke from the most of the zone-B2.4 & zone-B2.5 car park area.

3. As per the above CFD results for temperatures at the height of 1.7 m (refer to section–C, Fig. 6.

   ✓ The temperatures is less than 60°C all over the car park area at all times.
   ✓ The visibility is more than 5 meters all over the car park at all the times.

4. As per the above CFD results conclusions, temperature tenability criteria of "less than 60°C" and the visibility tenability criteria of "more than 5 meters" is maintaining all over the car park area at all times.

While the increase in air speeds affects the system performance positively, it can also affect the system performance negatively when a certain speed level is exceeded. While the air speeds inside continue to increase, if the amount of air activated by jet fans is more than the amount of exhaust air, the smoke will start to increase in the indoor environment since the exhaust fans cannot absorb the air that needs to be removed. For this reason, it is necessary to model jet fan systems based on calculations, to examine the speed profiles by CFD study and to determine the fan capacities in the system according to the results obtained from these CFD studies.

COMPETING INTERESTS

Authors have declared that no competing interests exist.

REFERENCES

1. Immonen E. CFD optimization of jet fan ventilation in a car park by fractional factorial designs and response surface

methodology. Building Simulation. 2016; 9(1):53-61.
2. Viegas JC. The use of impulse ventilation for smoke control in underground car parks. Tunnelling and Underground Space Technology. 2010;25(1):42-53.
3. Lu S, Wang Y, Zhang R, Zhang H. Numerical study on impulse ventilation for smoke control in an underground Car Park. Procedia Engineering. 2011;11:369-378.
4. Deckers X, Haga S, Tilley N, Merci B. Smoke control in case of fire in a large car park:CFD simulations of full-scale configurations. Fire Safety Journal. 2013; 57:22-34.
5. Li, S, Chen Y, Chen X, Jing G. Engineering numerical simulation on the amount of make-up air for mechanical smoke exhaust in atrium. Procedia Engineering. 2011;11:379-384.
6. Spiljar Z, Drakulic M, Schneider DR. Analysis of jet fan ventilation system installed in an underground car park with partition walls. journal of sustainable development of energy, Water and Environment Systems. 2018;6(2):228-239.
7. Dalkilic A, Wongwises S, Bilge S, Celen A, Dizman T, Senveli A. CFD Analysis of smoke and temperature control system of an indoor parking lot lot with jet fans. Journal of Thermal Engineering. 2015; 1(2):116-130.
8. Arini D, Pancawardani F, Santoso M, Sugiarto B, Nugroho Y. Froude modelling of fire phenomena: Observation of fire-induced smoke movement in basement structure for firefighting purpose. Procedia Engineering. 2017;170:182-188.
9. Vents, Smoke extraction ventilation for parking premises. Available:https://ventilationsystem.com/catalog/smoke-extraction-ventilation-for parking-premises
10. Blender, About Available:https://www.blender.org/about/.
11. National Institute of Standards and Technology. Fire dynamic simulator user's guide. (Sixth Edition), National Institute of Standards and Technology; 2013.
12. Cox G. Combustion fundamentals of fire, Academic press; 1995.

Peer-review history:
The peer review history for this paper can be accessed here:
http://www.sdiarticle4.com/review-history/57470