COMPUTATIONAL STUDY OF SMOKE FLOW CONTROL IN GARAGE FIRES AND OPTIMISATION OF THE VENTILATION SYSTEM

by

Miloš J. BANJAC and Barbara M. NIKOLIĆ

Original scientific paper
UDC: 692.842:532.57
BIBLID: 0354-9836, *13* (2009), 1, 69-78
DOI: 10.2298/TSCI0901069B

With the aim of evaluating capabilities of a 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, computational fluid dynamics simulation of a fire in a semi-bedded garage was conducted. Using the experimental results of combustion dynamics of a passenger car on fire, optimal positions of ventilation openings were determined. According to recommendations by DIN EN 12101 standard, the operating modes of a ventilation system were verified and optimal start time of the smoke extraction system was defined.

Key words: *fire, smoke control, ventilation system, computational fluid dynamics, smoke extraction*

Introduction

A proper design and construction of a smoke extraction system, either during construction of a new building or reconstruction of old ones, represents an important measure that has to be taken to reduce hazard that people may be exposed to in case of fire. An appropriate arrangement of smoke extraction openings, as well as proper selection of an operating mode of a ventilation system, enable prevention of further fire spreading by decreasing fresh air supply, *i. e*. provide conditions for safe evacuation of people from fire-struck area by efficient smoke extraction. Therefore, for proper design of these systems, apart from knowing combustion dynamics of particular materials, the amount of heat and smoke produced during their combustion, it is necessary to have a reliable methodology for predicting what would, in different operating modes of a smoke extraction system, happen with fire, *i. e.* how the air temperature and velocity fields will change in a fire-struck area and in which direction smoke will spread, *i. e.* remain.

"Conventional" experimental methods, as methodologies for solving the above-mentioned questions, are very rarely applied due to their high price and long period for calculation, especially in the phase of a building design. Other semi-empirical methods for predicting the aforementioned fields that rely on the use of experimental data on smoke spreading in seemingly immovable air or in the presence of air jets, *i. e.* on the behaviour of air near openings, are very unreliable. Among others, significant disadvantage of this approach is scarcity or complete lack of information about physical properties of substances in parts of area outside of the zones near extraction openings. Also, the use of such expressions and diagrams in real conditions usually leads to considerable differences between the estimated and achieved fields of the aforementioned properties. A similar, if not worse situation is with nowadays conventional methodologies based on empirical data on the necessary number of air replacements and other similar very rough calculations. Therefore, solving the described problem before the appearance of computational fluid dynamics (CFD) and its use in the design of ventilation and smoke extraction systems, as well as the combustion process and fire spreading, was a very difficult task.

Unlike the conventional approach, the numerical CFD-approach enabled, regardless of the complexity of 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 concentrations formed in the air in case of fire. This approach, based on the space discretization and mesh generation, thus forming a very large number of finitely small control volumes (CV) and setting and simultaneous solving of balance equations defined for each CV, made it possible to gather information on fluid velocity, pressure, temperature, density, turbulence level, concentration of particular substances, *etc*. for a significantly large number of points in volume. This particular advantage of the CFD-approach has made it an almost ideal method for designing ventilation, *i. e.* smoke extraction systems [1, 2].

Problem description

As part of a Mechanical Design (MD) of a heating and ventilation system of a particular block of flats, it was necessary to design the ventilating system, *i. e*. smoke extraction system of a semi-bedded garage beneath the same building. The rectangular garage, with inner dimensions of 15.8 × 15.8 × 3.2 m, had all four sides embedded into the ground up to 50% of its height. In the garage, there were 4 cylindrical concrete supporting pillars with the outer diameter of 0.7 m. According to the Civil Engineering Design, there is a movable door on the approach side, planned for entrance and exit of vehicles. The door dimensions are 3.2 × 2.5 m. The garage is supplied with three windows, with dimensions 1.0 × 0.6 m on one side and one opposite wall, at the height of 2.2 m measured from the garage floor. The door for entrance and exit of people, *i. e*. for their evacuation, stands on the opposite wall of the car door. The garage capacity is 12 passenger vehicles in two rows.

In normal operating conditions, garage space is ventilated by natural ventilation or by forced ventilation, in case of exceeding particular exhaust gas concentration in the air. In case of forced ventilation, efficiency of two different constructive solutions were analysed. The first technical solution included the ventilation channel routed to the centre of the garage, where the suction opening was placed at the bottom channel side and it was used for extracting the air out of the garage (optionally on lateral sides). The second technical solution of the design avoided routing channels inside the garage, and one of the existing window openings had been transformed into a suction opening of the ventilation system.

According to the MD, the use of a ventilation system was envisiged for the smoke extraction in the case of fire start in the garage space *i. e.* fire on one of the parked cars. Furthermore, in case it works as a ventilation system, in compliance with the existing legal regulations in this area [3], a fan should operate in a way that provides an air change rate of 6 changes per hour; in case it works as a smoke extraction system, the air change rate should be 12 changes per hour.

According to the terms of reference of the MD, in the part that elaborates on the problem of a smoke extraction system, the following items should be analyzed:

– determination of the optimal position of an extraction ventilation opening, *i. e.* extraction opening for the smoke extraction system in which, in the designed operating mode of 12 air changes per hour, smoke produced by the car on fire will not fill the whole garage space, *i. e*.:

  - the smoke concentration should be reduced to minimum in the selected reference point - near the door for entrance/exit of people, and
  - temperature in the selected reference point should be as low as possible,
- verification of the capability of operating modes of a smoke extraction system in terms of the ability to extract smoke out of the garage space for minimal time. In the absence of the quantitative values of produced smoke quantity, it was considered that the smoke concentration on boundary surfaces of the car on fire was equal to 100% ($\overline{C}$ = 1.00), and in "fresh" air 0% ($\overline{C}$ = 0.00). According to the above-mentioned, it was defined that a smoke extraction system should provide:
  - smoke concentration near the door for entrance/exit of people lower than 30%, and
  - temperature near the door for entrance/exit of people not higher than 40 °C.

Numerical calculation of velocity, temperature and smoke field in the garage

Numerical model

The first step for the calculation of flow and temperature fields, *i. e.* smoke concentration fields formed in case of fire within the garage space, was generating a 3-D garage space model by commercial CFD software package PHOENICS 3.4. According to the assumed physical situation, the facility layout was generated – 12 cars with dimensions 3.9   1.4   1.4 m, 4 concrete pillars and a ventilation channel, *i. e.* position of appropriate smoke extraction openings, windows and doors.

Inner, virtual garage space was divided into CVs in such a way that all boundary surfaces of CVs were aligned with the contours of appropriate solid bodies and barriers within the space. Thus generated mesh was additionally balanced, *i. e.* made thicker, by adding control volumes in zones near windows and openings. Total number of formed control volumes was 39   43   26 = 43.602 (fig. 1).

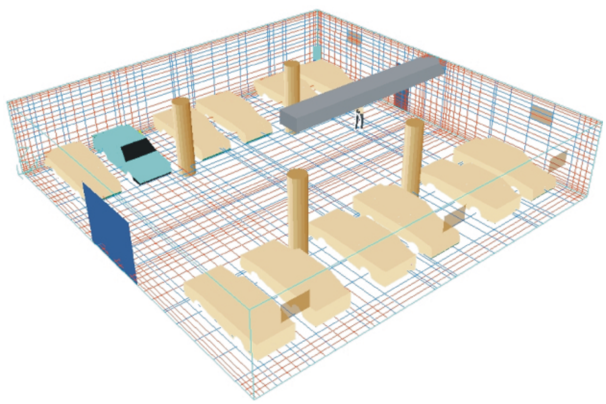

Figure 1. Mesh of control volumes in virtual garage space (color image see on our web site)

Mathematical model

For the calculation of flow and temperature fields of air formed within the garage, a two-equation *k*-ε turbulent model was used [4]. 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 [5]. Apart from three, *i. e.* four basic balance equations describing non-stationary incompressible fluid flow for each previously defined control volume:

– continuity equation

$$\frac{\partial \rho}{\partial t} \quad \frac{\partial}{\partial x_i}(\rho U_i) \quad 0 \tag{1}$$

– modelled Reynolds equation

$$\frac{\partial}{\partial t}(\rho U_i) \quad \frac{\partial}{\partial x_j}(\rho U_i U_j) \qquad \frac{\partial P}{\partial x_i} \quad \frac{\partial}{\partial x_j} \quad (\mu_{\mathrm{f}} \quad \mu_{\mathrm{t}})S_{ij} \quad \frac{2}{3}k\delta_{ij} \qquad \rho F_i \tag{2}$$

– and energy balance equation

$$\frac{\partial}{\partial t}(\rho H) \quad \frac{\partial}{\partial x_j}(\rho U_j H) \quad U_i \frac{\partial P}{\partial x_i} \quad 2\mu_t S_{ij} S_{ij} \quad \frac{\partial}{\partial x_i} \quad a_{\mathrm{f}} \quad \frac{\mu_{\mathrm{t}}}{\mathrm{Pr_h}} \quad \frac{\partial H}{\partial x_i} \tag{3}$$

with additional

– balance equation of smoke "concentration" (mass fraction of smoke in the air)

$$\frac{\partial}{\partial t}(\rho \overline{C}) \quad \frac{\partial}{\partial x_j}(\rho U_j \overline{C}) \quad U_i \frac{\partial P}{\partial x_i} \quad 2\mu_{\mathrm{t}} S_{ij} S_{ij} \quad \frac{\partial}{\partial x_i} \quad D_{\mathrm{C}} \quad \frac{\mu_t}{\mathrm{Sh_t}} \quad \frac{\partial \overline{C}}{\partial x_i} \tag{4}$$

this turbulence model was defined with

– transport equation for the turbulence kinetic energy

$$\frac{\partial(\rho k)}{\partial t} \quad \frac{\partial}{\partial x_i}(\rho U_i k) \quad \mathscr{P}_{\mathrm{k}} \quad \frac{\partial}{\partial x_i} \quad \mu_{\mathrm{f}} \quad \frac{\mu_{\mathrm{t}}}{\mathrm{Pr_k}} \quad \frac{\partial k}{\partial x_i} \quad \rho \frac{\mu_{\mathrm{t}} g_i}{\mathrm{Pr_h}} \frac{\partial \rho}{\partial x_i} \quad \rho\varepsilon \tag{5}$$

– and transport equation for the dissipation rate

$$\frac{\partial(\rho\varepsilon)}{\partial t} \quad \frac{\partial}{\partial x_i}(\rho U_i \varepsilon) \quad C_{\varepsilon 1} \frac{\varepsilon}{k} \mathscr{P}_{\mathrm{k}} \quad C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad C_{\varepsilon 3} \rho \frac{\mu_{\mathrm{t}} g_i}{\mathrm{Pr_h}} \frac{\partial \rho}{\partial x_i} \quad \frac{\partial}{\partial x_i} \quad \mu_{\mathrm{f}} \quad \frac{\mu_{\mathrm{t}}}{\mathrm{Pr}_{\varepsilon}} \quad \frac{\partial \varepsilon}{\partial x_i} \tag{6}$$

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

$$S_{ij} \quad \frac{1}{2} \quad \frac{\partial U_i}{\partial x_j} \quad \frac{\partial U_j}{\partial x_i} \tag{7}$$

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

$$\mathscr{P}_{\mathrm{k}} \quad \mu_{\mathrm{t}} \quad \frac{\partial U_i}{\partial x_j} \quad \frac{\partial U_j}{\partial x_i} \quad \frac{\partial U_i}{\partial x_j} \tag{8}$$

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

$$\tau_{ij} \quad \mu_{\mathrm{t}} \quad \frac{\partial U_i}{\partial x_j} \quad \frac{\partial U_j}{\partial x_i} \quad \frac{2}{3}k\delta_{ij} \quad \mu_{\mathrm{t}} S_{ij} \quad \frac{2}{3}k\delta_{ij} \tag{9}$$

where the eddy viscosity – μ_{t} was defined by the equation

$$\mu_{\mathrm{t}} \quad (C_{\mathrm{D}} C_{\mu})\rho \frac{k^2}{\varepsilon} \tag{10}$$

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 tab. 1.

Table 1. Empirical constants of k-ε model

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

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:

$$\rho\overline{hu_i} \quad \frac{\mu_t}{\mathrm{Pr_h}}\frac{\partial H}{\partial x_i} \tag{11}$$

i. e.

$$\rho\overline{cu_i} \quad \frac{\mu_\mathrm{t}}{\mathrm{Sh}_\mathrm{t}}\frac{\partial \overline{C}}{\partial x_i} \tag{12}$$

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:

$$\mu_\mathrm{f} = -4.9468\cdot10^{-6} + 4.5839\cdot10^{-8}T + 8.0974\cdot10^{-11}T^2 \ [\mathrm{m^2s^{-1}}]$$

whereas, for a specific thermal capacity of air at constant pressure, *i. e.* its thermal conductivity, it was assumed that they had constant values, $c_\mathrm{p} = 1004$ Jkg^{-1}K^{-1}, $\lambda_\mathrm{f} = 2.63\cdot10^{-2}$ Wm^{-1}K^{-1}.

Due to relatively high combustion product temperatures generated during car burning, in order to gain higher precision, heat transfer by radiation was also covered by numerical calculation. The so-called immersed-solid (Immersol) radiation model [6, 7] was used. Within the space between solids, the distribution of radiosity, *i. e.* σT_3^4, can be represented as the following equation:

$$\frac{\partial}{\partial x_i} \quad \Gamma_\mathrm{rad}\frac{\partial T_3}{\partial x_i} \qquad (a \quad s)(E_{12} \quad \sigma T_3^4) \tag{13}$$

where: E_{12} stands for the phase-surface-average of σT^4, $a = 0.1$ stands for the absorptivity of the fluid medium, and $s = 0.1$ stands for the scattering coefficient of that medium. Γ_rad is defined as the reciprocal of $\Gamma_\mathrm{rad} = 0.75[(a + s + 1/w_\mathrm{gap})]$, and w_gap stands for the distance between adjacent walls.

Boundary and initial conditions

According to the real physical situation, *i. e.* designed layout of the garage and smoke extraction system, it was necessary to specify boundary spatial and initial conditions for velocity, temperature (energy), and smoke concentration fields.

– *Boundary conditions for contact between air and solid surfaces*

The "wall" function model was used for specifying boundary conditions near the solid surfaces within the garage, related to the velocity field. Since the used turbulent model belongs to a class of the high Reynolds turbulent model, for determining values of variables, *i. e.* their flows next to the plate, wall functions of the logarithmic area of the turbulent boundary layer were used [5, 8].

At the same time, the following was assumed as boundary conditions for the temperature field:

- garage walls were adiabatic surfaces with thermal emissivity of 0.9,
- cars which were not on fire were made of steel sheet metal with thermal emissivity of 0.9, and

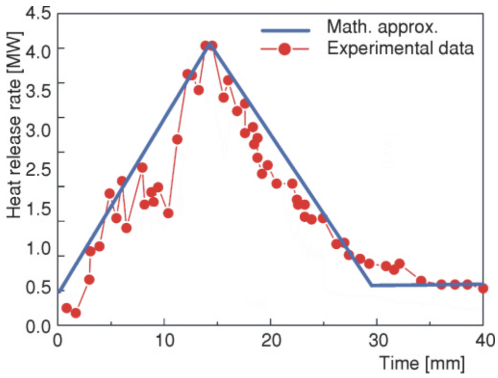

Figure 2. Heat release rate of one car

– contour of the automobile on fire was treated as a thermal source contour, *i. e.* smoke source.

Thermal emission dynamics of this thermal source had a shape of a "saw" function. This shape of the function was generated by the approximation of experimental data [9] in case of one automobile burning in an underground garage (fig. 2).

Smoke production dynamics was defined by assuming proportionality between smoke quantity produced during combustion and amount of heat released during that process [10]. During this, in the absence of quantitative values of produced smoke quantity, it was considered that the smoke concentration on boundary surfaces on the automobile on fire equalled 100% ($C = 1.00$), and in "fresh" air 0% ($C = 0.00$).

– *Boundary conditions on window panes and doors*

Since the variable values in the domain outside sections (windows) are completely unknown, except in the case of garage smoke extraction openings, the so-called condition of "constant" pressure [11] as a boundary condition was used. This condition consists of specifying zero derivatives in the direction normal to the outgoing plane, *i. e.* specifying second boundary conditions for all values of dependent variables, except for velocities normal to the outgoing plane, *i. e.* $\Phi / x_2 \,{}_{out} = 0$ and $\partial\Phi / x_3 \,{}_{out} = 0$ ($\Phi = k, \varepsilon, H$, and $\overline{C}$), and specifying a pressure value equal to the ambient pressure $P_{out} = P_{amb}$. Values of missing velocities were defined indirectly, by using pressure values.

The door for entrance of vehicles was shut, as well as the door for the evacuation of people. Thermal emissivity of these surfaces was specified to be 0.9.

The assumed outside air temperature was 17 °C.

– *Boundary conditions at smoke extraction openings*

Velocity field at the smoke extraction opening was specified by using appropriate values of the volumetric air flow.

– *Initial conditions*

The assumed temperature of the air and all objects within the garage space equalled at the initial moment. Smoke concentration in the air of the whole garage space equalled zero, and the air was assumed to be absolutely still.

The analysis was done on only one operating mode with 12 air replacements per hour and it was specified that the smoke extraction systems would be activated 1 minute and 2 minutes after the outbreak of fire.

Optimization of the position an extraction opening in the garage space

Optimal position of an extraction opening of a smoke extraction system, *i. e.* ventilation system was selected by performing numerical simulations. All simulations were performed for unsteady – transient conditions. According to the experimental data on the duration of fire, time domain for each simulation was 40 minutes. Good convergence of the solution was achieved for the time step of 5 seconds, with 200 iterations within each time step.

Numerical simulations were performed both for the case of routing the ventilation channel to the centre of the garage and for the case of smoke extraction through one of the window openings. Several simulations were made for the case of fire spreading from one car to another, but since such an event is unlikely to occur, in all other simulations, we considered the case where only one car was on fire. Also, the position of the car on fire in several initial simulations was changed. Afterwards, it was decided to use the second car from the entrance and exit door as the representative position of the vehicle (the worst scenario).

By viewing all achieved results, it was determined that the temperature near the area of the car on fire was between 200 and 370 °C, which corresponded to experimentally determined values [12]. According to the criteria set by the terms of reference (ToR) of the project, it was

Figure 3. Garage air velocity field in plane *z* = 3 m, *i. e.* on the surface with smoke concentration of 30% and air temperature of 40 °C, 5 minutes after the fire start, in case of two locations, (a) and (b), for smoke extraction (color image see on our web site)

noted that the location of a smoke extraction opening should be as close as possible to the place of fire, *i. e.* on the wall "behind" the car on fire. Since it was not possible to fulfil this condition – only one smoke extraction location was requested by the project ToR, it was chosen, as an optimal solution, that the smoke extraction location should be placed on the ceiling of the garage centre, *i. e.* that the ventilation channel should be placed into the garage space. By this technical solution, the smoke extraction opening must not be placed on the bottom, but exclusively on the lateral channel sides. This is the only way in which it will be possible to extract smoke accumulated in the ceiling area.

Since smoke spreads quickly within the garage space (fig. 3) in terms of smoke extraction, it was concluded that it would be more suitable if the smoke extraction system was activated at least within one minute following the fire outbreak.

Reviewing the achieved results, visually at first, and then by the verification of numerical values, it was concluded that, under the foreseen values of the volumetric flow of 12 air changes per hour, it was only possible to achieve specified parameters if the smoke extraction channel was placed in the central zone of the garage space (fig. 4). In that manner, in the area around the entrance and exit door for people, smoke concentration was not over 30% and the air temperature was not over 40 °C.

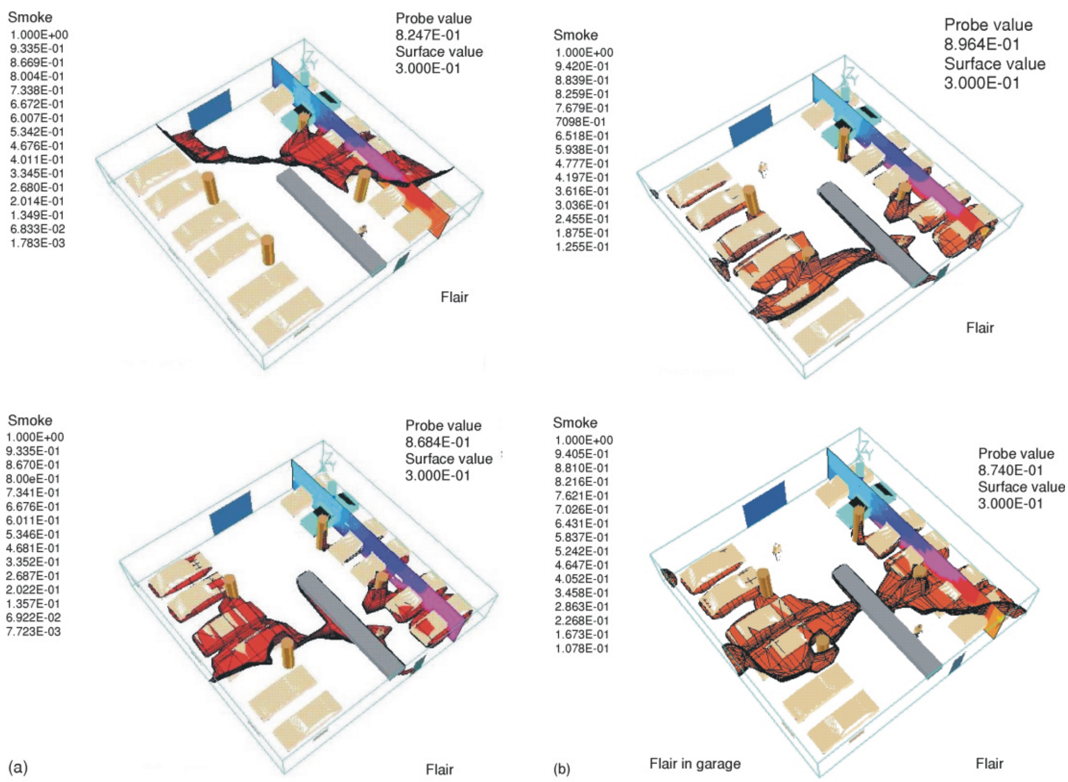

Figure 4. Smoke spreading – surfaces with smoke concentration of 30%
After (a) 30 seconds; (b) 1 minute; (c) 2 minutes; and (d) 15 minutes following the outbreak of fire
(color image see on our web site)

Conclusions

Analysing and reviewing the results of the numerical simulation of a fire spread in a semi-bedded garage, with dimensions 15.8 15.8 3.2 m, caused by the fire outbreak on one car, the following could be concluded:

- smoke extraction system should be as close as possible to the place of fire, *i. e*. on the wall "behind" the automobile on fire or, if there is only one location for smoke extraction, on the ceiling in the central zone of the garage; in that case, smoke extraction openings must not be placed on the bottom, but exclusively on the lateral channel sides, because only in this way will it be possible to extract smoke accumulated in the ceiling area.
- the operating mode of a smoke extraction system with the rate of 12 air changes per hour specified by DIN EN 12101 (Smoke and heat control systems) standard will make sure that, in the space around the entrance and exit door for people, smoke concentration will not exceed 30%, *i. e*. that the air temperature will not exceed 40 °C, and only will be provided if smoke extraction system is located in the central part of the garage space and smoke extraction systems are activated within 1 minute following the fire outbreak.

Nomenclature

a – absorption coefficient of the air, [m^{-1}]
a_f – thermal diffusivity, [m^{2}s^{-1}]
C – mass fraction of smoke in the air, [–]
D_C – mass diffusivity, [m^{2}s^{-1}]
F_i – body force per unit mass, [Nkg^{-1}]
g_i – gravity force per unit mass, [Nkg^{-1}]
H – mean enthalpy per unit mass, [Jkg^{-1}]
h – fluctuating enthalpy per unit mass, [Jkg^{-1}]
k – kinetic energy of turbulent fluctuation per unit mass, [m^{2}s^{-2}]
P – mean static pressure, [Pa]
Pr_h – Prandtl number, [–]
Pr_k – Prandtl number for k, [–]
S_{ij} – mean strain-rate tensor, [s^{-1}]
Sh_t – Schmidt turbulent number, [–]
s – scattering coefficient of air, [m^{-1}]
T – temperature, [K]
t – time, [s]
U_i – mean velocity vector in tensor notation, [ms^{-1}]
u_i – fluctuating velocity vector in tensor notation, [ms^{-1}]
w_{gap} – distance between adjacent walls, [m]
x_i – position vector in tensor notation, [m]

Greek letters

δ_{ij} – Kronecker delta, [–]
ε – dissipation of turbulent kinetic energy, [m^{2}s^{-3}]
μ_f – molecular viscosity, [Pa·s]
μ_t – eddy viscosity, [Pa·s]
ρ – mass density, [kgm^{-3}]
σ – Stefan-Boltzmann constant, (= $5.67\cdot10^{-8}$), [Wm^{-2}K^{-1}]
τ_{ij} – Reynolds stresses tensor, [Nm^{-2}]

References

[1] Vidmar, P., Petelin, S., Methodology of Using CFD-Based Risk Assessment in Road Tunnels, *Thermal Science, 11* (2007), 2, pp. 223-250
[2] Stevanović, Ž., Marković, Z., Turanjanin, V., Numerical Simulation of Fire Spread in Terminal 2 of Belgrade Airport, *Thermal Science, 11* (2007), 2, pp. 251-258
[3] Recknagel, H., *et al*., Heating and Air Conditioning: Including Hot Water and Cooling Technique (in Serbian), Interklima, Vrnjačka Banja, Serbia, 2004
[4] Launder, B. E., Spalding, D. B., The Numerical Computation of Turbulent Flows, *Computer Methods in Applied Mechanics and Engineering, 3* (1974), 2, pp. 269-275
[5] Wilcox, D. C., Turbulence Modelling for CFD, DCW Industries, La Canada, Cal., USA, 1993
[6] Spalding, D. B., Proposal for a Diffusional Radiation Model, Unpublished technical memorandum, CHAM, London, 1994
[7] Ozisik, M. N., Radiative Heat Transfer, John Wiley and Sons, New York, USA, 1973

[8] Stevanović, Ž., Numerical Aspects of Turbulent Momentum and Heat Transfer (in Serbian), Faculty of Mechanical Engineering, University of Niš, Niš, Serbia, 2008
[9] Zhang, X.G., *et al*., Numerical Simulations on Fire Spread and Smoke Movement in an Underground Car Park, *Building and Environment, 42* (2007), 10, pp. 3466-3475.
[10] Vidaković, M., Fire and Architectural Engineering (in Serbian), Handbook, Fahrenheit, Belgrade, 1995
[11] Versteeg, H. K., Malalasekera, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Prentice Hall, Upper Sadle River, N. J., USA, 1995
[12] Vidaković, M., Thermodynamics of Uncontrolled Combustion, Part I (in Serbian), Prometej, Novi Sad, Serbia, 1994

Author´s affiliation:

M. J. Banjac (**corresponding author**)
Faculty of Mechanical Engineering
University of Belgrade
16, Kraljice Marije Str., 11120 Belgrade, Serbia
E-mail: mbanjac@mas.bg.ac.yu

B. M. Nikolić
Tehnikum Taurunum – Technical School
Department for Fire Safety
Belgrade, Serbia

Paper submitted: December 1, 2008
Paper revised: December 24, 2008
Paper accepted: January 1, 2009