CFD ANALYSIS APPLIED TO SMOKE EXTRACTION IN CAR PARK CAUSED BY SINGLE FIRE SOURCE

Milan Marković[1]*,
Andrijana Đurđević[2]

[1]MTT - SRB Consultancy,
DOO Beograd-Stari Grad
Belgrade, Serbia

[2]Academy of Applied Technical
Studies Belgrade College for Traffic,
Mechanical Engineering and
Environmental Engineering,
Belgrade, Serbia

Abstract:

The aim of this paper is to present the application of CFD software to the calculation and to show the image of fluid flow during garage ventilation in an emergency situation. On that way, JET fans, arranged on the ceiling of the parking lot, blow smoke at high speed, transmitting complete energy to the surrounding air. Proper arrangement of JET fans ensures that the entire air mass of the parking space gets controlled movement in the desired speed range. This reduces the pressure required by the main fans to expel and inject outside air from and into the garage space, and thus the required power of these fans. Thanks to the application of CFD analysis, a graphical representation of the flow was obtained, based on when the JET fan can be selected. In this way, the selected fans and the designed ventilation system have significant advantages over the conventional duct flue system.

Keywords:

CFD analyzes, smoke extraction, JET fans, cark parks.

INTRODUCTION

The purpose of this paper is to present the findings of a Computational Fluid Dynamics (CFD) study of the Jet Thrust Fan system for car park project, which consists of three identical buildings.

Computational Fluid Dynamics (CFD) is a design tool used to aid the detailed design of Jet Thrust systems. The software allows the designer to model complex airflows within the car park, visually inspect and analyse the airflow patterns in order to make decisions as to how the final layout of the system will need to be configured.

Governing equations (the Navier-Stokes equations) are important in any fluid flow problems. Equations are either in two forms compressible and incompressible. With compressible fluids density changes significantly in response to a change in pressure (i.e. shock waves) whereas for incompressible fluids the density remains constant as a function of pressure (i.e. density is a property of the flow and not the fluid). When incompressible flow is used with an appropriate sub model for buoyancy, density variations due to changes in temperature can be computed.

Correspondence:
Milan Marković

e-mail:
kimi.kimi1988@gmail.com

DOI: 10.15308/Sinteza-2022-74-80

2. PREVIOUS RESEARCHES

The contribution of this paper is the presentation of solving specific requirements by applying the CFD method to the design of smoke systems. CFD analysis is widely used in the design of ventilation systems. During the preparation of this paper, numerous applications of this method were analyzed. The increasing application of the CFD method in the analysis of fluid flow is noted, primarily in JET ventilation systems. In paper [1], it is presented Proper Mesh Transitions between different parts and velocity analysis, where the currents are clearly defined. Paper [2] is very useful by the reason that reduced-scale modelling is applied to tunnels equipped with axial jet fans. By the analysis flow fields in tunnels and current images of temperature profile depending on tunnel sections and time are presented, Fig. 1.

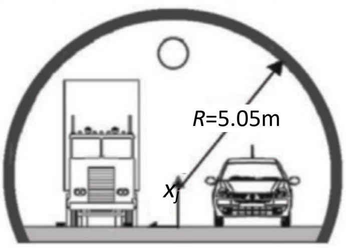
In paper [3], the application of CFD analysis to the fumigation of underground garages in the event of an initial fire on a vehicle is given. The analysis shows the spread of smoke along the height of the object as a function of time. The same paper gives the application of the finite volume method to the analysis of smoke propagation.

3. THEORETICAL BASES

Leading Navier-Stokes equations are important in any fluid flow analysis Equations are either for compressible and incompressible fluid. With compressible fluids density changes significantly in response to a change in pressure, i.e. for shock waves, whereas for incompressible fluids the density remains constant as a function of pressure because density is a property of the flow and not of the fluid. When incompressible fluid is analysed with an appropriate sub model for buoyancy analyzing, density variations due to changes of temperature can be computed. It is very practically, CFD - Solid Works Flow Simulation to be used for mathematical presentation of the car park fluid flow, by which numerically solving of 3- D Navier-Stokes equations is possible. Mesh generation is realized on the bases of the finite volume method, [4].

The law can be formulated mathematically in the fields of fluid mechanics and continuum mechanics, where the conservation of mass is usually expressed by the continuity equation, given in differential form as (1):

$$\frac{\partial \rho}{\partial t} + \nabla \rho u = 0 \tag{1}$$



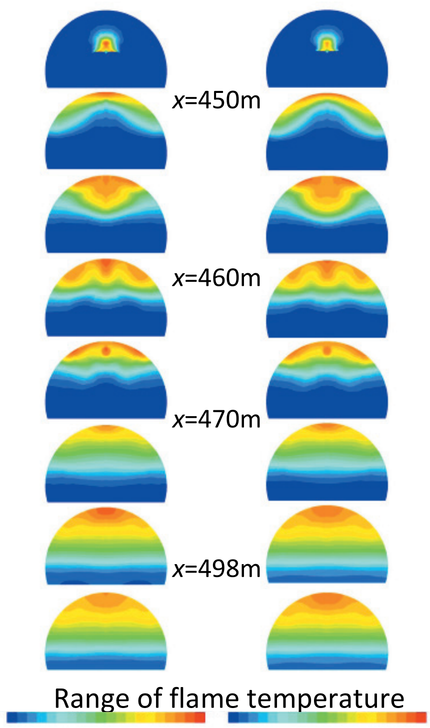


Figure 1 – Images of air flows field in tunnels and temperature profiles depending on tunnel sections and time

The first term describes the density changes with time and the second term defines the mass convection where u represents velocity vector.

Momentum equation: The equation for the conservation of momentum is derived from Newton's second law of motion, which states that the rate of momentum of a fluid element is equal to the sum of the forces acting on it. The equation is written in the form presented by (2):

$$\rho\left(\frac{\partial\rho}{\partial t}+(u\nabla)u\right)=\nabla\rho+\nabla\tau+\rho g+f \qquad (2)$$

The left hand side represents the increase in momentum and inertia forces, while the right hand side comprises forces acting on it. These forces include pressure *p*, gravity acceleration: *g*, and external force per unit mass vector f which represents a measure of the viscous stress tensor τ acting on the fluid within the control volume. The full buoyancy model will be used to calculate buoyancy effects from the fire.

Equation for conversation energy law [5] and [6], is defined by equation(3):

$$\left(\frac{\partial}{\partial t}(\rho E)+\nabla\cdot\left(\vec{v}(\rho E+\rho)u\right)\right)=\nabla\left(k_{eff}\nabla T-\sum_j h_j\vec{J}+\left(\overline{\overline{\tau}}_{eff}\cdot\vec{v}\right)\right)+S_h \qquad (3)$$

The term is the heat conduction coefficient and is the diffusion flux of species 'J'. The first three terms on the right-hand side of above equation represent energy transfer due to conduction, species diffusion and viscous dissipation respectively. The S_h term represents any other defined heat source.

4. POLLUTION VENTILATION BY CFD SIMULATION METHODOLOGY

The initial point for the construction of the geometric model is a sketch realized by AutoCAD of the car park's geometry, Fig. 2, extruded and hollowed to create the car park. The internal geometry of pillars, walls, service rooms and other reserved spaces are then extruded from inside the model using the same method. Where appropriate the geometry of the car park has been simplified to speed up the calculation and analysis time.

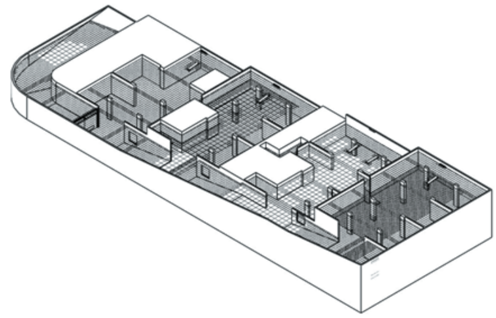


Figure 2 – Computational mesh generation

The mesh specifies the computational domain in which the simulation takes place by splitting the model into individual cells. The equations of state are applied to the individual cells using partial differential equations. The equations are solved on an iterative basis to calculate the air movement and dilution of passive scalar quantities such as Carbon Monoxide or fire smoke in order to [7] and [8].

The mesh size density is crucial to the desired accuracy of the simulation result. If the mesh is too coarse the airflow will not be correctly modelled leading to a misrepresentation of the system. Conversely a mesh which is too fine will exponentially increase the calculation time. Initial location of the fire source is presented on the sketch, realized by CAD software, Fig. 3.

The ventilation system is operating in high pollution mode at the start of the simulation, according to the ventilation mode set-points defined in Table 1. Post processing of CFD results characteristics of ventilation in mode 1, when main fan operation is on the rate of 60% speed. Decreasing of carbon monoxide pollution depending on time on that mode is presented by diagram, Fig. 4. Such analysis is equivalent to analyses realised and presented in [9], [10] and [11]. The main contribution of analysis presented in this paper and illustrated below is shift from channel conception of car parks which was designed by conventional approach to JET fans concept on which CFD method has been applied.

	Thrust fans reference	Thrust fan operation	Main fan reference	Main fan operation	Extract duty [m^3/h]
Mode 1 CO>30ppm	JT-01,JT-02, JT-03, JT-04,JT-05	ON at 50% speed	Extract	On at 60% speed	32500

Table 1 – Ventilation mode set – points for the speed on the rate of 60%

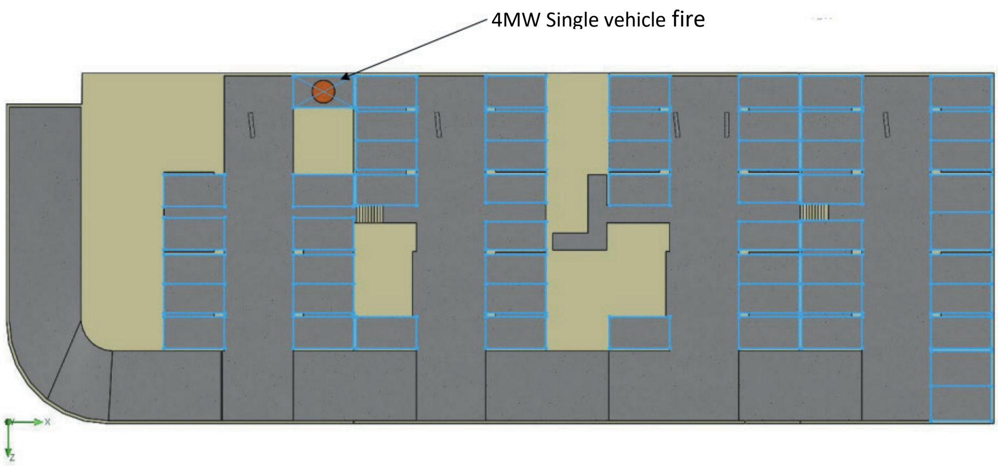


Figure 3 – Single vehicle fire source

The car park is filled with Carbon Monoxide [CO] to create a blanket condition of 50ppm throughout the entire car park (as a worst case scenario).

The ventilation system is activated at the start of the simulation. The equations of state are calculated as a transient/time dependant simulation. The gradient of the decay of CO can then be plotted against time.

The ventilation system is operating in emergency ventilation mode at the start of the simulation according, to the ventilation mode set-points defined in Table 2. Post processing of CFD results characteristics of ventilation in mode 2, when main fan operation is on the level of 100% speed.

The power of 4MW fire is started in the car park in a worst case location. The rate of growth of fire development is set to very fast so that the maximum heat release is reached quickly.

The quantity of smoke produced by the fire is also at its maximum value assuming a constant soot yield of 10%. The heat of combustion of the fire is assumed to be 26MJ/kg.

The time-step used for transient simulations of this type is typically in the range of 0.1 - 0.25s. The time-step is evaluated on a case by case basis according to:

- The rate of convergence of partial differential equations describing the motion and thermal properties of the fluid,
- The characteristic mesh length scale, aspect ratio and skew ness,
- The rate of change of goal dependant and non-goal dependant transport variables.

	Thrust fans reference	Thrust fan operation	Main fan reference	Main fan operation	Extract duty [m^3/h]
Mode 2 Fire detected	JT-01,JT-02,JT-03 JT-04,JT-05	ON at 100% speed	Extract	On at 100% speed	54000

Table 2 – Ventilation mode set – points for the worst case location and speed rate of 100%

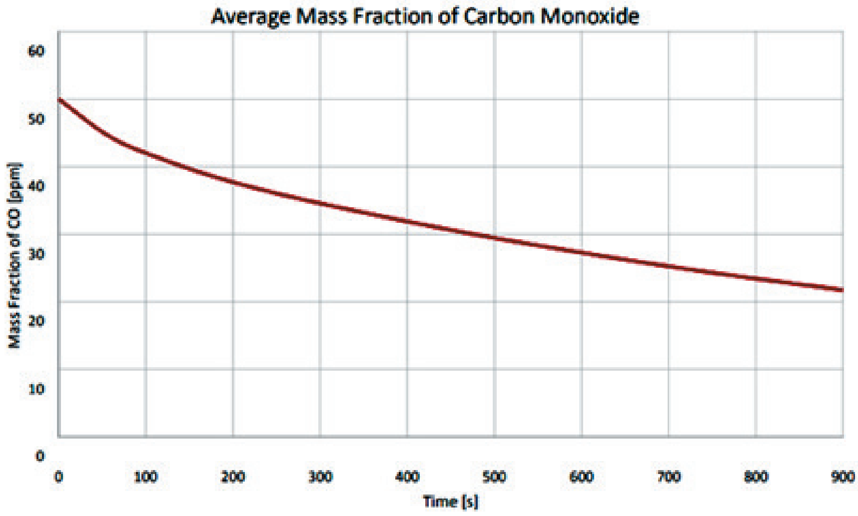


Figure 4 – Carbon monoxide pollution depending on time

Equations: (4) to (7) represent time dependent carbon monoxide rate:

$$VdC = -Q'Cdt\,, \tag{4}$$

$$\int_{C_1}^{C_2}\left(\frac{1}{C}\right)dC = \int_{t_1}^{t_2}\left(\frac{-Q'}{V}\right)dt\,, \tag{5}$$

$$ln\frac{C_2}{C_1} = \frac{-Q'}{V\left(t_2 - t_1\right)}\,, \tag{6}$$

$$t_2 - t_1 = \frac{V}{-Q'}ln\frac{C_2}{C_1}\,. \tag{7}$$

If initial time t_1 is equal to zero, then follow equations: (8) to (11) in the form:

$$t_2 = \frac{V}{-Q'}ln\frac{C_2}{C_1}\,, \tag{8}$$

$$Q' = \frac{Q}{K}\,, \tag{9}$$

$$t_2 = \left(\frac{K \times V}{-Q'}\right) \times ln\frac{C_2}{C_1}\,, \tag{10}$$

$$K = \frac{t_2 - Q' \times ln\dfrac{C_2}{C_1}}{V}\,. \tag{11}$$

By integrating time from t_1 to t_2 and concentration from C_1 to C_2, it is to be got:

$$K = \frac{720 \times \left(-9.03\right) \times ln\left(\dfrac{25}{50}\right)}{5333.27} = 0.84. \tag{12}$$

In equation (12) variables represent:

K - Ventilation performance factor for incomplete mixing,

t_1=0 [s] - initial time

t_2=720[s] - time necessary for carbon monoxide to be reduced half,

V =5333.27[m^3] - Air volume of the car park,

C_2[ppm] - Carbon monoxide concentration at time t_1, ppm represent flow of particles per minute,

C_1[ppm] - Carbon monoxide concentration at time t_2, ppm represent flow of particles per minute.

According to results presented above and CFD analysis illustration of the visibility in car park in is achieved as given on Fig. 5 .

5. CONCLUSION

The paper presents the CFD method used for the analysis of behaviour in the field. The application of this method is present both in the field of mechanical engineering and in the field of electrical engineering. This paper presents the application of this method for the calculation and visualization of fluids in ventilation systems.

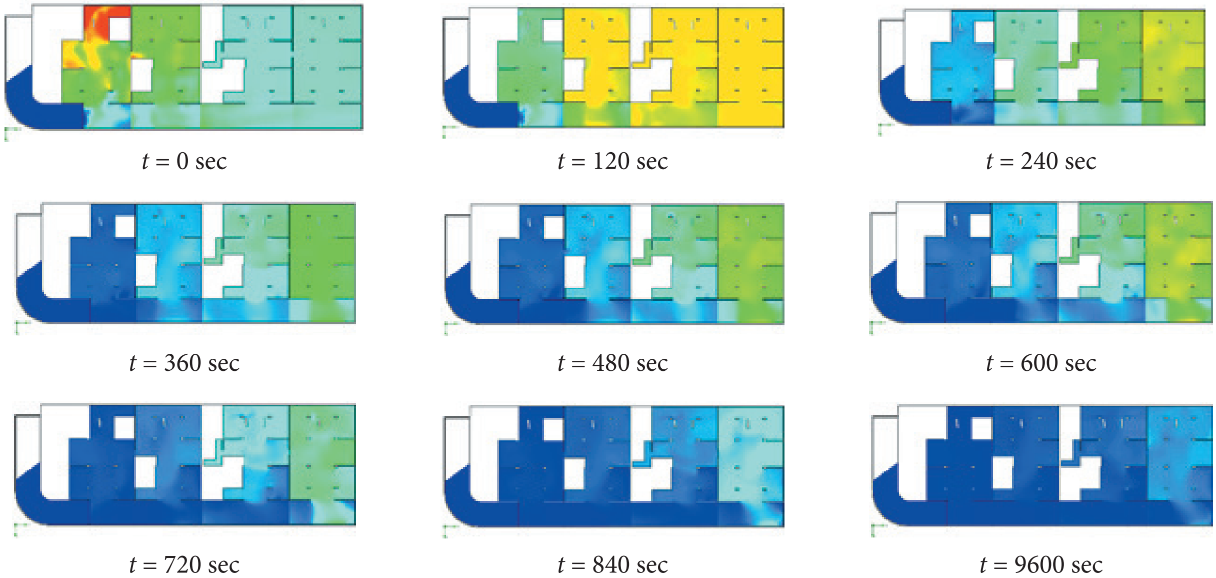


Figure 5 – CFD analysis of visibility progressing depending on time

The paper presents the application of this method to smoking a parking garage in case of fire. The aim was to give a graphical representation of the spread and removal of smoke as a function of time. In addition to the graphical representation, the method also performs the calculation of fluid flow. In the specifically processed example of parking garage smoking, this method was found to reduce the time of smoke emission, reduce the amount of smoke, increase the space for evacuation and allow easier access to the source of the fire to the fire brigade. Fluid flow behaviour is difficult and complicated to predict. The accuracy of the calculation is the most important for choosing the most efficient way of ventilation. CFD simulation checks whether there is enough air movement in all parts of the parking lot, whether a sufficient number of air changes is provided.

In order to perform CFD analysis, it is necessary to have a 3D model of the object that includes all openings, beams, ceiling surfaces as well as all other necessary details for CFD simulation. CFD simulation is used for the purpose of simulating air flow and to ensure good air distribution for efficient operation in normal and emergency situations. The program includes solutions for several relevant maintenance equations (mass, momentum and concentration). The software solves these problems until an accuracy is achieved that allows for accurate system design.

6. REFERENCES

[1] M. T. Çakir, Ç. Űn, "CFD analysis of smoke and temperature control system of an indoor parking lot with jet fans", Journal-of-thermal-engineering/Articles Yildiz Technical University Press, Istanbul, Turkey, Vol. 1, No. 2, pp. 116-130, Apr., 2015, DOI:10.9734/jerr/2020/v13i317102

[2] M. Musto, G. Rotondo, "CFD Analysis of a Realistic Reduced-scale Modeling Equipped with Axial Jet Fan". Journal on Fire Safety, Vol. 74, Pages 11-24, May 2015, https://doi.org/10.1016/j.firesaf.2015.03.006

[3] Van Oerle N., Lemaire A., Van de Leur P. "Effectiveness of Forced Ventilation in Closed Car Parks", TNO Report No. 1999-CVB-RR11442, TNO, Delft, 1999,

[4] Aveiro J. L., Viegas J.C. "Smoke control in an underground car park with impulse ventilation", Laboratorio Nacional de Engenharia Civil, Lisabon, 2010.

[5] Jović, V. "Osnove hidrodinamike", Element, Zagreb, 2006.

[6] Z. Virag, I. Džijan, "Numerička dinamika fluida", Fakultet strojarstva I brodogradnje, Zagreb, Hrvatska, 2014,

[7] M. W. Cheng, L. Y.Xing, L. Fang, V. Nielsen, "Full-scale experiment and CFD simulation on smoke movement and smoke control in a metro tunnel with one opening portal", Tunnelling and Underground Space Technology, Vol. 42, Pages 96-104, May 2014, https://doi.org/10.1016/j.tust.2014.02.007

[8] W. Węgrzyński, G. Krajewski, "Combined Wind Engineering, Smoke Flow and Evacuation Analysis for a Design of a Natural Smoke and Heat Ventilation System". Procedia Engineering, Vol. 172, 2017, Pages 1243-1251, https://doi.org/10.1016/j.proeng.2017.02.146

[9] G.V. Hadjisophocleous, G. D. Lougheed, S. Cao, "Numerical study of the effectiveness of atrium smoke exhaust systems", NRC Publications Archive (NPArC) Archives des publications du CNRC (NPArC), ASHRAE Transactions, 105, No.1, pp. 699-715, 1999, http://nparc.cisti-icist.nrc-cnrc.gc.ca/npsi/ctrl?lang=en

[10] S.Gannouni, R.B. Maad, CFD analysis of smoke backlayering dispersion in tunnel fires with longitudinal ventilation, An international journal on fire and materials, Vol. 41, No. 6, pp 598-613, okt. 2016, https://doi.org/10.1002/fam.2394

[11] H Zhu, Y. Shen, Z, Yan, Q, Guo, Q. Guo, "A numerical study on the feasibility and efficiency of point smoke extraction strategies in large cross-section shield tunnel fires using CFD modeling", Journal of Loss Prevention in the Process Industries, Vol. 44, pp. 158-170, Nov. 2016, https://doi.org/10.1016/j.jlp.2016.09.005