UDK: 624.19:614.841:517.938
COSATI: 13-12, 20-04

Influence of Thermal Boundary Conditions in a Numerical Simulation of a Small-Scale Tunnel Fire

Mirko Kozić, PhD (Eng)[1)]

The influence of different thermal boundary conditions, i.e. adiabatic and constant wall temperature is examined using the CFD simulation of a small-scale tunnel fire. The obtained temperature profiles along the vertical line in the plane of symmetry at three stations along the tunnel are compared with available experimental and numerical ones. Simple volumetric heat sources are used in modeling combustion due to little effect of different combustion models on the region of temperature field that is not too close to the fire source. In cases of natural convection and forced convection with low ventilation velocity, the turbulent flow of heated air and smoke is primarily affected by the buoyant force caused by changes in volume which are connected with the temperature gradient. The numerical results show that different thermal boundary conditions give considerable differences in temperature distribution in the tunnel, affecting the intensity of the buoyant force that influences the velocity field and the smoke spreading region. This fact must be taken into account in a fire simulation inside a real traffic tunnel, because the wall heat transfer coefficients may not be fully known.

Key words: **fluid dynamics, numerical simulation, fire, tunnel, boundary conditions, thermal conditions.**

Introduction

A FIRE initializing on the floor of a straight traffic tunnel without either artificial or natural ventilation, causes a hot plume above the fire, which entrains the surrounding colder air into the plume. The plume reaches the ceiling of the tunnel, causing two streams of the hot gases layers which flow in the opposite directions along the ceiling. The hot gases are combustion products, e.g. smoke and carbon monoxide. In the case without any ventilation, the flow field is symmetric with respect to the plume axis. Due to heat loss and mixing with cold air, two layering streams lose their buoyancy and identity when the distance from the plume increases. The symmetry in the plume and ceiling layers disappears if natural or artificial/forced ventilation is introduced.

The ventilation influences the lengths of two ceiling layers; the length of the one flowing against the ventilation stream is reduced. It is expected that the length of the hot ceiling layer mainly depends on the fire intensity and the ventilation velocity. The critical velocity needed to stop upstream movement of smoke from a fire in a tunnel can be obtained by simple empirical models [1-6]. But the existing experimental data still show an inadequate basic understanding of the interaction between buoyancy-driven combustion products and forced ventilation, the validity of small-scale results extrapolation to large scales, and the effect of the tunnel geometry [7]. For that reason, the CFD models are the best way for modeling fire in ventilated tunnels [8-13]. These models are based on the conservation laws of fluid mechanics, and give solution in terms of space coordinates and time. A resolution of the problem mainly depends on combustion, turbulent and radiation models, enclosed space complexity and a grid resolution.

In the present study the CFD simulations are used to show substantial influence of thermal boundary conditions on the length of the hot ceiling layer as well as temperature and smoke concentration distributions in the small-scale tunnel fire. Two different thermal boundary conditions are used at the tunnel walls, adiabatic with zero heat flux, and constant wall temperature fixed at ambient temperature, which gives the maximum rate of heat transfer. In the latter, heat transfer through the wall is modeled by solving the thermal conduction inside the wall. The results for temperature distribution at three stations along the tunnel are compared with available experimental and numerical data.

Mathematical model

The governing equations for the transport of mass, momentum, energy, turbulence parameters and species concentration can be written in a general form using the Cartesian tensor notation [14] as:

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

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}= -\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_i}{\partial x_j}-\rho\overline{u_i' u_j'}\right)+\rho g_i \tag{2}$$

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u_j\phi)}{\partial x_j}=\frac{\partial}{\partial x_i}\left(\Gamma\frac{\partial\phi}{\partial x_i}-\rho\overline{u_i'\phi'}\right)+S_\phi \tag{3}$$

1) Military Technical Institute (VTI), Ratka Resanovića 1, 11132 Belgrade, SERBIA

where Γ is the generalized diffusion coefficient whereas the term $S\phi$ represents the source or the sink of the appropriate variable. For enthalpy and species concentration it is the rate of heat release and the rate of creation or destruction of species, respectively. The perfect gas equation must be added in order to enable the calculation of the fluid density. Turbulence is modeled using a modified *k-ε* model that includes the effect of buoyancy and wall damping on the turbulence. The source terms for *k* and *ε* are given by

$$S_k = P + G_B - \rho\varepsilon \quad (4)$$

$$S_\varepsilon = c_1 \left(P + G_B \right) \frac{\varepsilon}{k} - c_2 \rho \frac{\varepsilon^2}{k} \quad (5)$$

In the above equations turbulence production due to shear and buoyancy are designated by P and G_B, respectively.

The latter is given by

$$G_B = -\frac{\mu_t}{\sigma_t} \frac{1}{\rho} g_i \frac{\partial \rho}{\partial x_i} \quad (6)$$

where μ_t is the turbulent viscosity, σ_t is the turbulent Prandtl number, and g_i is the component of the gravitational acceleration along the coordinate direction *i*. The term G_B describes the exchange of turbulence kinetic energy with potential energy of the system. Its sign depends on the stability of stratification. In regions where the flow is stable, it acts to suppress turbulence. The turbulent viscosity is expressed in terms of *k* and *ε* as

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon} \quad (7)$$

while the turbulent diffusion coefficient for a scalar (enthalpy, species concentration) is given as

$$\Gamma = \frac{\mu_t}{\sigma_t} \quad (8)$$

Combustion, radiation and smoke transport models

According to [15] there is little effect of different combustion models in predicting airflow and temperature distributions in a small-scale tunnel fire. Because of that, in the paper the fire is represented as volumetric distributions of heat and mass sources assumed to be uniform over the flame volume. Although the simplest, this model is able to provide results comparable to more complex combustion models in some cases, such as those where the shape and volume of the flaming region are known in advance, and do not depend on the local flow conditions, i.e. the cases in which the fire source is far from walls and openings. The model represented as sources distributions of heat and mass does not simulate the combustion process, but only the transport of heat and smoke away from the flame.

The heat release rate is controlled by the burner, i.e. by the rate of the fuel consumption $\dot{m}_{fu}$, the heating value of the fuel H_{fu}, combustion efficiency η, and is calculated as

$$Q = \dot{m}_{fu} H_{fu} \eta \quad (9)$$

Additional source terms must be included in the fire region in the continuity and energy equations. For the continuity equation the source term is determined from the rate of the fuel consumption, while for the energy equation it depends on the fraction of heat due to radiation. There are two approaches in treating radiation transport. The first is simple and ignores thermal radiation in the surrounding medium, assuming that a fixed fraction of the total heat released in the fire is lost without changing the temperature distribution within the tunnel, while the rest is transported by the fluid. Experiments indicate that this fraction is in the range from 0.2 to 0.4 [16]. The energy convected by the fluid can be expressed as

$$Q_C = \dot{m}_{fu} H_{fu} \eta \left(1 - \chi_R \right) \quad (10)$$

where χ_R is the radiation fraction.

In the detailed treatment of radiation the entire heat released in the fire is taken in the energy equation, and the radiation model determines the amount of energy lost to the walls. This model requires additional information about the radiation properties both of the fluid within tunnel and the tunnel walls. However, radiation can play a very important role typically when temperature is higher than 600 K. The model without radiation is used in the analysis because the temperature within the tunnel is well below 600 K, but the fire heat release rate is reduced to account for the radiation loss.

A transport equation for smoke mass concentration that changes as a function of time and position must be solved. The simplest approach uses an assumption that the smoke production is proportional to the heat output, with a constant conversion factor that is determined experimentally. In the analysis, the smoke is assumed to follow the motion of the heated air, whereas its concentration is obtained by solving a transport equation for a passive scalar. In combination with a volumetric heat source, the quantity of smoke released is uniformly distributed over the prescribed flaming volume.

Numerical solution

The numerical simulations are obtained using FLUENT, a commercial computational fluid dynamics (CFD) code. The governing equations are solved using the finite-volume method with a staggered mesh where the flow variables are located. The scalar quantities (pressure, temperature, smoke concentration) are stored at the centers of the control volumes i.e. cells, while the velocity components are located at the cells faces. In this way, a strong coupling between the velocity and pressure fields is maintained. The convection-diffusion terms in the conservation equations are represented by the Power-law scheme. The SIMPLER algorithm is employed to calculate the pressure field. At the convergence the sum for all cells of the normalized absolute residuals for all the dependent variables is to drop for three orders of magnitude.

Results and discussion

The experiment of a small-scale fire was performed in a tunnel in the laboratory [15]. The geometry of the tunnel is shown in Fig.1. The main test section is 6 m long, with

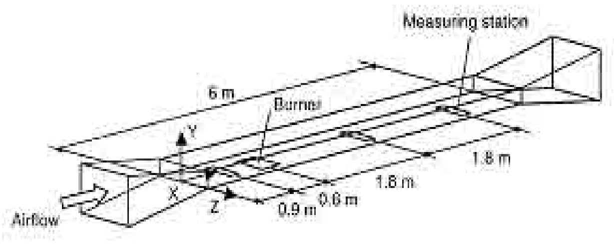


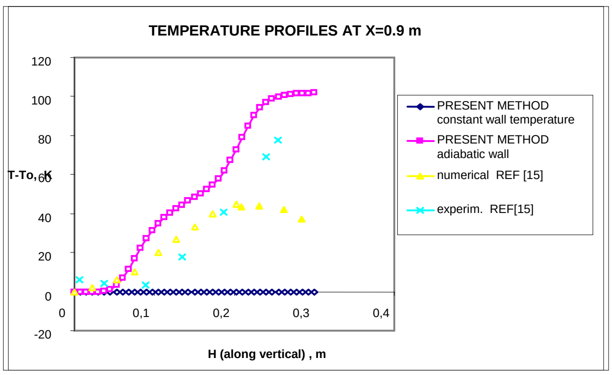
Figure1. Tunnel small-fire geometry

the rectangular cross section 0.9 m wide and 0.3 m high. The fire is placed 1.5 m from the inlet of the main test section in the middle of the floor. The burner has the dimensions 0.18 x 0.15 m with the longer side along the tunnel. The burner is supplied with liquid petroleum gas as fuel. The longitudinal ventilation is obtained by an axial fan placed at the end of the diffuser, i.e. in front of the main section.

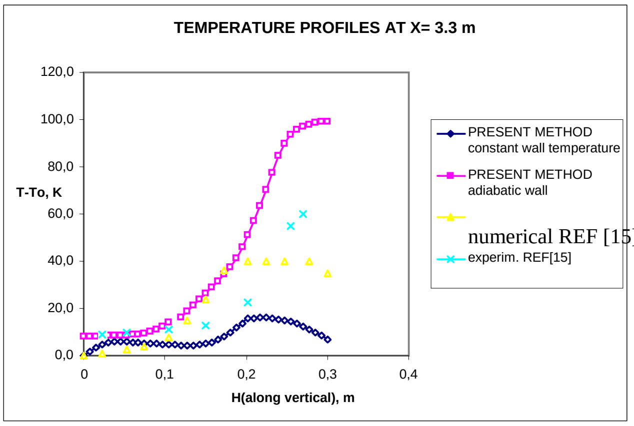
In the experiment two heat release rates (3.15 kW and 4.75 kW) and four different ventilation velocities (0.13, 0.31, 0.52 and 0.61 m/s) are examined [15]. In the paper the numerical simulation of the tunnel fire is carried out in order to obtain the influence of different thermal boundary conditions, i.e. adiabatic and constant wall temperature for a heat release rate of 3.15 kW and a ventilation velocity of 0.13 m/s. The numerical model uses a Cartesian grid to represent the geometry of the domain, and because of the well controlled heat release rate by the burner, the sources of heat and smoke are simulated inside the burner with a height equal to two grid size, i.e. 0.01 m. The tunnel is modeled with 150 x 40 x 15 (length x height x width) cells. A large number of cells in the direction of the height is placed in order to account for the quick changes in velocity and temperature fields caused by the buoyancy force.

The following boundary conditions are used in the analysis. At the inflow side of the tunnel, a uniform air velocity value is specified. At the outflow side, a uniform pressure value is specified. The non-slip condition is assumed at the tunnel walls. Two different thermal boundary conditions are used at the tunnel walls. The first is an adiabatic wall with zero heat flux, whereas the second is a constant wall temperature value that gives the maximum rate of heat transfer. In such case the heating of the wall is modeled by solving the thermal conduction inside the wall, i.e. heat loss due to conduction in the wall is taken into account.

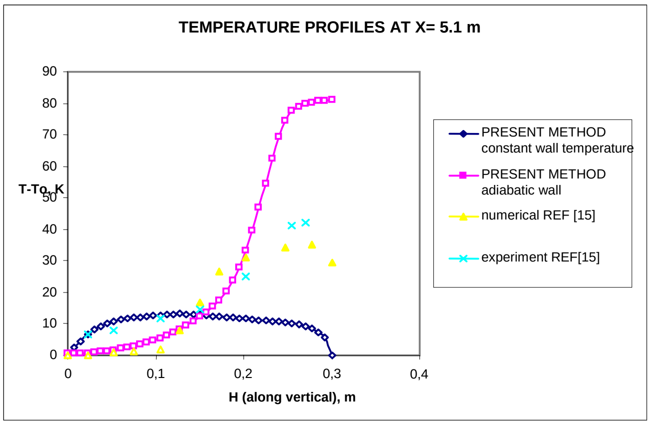
After the initiation of the fire an initial plume rises to the ceiling and ceiling layers develop during a short transition period. A steady state is reached after a short transition period whenever the heat release rate and the ventilation velocity are constant. The results in the paper are given for the steady state.



(a)



(b)



(c)

Figure 2. Temperature profiles along verticals in the plane of symmetry (a) at station x = 0.9 m, (b) at station x =3.3 m, (c) at station x = 5.1 m.

The calculated temperature profiles along the verticals in the plane of symmetry at three stations along the tunnel are compared with the experiment and numerical values from [15] in Figures 2a - 2c.

It can be seen that the largest difference of the calculated temperature profiles for the constant wall temperature and the adiabatic wall occurs at the station upstream of the fire (x = 0.9 m). The former gives no change in temperature due to conduction through the walls and the longitudinal ventilation from the entrance towards the fire, whereas the latter has a similar character as experimental measurements but shifted for about 20 K. At the first section downstream of the fire (x = 3.3 m), a temperature raise near the ceiling for the isothermal wall is much smaller than for the adiabatic wall. The character of the curve for the adiabatic wall is similar to the experimental results but the difference increases towards the ceiling where it reaches about 40 K. There is a poor agreement between the calculation and the experiment at the second section downstream of the fire (x=5.1m). The isothermal wall gives almost constant temperature from the bottom to the ceiling due to heat conduction and small buoyancy. The highest temperature for the adiabatic wall is obtained at the ceiling due to large buoyancy that lifts heated air and smoke.

The temperature and velocity fields are coupled so different thermal boundary conditions at the walls give not only different temperature distributions within the tunnel but also different velocity field and smoke concentrations. The resultant velocity field consists of the fire induced airflow and the airflow caused by the ventilation. In Fig.3 it is seen that a significant upstream flow of heated air and smoke appears near the entrance of the main section for the adiabatic wall. Although there is longitudinal ventilation, a recirculating flow near the bottom carries smoke opposite to the ventilation. It is shown in Fig.4 that for the isothermal wall there is no upstream back flow at the inlet of the tunnel main section. However, in both cases recirculating flow exists downstream of the fire source, Figures 5a and b.

Due to completely different velocity fields upstream of the fire for the isothermal and the adiabatic wall the smoke concentrations are also entirely different. For the adiabatic wall the temperature within the tunnel is much higher especially at the ceiling, increasing in this way the buoyancy that carries the smoke up to the entrance i.e. in the opposite direction from the ventilation, Fig.6. For the isothermal wall the buoyancy is small so the smoke concentration is only important at the fire as well as downstream, Fig.7.

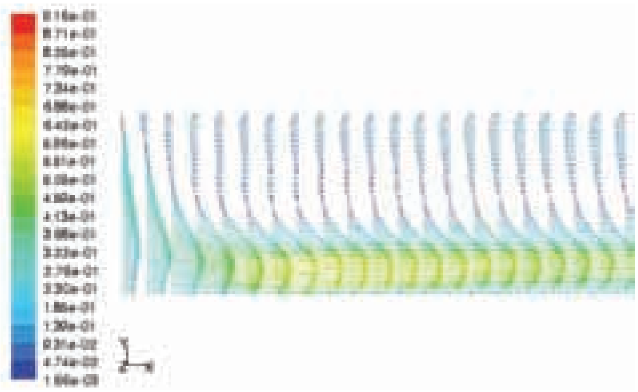


Figure 3. Velocity vectors in the plane of symmetry along 1 m behind the tunnel entrance for the adiabatic boundary condition

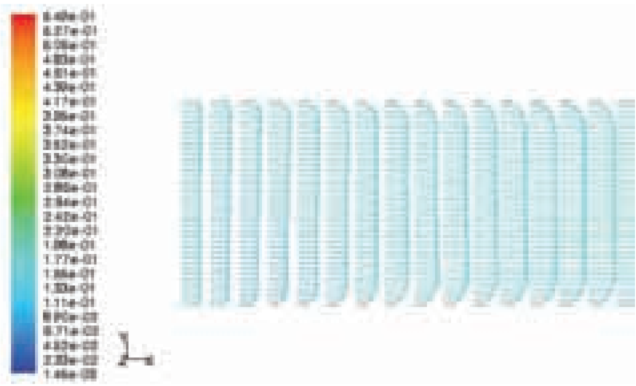
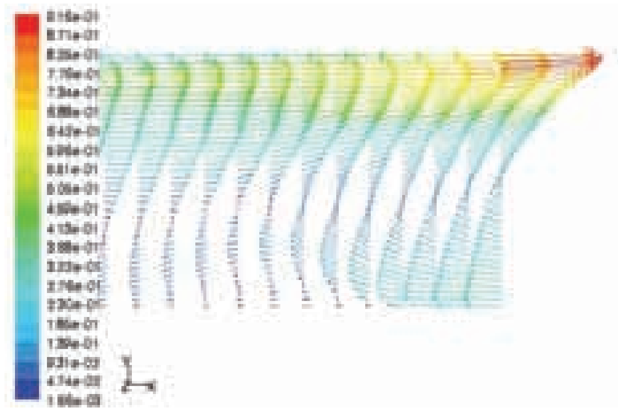
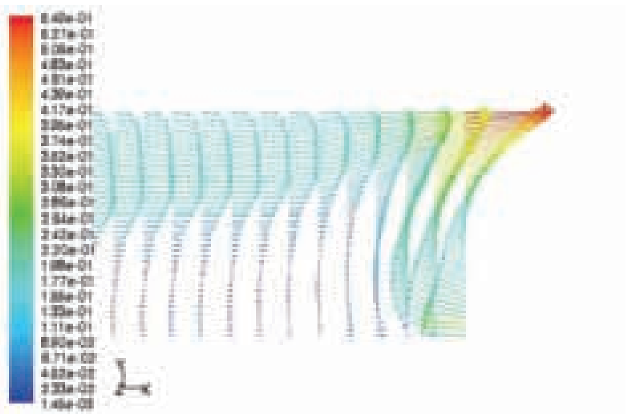


Figure 4. Velocity vectors in the plane of symmetry along 1 m behind the tunnel entrance for the constant wall temperature boundary condition



(a)



(b)

Figure 5. Velocity vectors in the plane of symmetry along 0.5 m ahead of the tunnel exit for the adiabatic (a) and the constant wall temperature boundary condition (b)

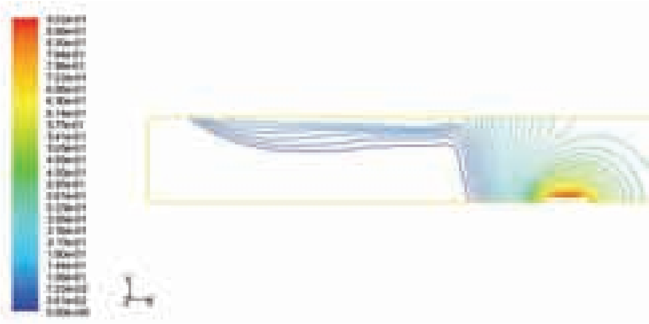


Figure 6. Smoke concentration in the plane of symmetry near the entrance of the tunnel for the adiabatic boundary condition

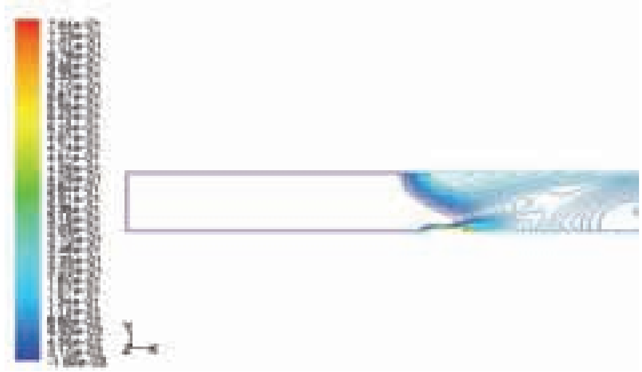


Figure 7. Smoke concentration in the plane of symmetry near the entrance of the tunnel for the constant wall temperature boundary condition

Conclusion

Numerical simulations of a small-scale tunnel fire were conducted to show the influence of thermal boundary conditions on fire-induced airflow, temperature and smoke concentration within the tunnel. The main features such as an upstream back flow and a circulating flow downstream of the fire source are captured. The temperature distributions along three verticals in the plane of the symmetry are compared with numerical and experimental results. Considerable differences are obtained pointing out to the importance of the appropriate thermal boundary conditions even in such a case as a small-scale tunnel fire.

This must be taken into account for numerical simulations of large tunnel fires where uncertainty in wall heat transfer coefficients is present, together with unknown turbulence flow parameters that enters the computational domain, unsteady fire growth rate and its shape, pressure distributions arising from natural or forced ventilation as well as soot formation and radiation. Therefore, sensitivity tests to evaluate the influence of a range of possible values for boundary conditions should be performed in CFD simulations that serve for ventilation system design.

The presented approach can be used in numerical simulations of enclosure fires, e.g. fires in rooms, shopping centers, hotels, airport terminals, hospitals, etc.

References

[1] THOMAS,P.H.: *Movement of Smoke in Horizontal Corridors against an Air Flow*, Inst. Fire Engrs Q., 1970,Vol.30, pp. 45-53.

[2] HINCKLEY,P.L.: *The Flow of Hot Gases along an Enclosed Shopping Mall: a Tentative Theory*, Fire Research Note No. 807, Fire Research Station, 1970.

[3] HESELDEN,A.J.M.: *Studies of Fire and Smoke Behavior Relevant to Tunnels*, Proc.2nd Int. Symp.On the Aerodynamics of Vehicle Tunnels, Cambridge UK, March 1976, BHRA Fluid Engineering, 1976, pp.23-25.

[4] GUELZIM,A., SOUIL,J.M., VANTELON,J.P., SOU,D.K., GABAY,D.K. and DALLEST,D.: *Modelling of the Reverse Layer of Fire-Induced Smoke in a Tunnel*, Fire Safety Science, Proceedings of the Fourth International Symposium, pp.277-288.

[5] KENNEDY,W,D., GONZALES,J.A., SANCHEZ,J.G.: *Derivation and Application of the SES Critical Velocity Equations*, ASHRAE Transactions: Research, 1996, Vol.102, No 2, pp.40-44.

[6] OKA,Y. ATKINSON,G.T.: *Control of Smoke Flow in Tunnel Fires*, Fire Safety JL, 1996, Vol 25, pp.305-322.

[7] GRANT,G.B., JAGGER,S.F. and LEA,G.J.: *Fires in Tunnels*, Phil. Trans. R. Soc. London, A 356, 1998, pp.2783-2906.

[8] KUMAR,S and COX,G.: *Mathematical Modeling of Fires in Road Tunnels*, Proc.of 5nd Int. Symp.On the Aerodynamics and Ventilation of Vehicle Tunnels, Paper B1, Lille, France, 20-22 May, 1985.

[9] SIMCOX,S., WILKES,N.S., JONES,I.P.: *Computer Simulation of the Flows of Hot Gases from Fire at King's Cross Underground Station*, Fire Safety Journal, 1992, Vol.18, pp.49-82.

[10] RHODES,N: *Review of Tunnel Fire and Smoke Simulation*, Proc .of 8th Int. Confer. on the Aerodynamics and Ventilation of Vehicle Tunnels, Liverpool, 6-8 July 1994, pp.471-486.

[11] WOODBURN,P.J., BRITTER,R.E.: *CFD Simulation of a Tunnel Fire- Part I*, "Fire Safety Journal, 1996, Vol.26, pp.35-52.

[12] LEA,C.J., OWENS,M., JONES,I.P.: *Computational Fluid Dynamics Modeling on a One –third Scale Model of a Channel Tunnel Heavy Goods Vehicle Shuttle*, Proc. of 9th Int. Confer. on the Aerodynamics and Ventilation of Vehicle Tunnels, pp. 681-692, 6-8 October, Aosta Valley, Italy, 1997, pp.681-692.

[13] EDWARDS,J.C., HWANG,C.C.: *CFD Analysis of Mine Fire Smoke Spread and Reverse Flow Conditions*, Proceeding of 8th US Mine Ventilation Symposium, 1999, pp.417-422.

[14] PTANKAR,S.V.: *Numerical Heat Transfer and Fluid Flow*, Taylor and Francis, 1980.

[15] XUE,H., HO,J., CHENG,Y.M.: *Comparison of different combustion models in enclosure fire simulation*, Fire Safety Journal, 2001, pp.37-54.

[16] MARKSTEIN,G.H.: *Relationship Between Smoke and Radiant Emission from Buoyant Turbulent and Laminar Diffusion Flames*, 20th Symposium (International) on Combustion, The Combustion Institute, Pittsburg, 1984, pp.1055-1061.

Received: 15.04.2009.

Uticaj termalnih graničnih uslova u numeričkoj simulaciji požara u malom tunelu

Korišćenjem CFD simulacije požara u malom tunelu, ispitan je uticaj različitih termičkih graničnih uslova, odnosno adijabatskog i zida sa konstantnom temperaturom. Dobijeni profili temperature duž vertikale u tri preseka u tunelu, upoređeni su sa eksperimentalnim i numeričkim rezultatima iz literature. U modeliranju sagorevanja korišćeni su jednostavni zapreminski izvori toplote, usled malog efekta različitih modela sagorevanja na deo temperaturnog i brzinskog polja koji nije preblizu vatri. U slučajevima prirodne konvekcije i prinudne konvekcije sa malim brzinama ventilacije, sila potiska prouzrokovana promenama zapremine koje su povezane sa gradijentom temperature, najviše utiče na turbulentno kretanje zagrejanog vazduha i dima. Numerički rezultati pokazuju da različiti termički granični uslovi daju značajne razlike u raspodeli temperature u tunelu, čime utiču na veličinu sile potiska odnosno na brzinu i oblast širenja dima. Ova činjenica mora se uzeti u obzir pri simulaciji požara u realnom saobraćajnom tunelu, jer koeficijenti prelaza toplote na zidu najčešće nisu potpuno poznati.

Ključne reči: **dinamika fluida, numerička simulacija, požar, tunel, granični uslovi, termalni uslovi.**

Влияние термических предельных условий в цифровом моделировании пожара в маленькой трубе (туннеле)

Пользованием ЦФД моделирования (симуляции) пожара в маленькой трубе, исследовано влияние различных термических предельных условий, т.е. адиабатической и стены со постоянной температурой. Полученые графические изображения температуры вдоль перпендикуляра в трёх сечениях в трубе, сопоставлены с экспериментальными и цифровыми результатами из инструкций по эксплуатации. В моделировании сгорания использованы простые объёмные источники теплоты, из-за маленького (слабого) эффекта различных моделей сгорания на часть температурного и скоростного поля, которое не находится слишком близко огня. В случаях естественной конвекции и вынуждённой конвекции со маленькими скоростями вентиляции, сила тяги вызвана изменениями объёма связаными с градиентом температуры, больше всего влияет на бурное движение нагретого воздуха и дыма. Цифровые результаты показывают, что различные

термические предельные условия дают значительные разницы в распределении температуры в трубе, чем влияют на величину (размер) силы тяги, т.е. на скорость и область расширения дыма. Этот факт надо учитывать при симуляции пожара в действительной трубе для транспорта, ибо коэффициенты перехода теплоты на стене может быть не совсем известны.

Ключевые слова: **Динамика жидких тел (флуидов), цифровое моделирование, пожар, туннель (труба), предельные условия, термические условия.**

Influence des conditions thermales de limite dans la simulation numérique de l'incendie dans le petit tunnel

On a examiné l'influence de différentes conditions thermique de limite, notamment adiabatique et le parois à la température constante, au moyen de la simulation CFD de l'incendie dans le petit tunnel. On a obtenu les profils de température le long de la verticale aux trois intersections dans le tunnel. Ils ont été comparés avec les résultats d'essais et les résultats numériques figurant dans la littérature. Dans la modélisation de combustion on a utilisé les sources simples de chaleurs à cause du faible effet de divers modèles de combustion sur la région de température et de vitesse qui n'est pas située prés du feu. Dans les cas de la convection naturelle et la convection forcée avec les petites vitesses de ventilation, la force de poussée, causée par les variations du volume liées au gradient de température influe le plus au mouvement turbulent de l'air chauffé et de fumée. Les résultats numériques démontrent que les différentes conditions de limite provoquent de grandes différences dans la distribution de la température dans le tunnel, ce qui influe à l'intensité de la force de poussée, notamment à la vitesse et la région de la propagation de fumée. Ce fait doit être considéré lors de la simulation de l'incendie dans le tunnel réel de circulation car les coefficients du passage sur les parois ne sont peut-être pas connus complètement.

***Mots clés*: dynamique des fluides, simulation numérique, incendie, tunnel, conditions de limite, conditions thermales.**