FACTA UNIVERSITATIS
Series: **Working and Living Environmental Protection** Vol. 11, N° 2, 2014, pp. 65 - 74

THE IMPACT OF WIND INERTIAL AND FIRE BUOYANCY FORCES ON AIR POLLUTION IN STREET CANYON

UDC 551.55:614.842]:504.064

Dušica J. Pešić[1], Radoslav H. Kartov[2], Darko N. Zigar[1], Valentin Chochev[2]

[1]University of Niš, Faculty of Occupational Safety, Serbia
[2]Academy of MOI, Faculty of Fire safety and Civil Protection, Bulgaria

Abstract. *The air quality has become an important environmental problem worldwide. The pollutants transport in and over urban "street canyons" has attracted great attention due to their negative effects on human health and environment. The aim of this paper is to analyze the impacts of wind and fire on air pollution in a street canyon. For simulations of wind flow and fire plume flow Large Eddy Simulation method of Fire Dynamics Simulator software has been used. Numerical results show that the recirculation flow and the vortices inside the canyon are dependent on a wind inertial force and a fire buoyancy force. The fire buoyancy effect may have a significant impact on the pollutants transport in and over the canyon. For the low wind velocity cases, the dispersion of fire pollutants is controlled by its buoyancy. On the other hand, when the wind effect dominates the buoyancy effect, the pollutants are primarily driven by the wind inertial force. This complex phenomenon affects the pollutant dispersion patterns within many urban streets.*

Key words: *street canyon, wind force, fire buoyancy force, air pollution*

1. INTRODUCTION

The wind flow and pollutant transport inside and over urban street canyons have attracted great attention, during the past decades, mainly due to pollution of the local air and adverse impacts on human health and environment. A "street canyon" generally refers to a relatively narrow street flanked by buildings on both sides. It constitutes the basic geometric unit of many urban areas. This unit is also bounded by the ground-level at the bottom and the roof-level at the top. The street canyon dimensions are usually expressed by the ratio of the street width to the building height. A "regular" street canyon has an aspect ratio of one if the street width is equal to the building heights [12].

Received December 8, 2014 / Accepted December 30, 2014
Corresponding author: Dušica J. Pešić
Faculty of Occupational Safety in Niš, Čarnojevića 10A, 18000 Niš, Serbia
Phone: +381 18 529 821 • E-mail: dusica.pesic@znrfak.ni.ac.rs

The transport of gaseous pollutants in a street canyon depends generally on the rate at which the street exchanges the air vertically with the atmosphere above the roof-level and laterally with connecting streets. Street canyon is usually rather poorly ventilated by natural motions of the air. The various factors affect the air motion and the flow fields inside and above the street canyon. The wind flow is the main parameter that determines the air flow patterns inside the street canyon. Even when the wind is strong, the pollutants can be trapped in the lower part of the canyon. For this reason, the wind flow in and over a street canyon is particularly important for determination of the pollutant distribution inside the canyon. The direction of wind at the roof-level of the street canyon affects tremendously the wind flow and pollutant transport inside the street canyon. The most commonly found cases in literature are those with wind perpendicular to the street axis, because this is the worst situation for air pollutants to dilute from street canyons [1,4,11]. Apart from wind direction, the wind velocity also has an important effect on the flow fields. The strength of the vortices inside the street canyon mainly depends on the wind velocity at the street roof-level. The higher wind velocity is the more effectively for the pollutants dilution in the street [5].

The studies mentioned above have focused on vehicle exhaust gases which are trapped inside poorly ventilated urban complex geometries. However, there is another realistic scenario when the fire products rapidly contaminate the air inside the street canyon. The buoyancy-driven plume rises up straight vertically, and in case of the wind flow at the street roof-level the plume starts to bend toward the wind direction. If there is strong wind, the portion of the fire plume is re-entrained into the street canyon. Therefore, it is necessary to examine this recirculation phenomenon in a street canyon caused by buoyancy effect of fire and inertial effect of wind flow.

In this paper, the wind effect on the dispersion of fire pollutants in a street canyon is analyzed. Mean flow and turbulent fluctuations in a "regular" street canyon have been performed. Large Eddy Simulation (LES) method of Fire Dynamics Simulator (FDS) software is used for prediction of flow field patterns occurring in and over street canyon.

2. Theoretical Background

The most important features of local air quality are the wind-induced flow patterns in a street canyon. There are intermittent and unsteady wind flow fields inside the canyon. The wind flow pattern inside street canyons depends on their geometry and wind direction at the roof-level. When the wind direction is perpendicular to the street axis, a vertically rotating wind flow is created with a centered primary vortex inside street canyon whereby the wind flow at street ground-level is opposite to the flow above roof-level. The formed vortex confines the air flow, reduces the pollutants dispersion and increases the pollution concentrations inside the street canyon.

The total number of vortices and their intensities depend mainly on the canyon aspect ratio. Oke [8] has identified three flow regimes for wind direction perpendicular to the street axis in neutral stratification: skimming flow, wake interference flow, and isolated roughness flow. In case of widely spaced buildings, the flow fields do not interact with the buildings, which results in the isolated roughness flow regime. The reducing spacing between buildings results in the wake interference flow regime when the wake behind the upwind building is disturbed by the recirculation created in front of the windward building. Further reduction of spacing between the buildings results in the skimming flow regime. In this regime a stable recirculation flow is developed inside the canyon. Under these conditions, the pollutants at the street ground-level could not be easily ventilated and consequently this results in high pollutant concentration and poor quality of the air.

All the above-mentioned regimes are for situations without a pollutant source in the street canyon. The more realistic situation is the fire as a strong buoyancy and turbulence source that changes the vortex characteristics in the canyon. The competition of buoyant effect of fire and inertial effect of the wind flow also affects the pollutant transport within the street canyon.

Buoyancy forces arise as a result of variations of density in a fluid subjected to gravity, and produce a wide range of important phenomena for fluid mechanics. The strong buoyancy forces occur in case of the fire. Due to high temperature and low density, they force the gaseous products directly upwards. In fluid dynamics, the turbulent flow conditions can be described by numerous dimensionless numbers. The Froude number and the Richardson number can be used to characterize the flow field inside and above the street canyon [9].

Froude number modelling is often used as the scaling technique in fire situations where the Reynolds number is sufficiently large, turbulent conditions prevail and buoyancy forces are dominant. The Froude number represents the ratio of inertial force and buoyancy force. It is defined as:

$$Fr = \frac{u^2}{gH} = \frac{\text{inertial force}}{\text{gravity force}} \tag{1}$$

where g - acceleration due to gravity, u - wind velocity at roof-level, H - building heights in the street canyon.

The Froude number is used to examine the relative importance of buoyancy effects on flow. The buoyancy force exceeds the inertial force when $Fr < 1$. In fires, the flow is usually buoyancy dominated with a Froude number below one [3].

Froude number is also combined with the density ratio of the fire gaseous products and the ambient air ($\Delta\rho/\rho_\infty$) to include the effects of stratification. It is then called modified Froude number or the Richardson number, defined by:

$$Ri = \frac{gH}{u^2}\frac{\Delta\rho}{\rho_\infty} = \frac{1}{Fr}\frac{\Delta\rho}{\rho_\infty}\,. \tag{2}$$

At low Richardson number, the density difference between the pollutants and the air is large and the wind velocity high enough so that the buoyancy effect may be ignored. However, below a critical value, buoyancy force becomes important enough to affect the overall fluid flow pattern. A high Richardson number indicates that the buoyancy force will dominate the inertial force and minimize mixing between the fire pollutants and the fresh air [9].

3. NUMERICAL SIMULATION

3.1. Methodology

The rapid development of numerical algorithms and computer hardware has led to the development of Computational Fluid Dynamics (CFD) based field models applied to fire research problems. Virtually all CFD models are based on the conceptual framework provided by the Reynolds-averaged form of the Navier-Stokes equations.

FDS software, developed by National institute of standard and technology, is now a popular CFD tool in fire researches. But, it is also widely used for investigation of wind flow fields and pollutant dispersion in street canyons. FDS solves numerically a form of the Navier-Stokes equations approximate for low-speed, thermally-driven fluid. The transport equations are simplified using techniques developed by Rehm and Baum [2] and they are widely referred to as the low Mach number equations, which describe the low speed motion of a gas driven by chemical heat release and buoyancy forces.

FDS uses Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) methods for calculation the fire parameters. In this study, LES is used because it can predict the unsteadiness and intermittency of the turbulence structure, which is the most important feature of a strong buoyancy-driven flow. LES uses the classical Smagorinsky model as its sub-grid scale (SGS) model for turbulence modelling [10]. The governing equations for LES simulation are, as follows [7]:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{u} = \dot{m}_b''' \tag{3}$$

Conservation of individual gaseous species:

$$\frac{\partial}{\partial t}(\rho Y_\alpha) + \nabla \cdot \rho Y_\alpha \mathbf{u} = \nabla \cdot \rho D_\alpha \nabla Y_\alpha + \dot{m}_\alpha''' + \dot{m}_{b,\alpha}''' \tag{4}$$

Conservation of momentum:

$$\frac{\partial}{\partial t}(\rho \mathbf{u}) + \nabla \cdot \rho \mathbf{u}\mathbf{u} + \nabla p = \rho \mathbf{g} + \mathbf{f}_b + \nabla \cdot \tau_{ij} \tag{5}$$

Conservation of energy which is written in terms of the sensible enthalpy:

$$\frac{\partial}{\partial t}(\rho h_s) + \nabla \cdot \rho h_s \mathbf{u} = \frac{Dp}{Dt} + \dot{q}''' - \dot{q}_b''' - \nabla \cdot \dot{q}'' + \varepsilon \tag{6}$$

Equation of state for a perfect gas

$$p = \frac{\rho R T}{M} \tag{7}$$

where ρ - density, $\mathbf{u}$ - velocity vector, $\mathbf{u} = [u,v,w]^T$; T - temperature; D_α - diffusion coefficient; Y_α - mass fraction of αth species; $\dot{m}_\alpha'''$ - mass production rate per unit volume of species α by chemical reactions; $\dot{m}_{b,\alpha}'''$ - production of species α by evaporating particles; p - pressure; $\mathbf{g}$ - is gravity vector $(0,0,-g)$; $\mathbf{f}_\mathrm{b}$ - external force vector; τ_{ij} - viscous stress tensor; h_s - sensible enthalpy; $\dot{\mathbf{q}}''$ - heat flux vector; $\dot{q}'''$ - heat release rate per unit volume from a chemical reaction; $\dot{q}_b'''$ - energy transferred to the evaporating droplets; $\dot{q}''$ - conductive and radiative heat fluxes; ε - dissipation rate R - universal gas constant, M - molecular weight of the gas mixture, t - time.

Since mixing of air and combustion products occur in the plume above the fire where buoyancy is the dominant, it is necessary to identify the vorticity sources. In order to determine the sources of vorticity as well as the boundary and shear layers, in case when the fire products move without influence of any external forces, the following equation is applied for the numerical solution of the momentum equation

$$\frac{d\Gamma}{dt} = \oint \frac{1}{\rho_\infty}\left(1 - \frac{\rho_\infty}{\rho}\right)\nabla p \cdot dx + \oint \frac{\rho\text{-}\rho_\infty}{\rho} g \cdot dx + \oint (\nabla \cdot \tau_{ij}) dx \tag{8}$$

The first term on the right hand side represents the baroclinic torque; the second term is buoyancy-induced vorticity; and the third term represents the vorticity generated by viscosity or sub-grid scale mixing, as in boundary and shear layers.

3.2. Model configuration

A computational domain (24 m wide, 32 m long and 26 m high) was designed for CFD LES simulation. The spanwise computational domain consists of a street canyon at the bottom and a shear layer over the buildings. The building rows at two opposite sides of the domain were formed a so called “regular” street canyon (the street width and the building heights are set at 16 m). Two parameters of canyon shape, H/W and H/L, were defined as 1.0 and 0.5, respectively (H - building heights, W - distance between buildings, e.g. street canyon width, and L - building lengths). This geometry was expected to produce a skimming flow regime in the street canyon. The 3-D simulation domain built by FDS is shown in Figure 1.

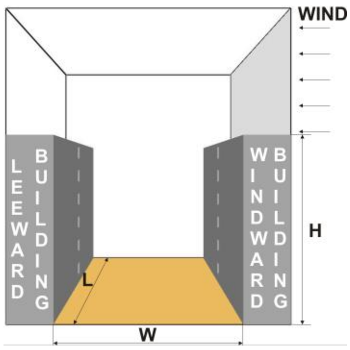

Fig. 1 The 3-D simulation domain built by FDS

The buoyancy sources were car fire and compartment fire in the windward building. Pool fire (i.e. car fire) was set at the centre of the street canyon. The compartment fire was set at the first floor of windward building besides the street canyon. The compartment had the open window (2 m wide and 2 m high), which was in the middle of the wall facing street canyon. The other windows at the upper floors of this building and all windows of the leeward building were closed during the simulations. Reaction types of "OCTANE" for car fire, and "SPRUCE" for compartment fire, according to FDS reaction database [7] were specified for generating combustion products from the fire source. The both fire buoyancy sources had the heat release rate (HRR) of 5 MW.

When applying LES simulation, the grid size is the key parameter which has to be considered very carefully, in order to produce reliable simulation results. The grid size is initially determined by the non-dimensional expression $D^*/\delta x$ where D^* is the characteristic fire diameter and δx is the nominal size of a grid cell. Characteristic fire diameter is defined as [7]

$$D^* = \left(\frac{\dot{Q}}{\rho_\infty c_p T_\infty \sqrt{g}} \right)^{\frac{2}{5}} \tag{9}$$

where, $\dot{Q}$ - HRR, c_p - specific heat of smoke, T_∞ - ambient air temperature.

Investigations of McGrattan et al. [6] suggest that the grid size must be no larger than $0.1D^*$ to obtain viable simulation results. For the HRR of fires ($\dot{Q}$ = 5 MW) used in the simulations, D^* is computed to be 1.97 m, then $0.1D^*$ is approximately 0.2 m which can be taken as a reasonable grid size. For this reason, the grid was uniform of 0.2 m in the three spatial directions (*x*-, *y*- and *z*- direction). The number of grid cells was 2,457,600 (120×160×128 in the *x*, *y* and *z*- direction, respectively).

This grid is not fine enough to resolve the boundary layer at the building walls. For the wall function and the treatment of the vicinity, a tangential velocity boundary condition is used to control how the gases "stick" to solid surfaces. Ideally, the tangential component of velocity is zero at the surface, but increases rapidly through a boundary layer that is only a few millimeters thick to its "free-stream" value. For this reason, the FDS LES turbulence model permits to specify either no slip or half-slip boundary conditions for near-boundary velocities. The half-slip assumption was used in the simulations since the finer grid requires more computation resources and a longer computing time.

To model the atmospheric air flow, the instantaneous or "real" wind has to be specified. The typical atmospheric wind profile of the form $u = u_o(z/z_o)$ was used at the right side of the simulation domain. The wind was set to be perpendicular to the axis of the street canyon. The top and the other three sides of the domain were all set to be naturally opened in order to gases flow freely in and out of the simulation domain.

The canyon vortex dynamics and the interaction of the wind force and fire buoyancy force were investigated for the cases of the wind flows at the roof-level of the street canyon. The simulations were carried out for different wind velocities, i.e 0.3; 5; 7.5; 10, 15 and 20 m/s.

4. RESULTS AND DISCUSSION

The air pollution investigations require the examination of interaction of the air in the street canyon with the air flow above the canyon. In this respect, the wind vertical velocity and the air distribution in and over the canyon were estimated for the wind flow cases across the top of the canyon. Due to the fact that the particularly unfavourable ventilation conditions exist in the street canyon when the wind is oriented perpendicular to the canyon axis, in order to investigate the flow patterns and the air pollution inside the street, the velocity vectors of the wind and fire buoyancy sources were analyzed.

4.1. Wind flow fields without fire

The velocity vector fields in and over the street canyon, under different velocities of perpendicular wind flow, are shown in Figure 2.

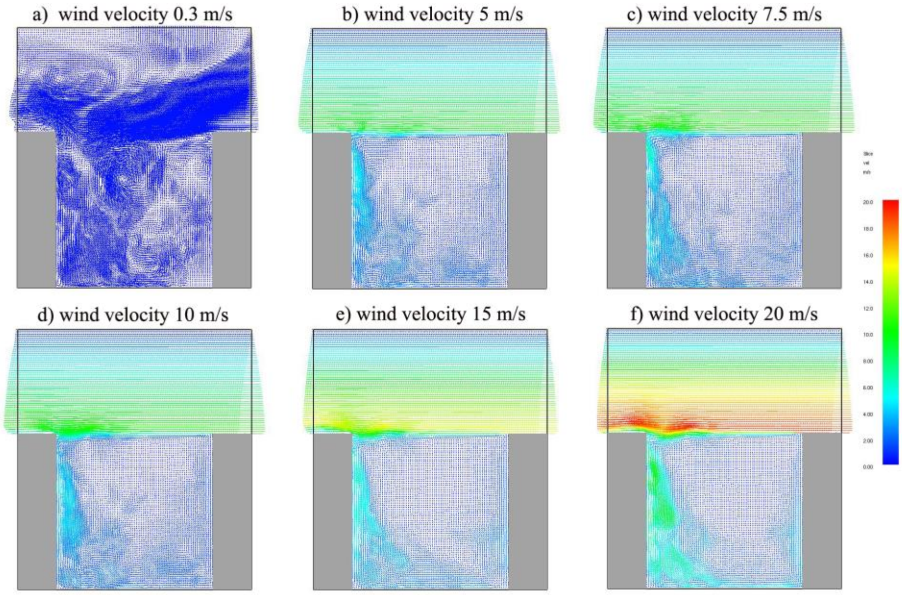

Fig. 2 Velocity vector fields without fire

As shown in Figure 2, a large recirculation flow is mainly formed with still some small vortices inside the street canyon. The main recirculation flow came down into the street canyon along the wall of the leeward building and flowed, along the street ground-level in the windward direction. Later, the recirculation flow touched the wall of the windward building and turned upward along the wall. Then it is suppressed by the wind to flow in the horizontal downstream direction when reached the street roof-level and is re-entrained back into the street canyon.

In case of the wind velocity of 0.3 m/s, the wind flows slowly over the street canyon and when it touches the leeward building, its direction is changed and then it flows along the windward side of the street forming a vortex inside the canyon. In conditions of higher wind velocities, the wind intensifies the flow over the street canyon. When the wind touches the leeward building, there is lower inflow inside the street canyon**.** On the other hand, there is a large vortex that covers most of the street canyon. It should also be noted, that under the high wind velocities, there is an “internal boundary layer”, which means that the air exchange between the flow within the street and the outer flow is impeded.

The results indicate that the canyon vortex dynamics is strongly dependent on the wind velocity at the roof-level of the street canyon. With the increase of the velocity of the wind flow, the recirculation flow velocity also increases. The variations of wind velocity have an influence mainly on the air vertical exchange between in the canyon and the outer flow.

4.2. Flow fields with fires in street canyon

The fire in the street canyon causes the complex interaction between the wind recirculation flow and turbulent fire plume as a strong buoyancy force. The vector field patterns in and over the street canyon, under the fires inside the canyon and different velocities of wind flow at the canyon roof-level, are shown in Figures 3 and 4.

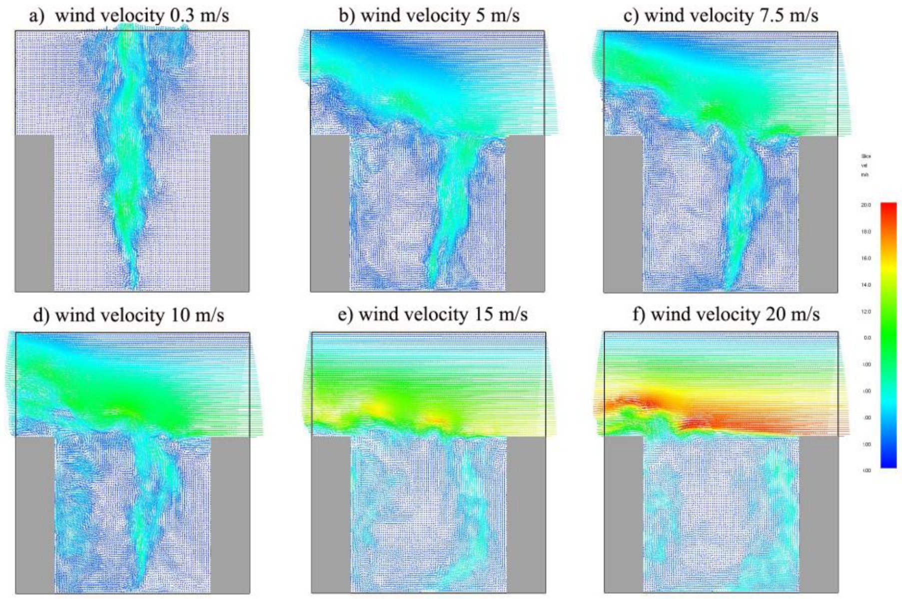

Fig. 3 Velocity vector fields with the car fire in the street center

In case of the car fire (Fig. 3), a freely standing plume is formed at the street canyon center. It is carried by strong buoyancy force directly upwards. The ambient fresh air from surroundings enters into the fire plume due to high difference of their temperatures. In compartment fire case (Fig. 4), a spill plume is formed and attached to the external wall of windward building, which differs from a freely standing plume in the street center. For such plume, the fresh air entrainment from the wall side is restrained. Both plumes generated by the fires rise up vertically with constant velocity of 7.5 m/s.

When the wind velocities at the street roof-level are lower then the fire plumes velocity, due to turbulent diffusion of the air into the fire plumes, the small vortices are formed inside the street canyon. But, when the wind velocities are higher then the plumes velocity the strong vortex is formed in the canyon. It should be noted that both flow velocity and flow turbulent fluctuation velocity increase with increase of the wind velocity. The vortices formed into the street lead the emitted fire pollutants in a direction that is opposite to wind direction above the roof-level. The leeward side of the street is directly exposed to pollutants emission. The windward side of the street is, on the contrary, only exposed to background pollution and to pollutants emission from the fires, but significantly diluted by the recirculation vortex. Consequently, the concentrations are much lower than on the leeward side of the street canyon.

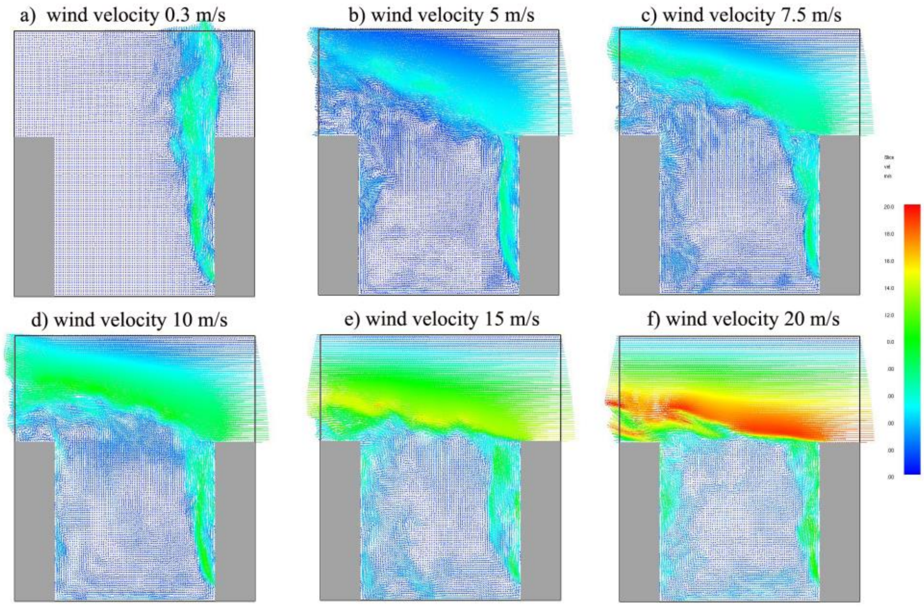

Fig. 4 Velocity vector fields with the compartment fire in the windward building

In cases of the strong wind, the exchange of the pollutants and the air above the street roof-level is limited. Near the street ground-level the fire pollutants are not diluted and removed, which results in high pollutant concentrations inside the street.

The obtained results indicate that the flow characteristics and the dispersion of fire pollutants in and over the street canyon are controlled by inertial force of the wind and buoyancy force of the fire. The interaction of the buoyancy force and the inertial force is a complex phenomenon. Namely, the buoyancy force makes effort in making the pollutants leave the top of the street canyon. On the other hand, the inertial wind force opposes the buoyancy force and tends to push the pollutants to return back into the street canyon.

5. CONCLUSION

In this paper FDS LES simulations were performed to study the interaction of a buoyancy effect of the fire and an inertial effect of the wind flow in and over an urban street canyon. The formed vortices inside the canyon depend on those effects.

Visualization using the vector velocity fields revealed that there are two different flow patterns. The first flow pattern arises inside the street canyon as a consequence of the wind flow at the street roof-level. Since street canyon has an aspect ratio of one, there is the large recirculation flow with the vortex that covers most of the canyon. The wind velocity at the roof-level affects the air exchange between the street and the outer flow. The second flow pattern arises as consequence of the fires inside the street canyon. The vortices formed inside the canyon mainly depend on the fire buoyancy force. If the buoyancy force is larger than the wind inertial force, there is not the wind vortex. When the wind flow is larger than the plume flow, the wind inertial force has the significant impact on the transport of fire pollutants in and over the street canyon.

However, there are many other factors that affect the pollution of the air in the urban street canyons. Although this is a simple canyon configuration for investigation the flow and dispersion of the air pollutants in a street canyon, a parametric study with different street aspect ratios could be subject of further investigations.

REFERENCES

1. Assimakopoulos, V.D., Apsimon, H.M., Moussiopoulos, N., (2003), A numerical study of atmospheric pollutant dispersion in different two-dimensional street canyon configurations, *Atmospheric Environment* Vol.37(2003), pp. 4037–4049.
2. Baum, H.R., MaCaffery, B.J., (1989) Fire induced flow field—theory and experiment. Fire safety science. In: Proceedings of the second international symposium, pp. 129–48
3. Drysdale, D., (2011), An introduction to fire dynamics, Wiley & Sons
4. Kim, J.J., Baik, J.J., (2001), Urban street-canyon flows with bottom heating, *Atmospheric Environment* Vol.35(2001), pp. 3395–3404.
5. Li, X.X., Liu, C.H., Leung, D.Y.C., Lamc, K.M., (2006), Recent progress in CFD modelling of wind field and pollutant transport in street canyons, *Atmospheric Environment* Vol.40(2006), pp. 5640–5658
6. McGrattan, K.B., Baum, H.R., Rehm, R.G., (1998), Large eddy simulations of smoke movement, *Fire Safety Journal*, Vol.30(1998), pp.161–178.
7. McGrattan, K., Hostikka, S., Floyd, J., Baum, H., Mell, R.R.W., McDermott, R., (2009), Fire Dynamics Simulator (Version 5.4) Technical reference guide. National Institute of Standards and Technology. Washington.
8. Oke, T.R., (1988), Street design and urban canopy layer climate, *Energy and Buildings* Vol.11(1988), pp. 103–113.
9. Pesic, D.J., Blagojevic, M.D., Zivkovic, N.V., (2014) The model of air pollution generated by fire chemical accident in an urban street canyon, *Environmental Science and Pollution Research*, Vol.21(2014), pp. 1270–1284.
10. Smagorinsky, J., (1963) General circulation experiments with the primitive equations. I. The basic experiment, *Monthly Weather Review*, Vol.91(1963), pp.99–164.
11. Tominaga Y., Stathopoulos T., (2011) CFD modeling of pollution dispersion in a street canyon: Comparison between LES and RANS, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 99(2011), pp. 340–348.
12. Vardoulakis S, Fisher BEA, Pericleous K, Gonzalez-Flesca N (2003) Modelling air quality in street canyons: a review, *Atmospheric Environment* Vol.37(2003), pp.155-182.

UTICAJ SILA INERCIJE VETRA I UZGONA POŽARA NA ZAGAĐENJE VAZDUHA U "KANJON ULICI"

Kvalitet vazduha je postao važan ekološki problem širom sveta. Kretanje zagađivača unutar i iznad urbanih "kanjon ulica" je privuklo veliku pažnju javnosti zbog njihovih negativnih uticaja na zdravlje ljudi i životnu sredinu. Cilj ovog rada je analiza uticaja vetra i požara na zagađenje vazduha u ulici „kanjonu". Metod velikih vrtloga softvera Simulator dinamike požara je korišćen za simulaciju strujanja vetra i kretanja „oblaka" požara. Numerički rezultati pokazuju da recirkulacioni protok i vrtlozi unutar „kanjona" zavise od inercijalne sile vetra i uzgonske sile požara. Uzgonski efekat požara može imati značajan uticaj na kretanje zagađivača unutar i iznad „kanjona". Za slučajeve kada su male brzine vetra, disperzija zagađivača požara je uslovljena njegovom uzgonskom silom. Sa druge strane, kada je efekat vetra dominantniji od uzgonskog efekta, kretanje zagađivača je prvenstveno uslovljeno inercijalnom silom vetra. Ovaj složeni fenomen utiče na disperziju zagađivača unutar mnogih urbanih ulica.

Ključne reči: *„kanjon ulica", snaga vetra, uzgonska sila požara, zagađenje vazduha*