Numerical Study of Smoke Flow Control in Tunnel Fires Using Ventilation Systems

Miloš Banjac
Assistant Professor
University of Belgrade
Faculty of Mechanical Engineering

Barbara Nikolić
Teaching Assistant
Tehnikum Taurunum
High Technical School, Belgrade

With the aim of evaluating capabilities of a tunnel ventilation system to control the spread of smoke in the emergency operating mode, thereby providing conditions for safe evacuation of people from a fire-struck area, a CFD (Computational Fluid Dynamics) simulation of a fire in a double-tube tunnel was done. By the use of experimental results regarding the combustion dynamics of a passenger car, that is truck on fire and ventilation system operating modes determined according to PIRAC recommendations, a check of critical air velocity required to prevent smoke penetration into the evacuation hallways was performed, as well as the check of the optimum number and positions of ventilators in the tunnel tubes.

Keywords: *fire, smoke control, backlayering, ventilation system, CFD simulations.*

1. INTRODUCTION

In the event of a fire in an unventilated tunnel, due to difference in densities, hot combustion products rise above the fire and entrain the surrounding cold air forming a plume. The rising plume reaches the ceiling and forms two smoke streams flowing in opposite directions along the ceiling. When a longitudinal ventilation system exists, the symmetry of the rising plume and the ceiling smoke streams is broken. The rising plume bends and the length of the ceiling layer flowing against the ventilation current is reduced. The reversal of the flow of the plume is referred to as backlayering. The described situation is shown in Figure 1.

Knowing the behaviour of the reversal of the flow of the plume is closely related to the strategy for rescuing passengers in the event of a tunnel fire, since it is usually based on providing an evacuation path, that is, sufficient air space free of smoke and hot combustion products. In case of one-way tunnels with a longitudinal ventilation system, providing that "path", i.e. the control of the smoke flow usually refers to supplying the sufficient quantity of fresh air by ventilators. The flow of that fresh air should be just enough to prevent the flow of smoke in the opposite direction, i.e. to prevent the formation of a reverse stratified layer. The velocity of the air in the tunnel, which corresponds to the minimum air flow that can prevent the formation of the reverse stratified layer, is the critical velocity.

Until recently this critical velocity was determined solely by the formulae based on the use of the Froude number, adjusted by certain experimental constants. From the formulae based on this number [1-4], which represents a dimensionless ratio of buoyancy forces in the smoke caused by thermodynamics of the fire to the inertia forces caused by forced ventilation, it is only possible to determine the required, i.e. critical volumetric flow of air through the tunnel or mean critical velocity of fresh air (Tab. 1). At the same time, the phenomena caused by the operation of ventilators, a non-uniform velocity field, ejection and other air receding effects could not be included in this approach. That fact, together with the impossibility of including all the specifics related to the geometry of a tunnel, as well as numerous idiosyncrasies concerning the cases of the existence of lateral evacuation hallways, differences between the heights of hallways and main tunnels represented basic shortcomings of such an approach. In many cases, thus determined required volumetric flows, i.e. the required flows of air at a ventilator exceeded by far the actually required critical values, which led to unnecessarily high investment costs of a ventilation system.

Unlike the described conventional approach, nowadays, the numerical CFD approach, which completely overcomes the stated problems, is becoming increasingly used. This method has enabled, regardless of the complexity of a geometrical area and boundary spatial and time conditions, relatively easy and, at the same time, very precise prediction of even very complex fields of velocity, temperature and smoke, that is, concentrations of smoke formed in the air in case of fire. This particular advantage of the CFD approach has made it an almost ideal method for designing and optimization of the ventilation and smoke extraction systems.

2. PROBLEM DESCRIPTION

As part of a Mechanical Design for a ventilation system for a road traffic tunnel, consisting of two parallel tunnel tubes and for the already designed ventilation system, it was necessary to check its capabilities to extract smoke from the tunnel in case of a fire caused by a vehicle on fire.

Received: November 2008, Accepted: December 2008
Correspondence to: Dr Miloš Banjac
Faculty of Mechanical Engineering,
Kraljice Marije 16, 11120 Belgrade 35, Serbia
E-mail: mbanjac@mas.bg.ac.rs

Figure 1. Photograph of a spread of smoke in a tunnel for the 15 kW fire [4]

Table 1. Critical velocity

Formulae	Remark	Source
$U_c = \left(\frac{g \cdot H \cdot \dot{Q}}{\rho_0 \cdot T_0 \cdot c_p \cdot A} \right)^{1/3}$		Thomas [3]
$U_c = \left(\frac{g \cdot H \cdot \dot{Q}}{\rho_0 \cdot T_0 \cdot c_p \cdot A \cdot R_{i,c}} \right)^{1/3}$	$R_{i,c} = 4.5$	Danziger & Kennedy [1]
$U_c^* = 0.35 \left(\frac{\dot{Q}^*}{0.124} \right)^{1/3}$ for $\dot{Q}^* < 0.124$ $U_c^* = 0.35$ for $\dot{Q}^* > 0.124$	$U_c^* = U_c / \sqrt{g \cdot H}$ $\dot{Q}^* = \frac{\dot{Q}}{\rho_0 \cdot T_0 \cdot c_p \cdot g^{1/2} \cdot H^{1/2} \cdot A}$	Oka & Atkinson [2]
$U_c' = 0.4 \left(\frac{\dot{Q}'}{0.20} \right)^{1/3}$ for $\dot{Q}' \le 0.20$ $U_c' = 0.40$ for $\dot{Q}' > 0.20$	$U_c' = U_c / \sqrt{g \cdot \overline{H}}$ $\dot{Q}' = \frac{\dot{Q}}{\rho_0 \cdot T_0 \cdot c_p \cdot g^{1/2} \cdot \overline{H}^{5/2}}$ $\overline{H} = 4 \cdot A / P$	Wu & Barker [4]

The tunnels, each 1800 m long, are planned for one-way road traffic in two road lanes. The tunnels are interconnected with 11 regularly arranged hallways. These hallways are used for rescuing passengers in emergency situations. The ventilation system, i.e. the required volumetric flow of fresh air according to the allowed concentrations of CO and allowed content of smoke particles produced by diesel engines, was calculated in accordance with the recommendations of PIRAC (Permanent International Association of Road Congress) [5]. This calculation determined that the ventilation of each tunnel is to be carried out by 2 × 8 pairs of regularly arranged jet axial ventilators with reversed effect. The ventilators are hung on shock absorbers on the ceiling of a tunnel tube in the sector above the free traffic profile. The maximum volumetric flow of fresh air through these ventilators, in case of maximum CO and diesel engine particles pollution should be 14 m^3/s.

Checks were to be carried out in case a vehicle was set on fire (a passenger car, or truck/bus) and they were to provide answers to the following questions:

- Will, if the ventilation system works at maximum flow rate designed for the ventilation of the tunnel, smoke and combustion products penetrate the evacuation hallways?
- What will the temperature field be like in the case described above?
- If in the described case smoke still penetrates the evacuation hallways, which is the critical velocity, i.e. flow rate of air through the ventilators that will prevent this?
- How much lower are the obtained values of air velocity than the values recommended by PIRAC (Tab. 2)?

Table 2. Critical air velocity according to PIRAC [5]

No.	Cause of fire	Critical velocity, V_{kr} [m/s]
1.	passenger vehicle	1.00
2.	bus or truck	2.7

In making the calculations, the direction of ventilation and control of the smoke flow had to be taken into account. In case of an accident at the front of the tunnel, smoke control should be carried out in the opposite direction from the one in which the vehicle is moving, while if a fire occurs behind the first half of the tunnel, it should be carried out in the direction in which the vehicle is moving.

3. NUMERICAL MODEL

For the calculation of flow and temperature fields, i.e. smoke concentration fields formed in case of fire within the tunnel space, the CFD software package PHOENICS 3.4 was used. In accordance with the assumed physical situation, because of certain identical segments of the tunnel and the high length of the entire tunnel, a 3-dimensional space model was generated with this software for three geometrically different tunnel segments. The length of each segment was 250 m, while the basic geometrical difference referred to the distances between the ventilators and evacuation hallways, which were 30, 60 and 90 m. Apart from the ventilators, cars were also generated in the virtual space of the tunnel, one of which (a truck), 5 m away from an evacuation hallway, was defined as the source of heat, i.e. “the source” of combustion products – smoke. The amount of heat, that is, the amount of smoke released per time unit from the fire-struck car (truck) was defined according to the recommendations of PIRAC (Tab. 3). The whole virtual garage space was divided into 50 × 120 × 40 = 240,000 control volumes (Fig. 2).

4. MATHEMATICAL MODEL

For the calculation of flow and temperature fields of the air formed within the tunnel, a two-equation $k-\varepsilon$ turbulent model was used [6]. This universal turbulent model was chosen due to its confirmed reliability in predicting the flow fields during flows with the Mach number considerably lower than 1 [7]. Apart from three, i.e. four basic balance equations describing non-stationary incompressible fluid flow for each previously defined control volume:

- continuity equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho U_i)=0\,, \tag{1}$$

- modelled Reynolds equation

$$\frac{\partial}{\partial x_j}(\rho U_i U_j)=$$

$$=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[(\mu_\mathrm{f}+\mu_t)S_{ij}-\frac{2}{3}k\delta_{ij}\right]+\rho F_i\,, \tag{2}$$

- and energy balance equation

$$\frac{\partial}{\partial t}(\rho H)+\frac{\partial}{\partial x_j}(\rho U_j H)=$$

$$U_i\frac{\partial P}{\partial x_i}+2\mu_\mathrm{t}S_{ij}S_{ij}+\frac{\partial}{\partial x_i}\left[\left(a_\mathrm{f}+\frac{\mu_t}{Pr_h}\right)\frac{\partial H}{\partial x_i}\right], \tag{3}$$

with additional

- balance equation of smoke “concentration” (mass fraction of smoke in the air)

$$\frac{\partial}{\partial t}(\rho\bar{C})+\frac{\partial}{\partial x_j}(\rho U_j\bar{C})=$$

$$=U_i\frac{\partial P}{\partial x_i}+2\mu_\mathrm{t}S_{ij}S_{ij}+\frac{\partial}{\partial x_i}\left[\left(D_C+\frac{\mu_t}{Sh_t}\right)\frac{\partial\bar{C}}{\partial x_i}\right], \tag{4}$$

this turbulence model was defined with:

- transport equation for the turbulence kinetic energy

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}(\rho U_i k)=$$

$$=\mathscr{P}_k+\frac{\partial}{\partial x_i}\left[\left(\mu_\mathrm{f}+\frac{\mu_\mathrm{t}}{Pr_k}\right)\frac{\partial k}{\partial x_i}\right]-\rho\frac{\mu_\mathrm{t}g_i}{\sigma_h}\frac{\partial\rho}{\partial x_i}-\rho\varepsilon\,, \tag{5}$$

- and transport equation for the dissipation rate:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_i}(\rho U_i\varepsilon)=C_{\varepsilon 1}\frac{\varepsilon}{k}\mathscr{P}_k-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}+$$

$$+C_{\varepsilon 3}\rho\frac{\mu_\mathrm{t}g_i}{\sigma_h}\frac{\partial\rho}{\partial x_i}+\frac{\partial}{\partial x_i}\left[\left(\mu_\mathrm{f}+\frac{\mu_\mathrm{t}}{Pr_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]. \tag{6}$$

In the aforementioned equations, according to a standard procedure, S_{ij} was defined as the main strain-rate tensor:

$$S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right), \tag{7}$$

Table 3. Empirical constants of $k-\varepsilon$ model of turbulent stresses for low Reynolds turbulent numbers

Pr_k	Pr_ε	$C_D C_\mu$	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	$C_{\varepsilon 3}$	κ	Pr_h	Sh_t
1.0	1.314	0.09	1.44	1.92	1.0	0.41	0.41	0.81

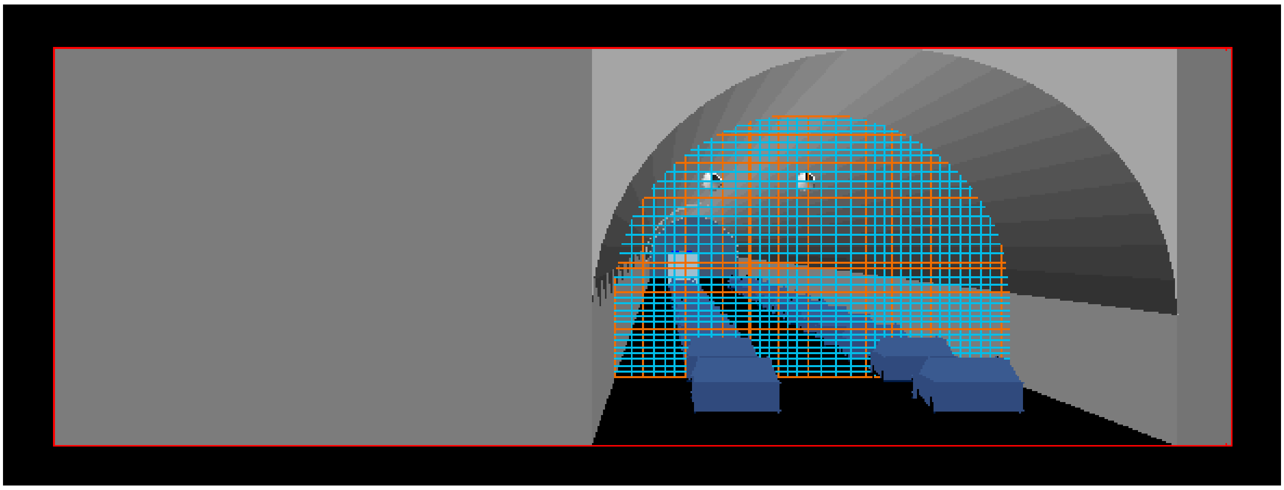

Figure 2. Mesh of control volumes in virtual tunnel space

and $\mathscr{P}_k$, the volumetric production rate of k by shear forces is:

$$\mathscr{P}_k = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} . \quad (8)$$

Modelling of the Reynolds stresses tensor was based on the Boussinesq hypothesis:

$$\tau_{ij} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} = \mu_t S_{ij} - \frac{2}{3} k \delta_{ij} , \quad (9)$$

where the eddy viscosity – μ_t was defined by the equation

$$\mu_t = (C_D C_\mu) \rho \frac{k^2}{\varepsilon} . \quad (10)$$

Since the value of molecular diffusivity of smoke into the air was negligible compared to the turbulent (molar) diffusivity, it was neglected during the calculation.

The values of the empirical constants of this model, as well as the values of the Prandtl (enthalpy), i.e. Schmidt turbulent number, are given in Table 3.

Apart from the k – ε turbulent model, and as a standard procedure for two-equation turbulent models, the Reynolds enthalpy flux, i.e. the Reynolds flux of smoke concentration was modelled in accordance with the principles of the simple gradient-diffusion hypothesis:

$$-\rho \overline{h u_i} = \frac{\mu_t}{Pr_h} \frac{\partial H}{\partial x_i} , \quad (11)$$

i.e.

$$-\rho \overline{c u_i} = \frac{\mu_t}{Pr_h} \frac{\partial \overline{C}}{\partial x_i} . \quad (12)$$

Since both fluids – air and smoke – can be considered as ideal gases, i.e. their mixture, regardless of fractions of particular components, can be treated as ideal gas, for determining flow and temperature fields and smoke concentration fields, the so-called scalar variable marking method was used.

Regarding (molecular) viscosity, it was assumed that there was square thermodynamic temperature dependency:

$$\mu_f = -4{,}9468 \cdot 10^{-6} + 4.5839 \cdot 10^{-8} T + 8.0974 \cdot 10^{-11} T^2 ,$$

whereas, for a specific thermal capacity of air at constant pressure, i.e. its thermal conductivity, it was assumed that they had constant values, c_p = 1004 J/(kg K), $\lambda_f = 2.63 \cdot 10^{-2}$ W/(m K).

5. RESULTS OF NUMERICAL CALCULATIONS

By reviewing the results of the simulations, Figs. 3 and 4, it was concluded that regardless of the mutual distances of an axial ventilator from an evacuation opening, relatively similar velocity, i.e. smoke concentration fields are formed. It is interesting to note that a somewhat greater difference is created if the distance of an axial ventilator from an evacuation opening is 30 m.

If the ventilation system operates at maximum flow rate designed for the ventilation of the tunnel, smoke and combustion products will not penetrate the evacuation hallways if the cause of fire is a passenger vehicle. That happens primarily because axial ventilators, placed close to the ceiling provide a non-uniform velocity field which reaches its maximum values precisely in the zone of the formation of the reverse stratified layer. Thus, maximum air velocities are reached in the zone in which that is the most needed. Furthermore, two effects have beneficial impact on preventing the said penetration of smoke: the so called ejection and the effect of air “receding”, which cause the volumetric air flow through the tunnel to be higher than the air flow through the ventilation pair.

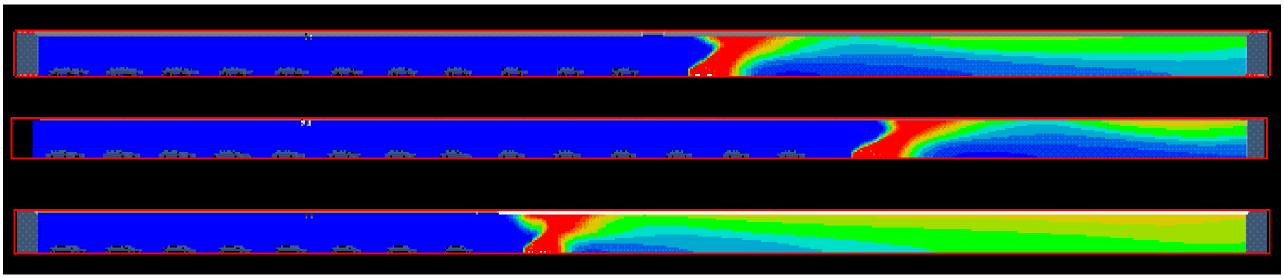

Figure 3. Smoke concentration field on the plane *x* = 3 m from the axis of the 250 m long part of the tunnel, in case of a fire caused by a passenger vehicle on fire, at maximum air flow rate determined for the ventilation of the tunnel, for three distances of 30, 60 and 90 m of the ventilators from the evacuation openings

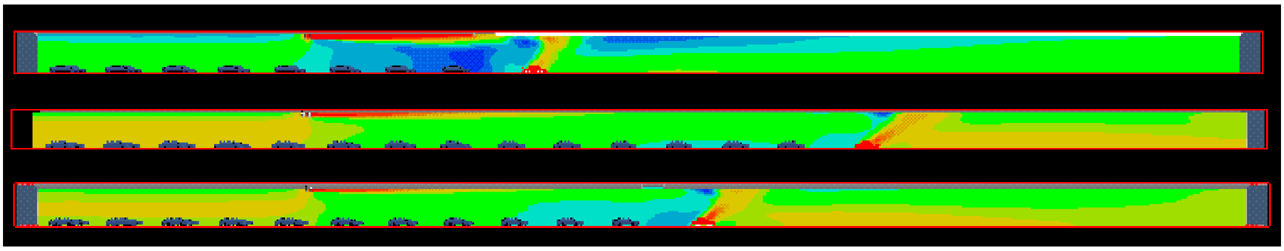

Figure 4. Air velocity field on the plane *x* = 3 m from the axis of the tunnel, in case of a fire caused by a passenger vehicle on fire, at maximum air flow rate determined for the ventilation of the tunnel, for three distances of 30, 60 and 90 m of the ventilators from the evacuation openings

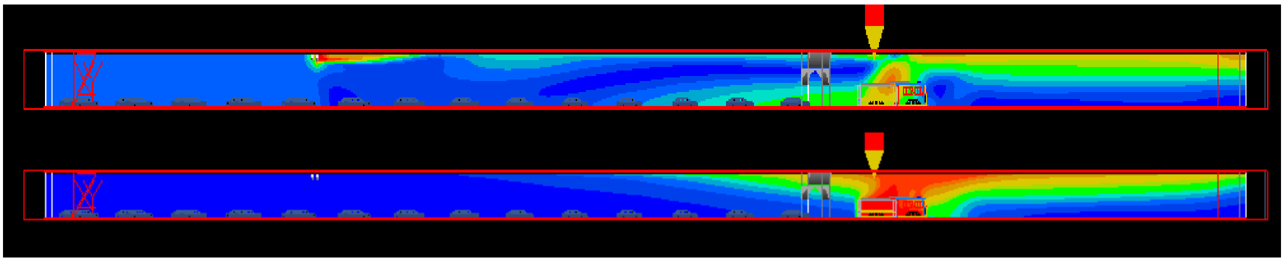

Figure 5. Air velocity and smoke concentration field on the plane *x* = 3 m from the axis of the tunnel, in case of a fire caused by a bus on fire, at maximum air flow rate determined for the ventilation of the tunnel, for the distance of 90 m between the ventilator and evacuation openings

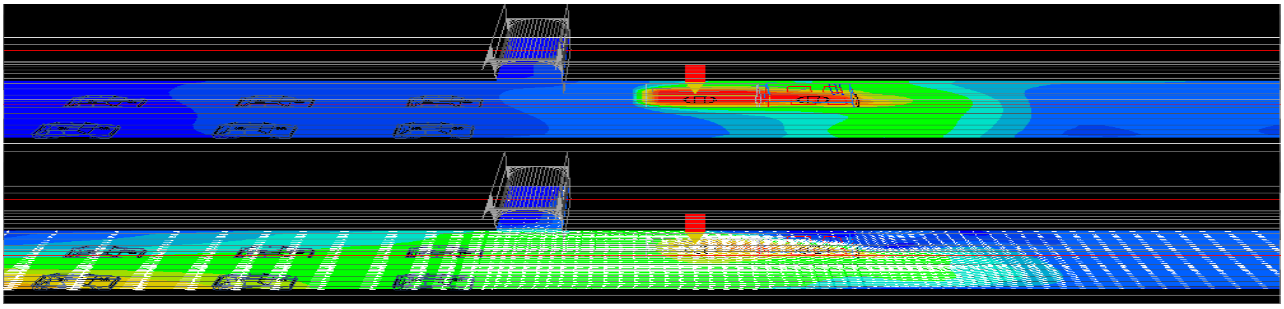

Figure 6. Air velocity and smoke concentration field at the height of *z* = 2 m, in case of a fire caused by a bus on fire, at maximum air flow rate determined for the ventilation of the tunnel, for the distance of 90 m between the ventilators and evacuation openings

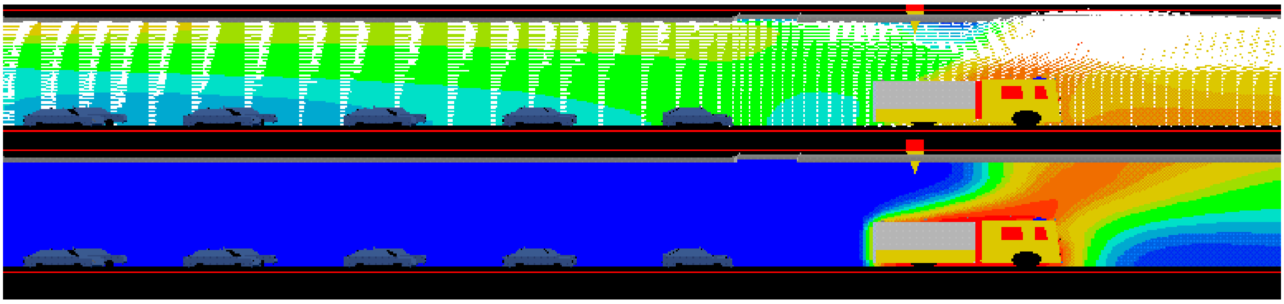

Figure 7. Smoke concentration field on the plane *x* = 3 m from the axis of the tunnel, in case of a fire caused by a bus on fire, at 1.6 times higher air flow rate than the maximum determined for the ventilation of the tunnel, for the distance of 90 m between the ventilators and evacuation openings

If the ventilation system operates at maximum flow rate designed for the ventilation of the tunnel, smoke and combustion products will penetrate the evacuation hallways if the cause of fire is a bus or truck (Figs. 5 and 6). In that case, in order to prevent the penetration, it is necessary to increase the velocity of air flow through ventilators by 1.6 times, at a minimum (Fig. 7).

By reviewing the obtained results, firstly by visual check and then the check of numerical values, it was concluded that critical velocities obtained by the numerical simulation were by about 20 % lower than the ones recommended by PIRAC. It should be taken into account that the numerically obtained value represents mean air velocity in a horizontal cross-section of the tunnel.

6. CONCLUSION

For a road traffic tunnel consisting of two parallel tunnel tubes and for the already designed ventilation system, it was necessary to check its capabilities to extract smoke from the tunnel. Based on the performed numerical calculations, it was concluded that:

- if the cause of fire is a passenger vehicle, smoke and combustion products will not penetrate the evacuation hallways when the ventilation system operates only at the maximum flow rate designed for the ventilation of the tunnel,
- if the cause of fire is a bus or truck, smoke will not penetrate the evacuation hallways when the ventilation system operates at a 1.6 times higher rate than the one designed solely for the ventilation of the tunnel and
- critical velocities obtained by numerical simulations are by approximately 20 % lower than the ones recommended by PIRAC, which can be considered an appropriate safety measure.

REFERENCES

[1] Danziger, N.H. and Kennedy, W.D.: Longitudinal ventilation analysis for the Glenwood canyon tunnels, in: *Proceedings of the Fourth International Symposium Aerodynamics and Ventilation of Vehicle Tunnels*, March 1982, York, England, pp. 169-186.

[2] Oka, Y. and Atkinson, G.T.: Control of smoke flow in tunnel fires, Fire Safety Journal, Vol. 25, No. 4, pp. 305-322, 1995.

[3] Thomas, P.H.: *The Movement of Smoke in Horizontal Passages Against an Air Flow*, Fire Research Note No. 723, Fire Research Station, Watford, UK, 1968.

[4] Wu, Y. and Bakar, M.Z.A.: Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity, Fire Safety Journal, Vol. 35, No. 4, pp. 363-390, 2000.

[5] *Fire and Smoke Control in Road Tunnels*, PIRAC Publications, Paris, 1999.

[6] Launder, B.E. and Spalding, D.B.: The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, Vol. 3, No. 2, pp. 269-289, 1974.

[7] Wilcox, D.C.: *Turbulence Modelling for CFD*, DCW Industries, La Cañada, California, USA 1993.

НУМЕРИЧКА СИМУЛАЦИЈА КОНТРОЛЕ КРЕТАЊА ДИМА ПРИ ПОЖАРУ У ТУНЕЛУ

Милош Бањац, Барбара Николић

Са циљем процењивања могућности једног тунелског вентилационог система, да у инцидентном радном режиму контролише ширење дима и да тако обезбеди услове за безбедну евакуацију људи из ватром захваћеног простора, извршена је *CFD* (*Computational Fluid Dynamics*) симулација пожара у једном двоцевном тунелу. Коришћењем експерименталних резултата, о динамици сагоревања запаљеног аутомобила и према препорукама *PIRAC*-а одређених радних режима вентилатора, извршена је провера критичне брзина струјања ваздуха којом ће се спречити продор дима у евакуационе ходнике, те извршена провера потребног броја и распореда вентилатора у тунелским цевима.