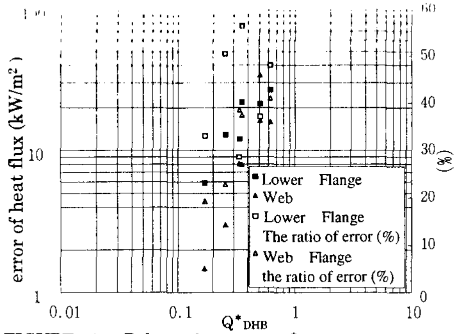

FIGURE.28 Relation between Q^*_{DHB} and error of heat flux.

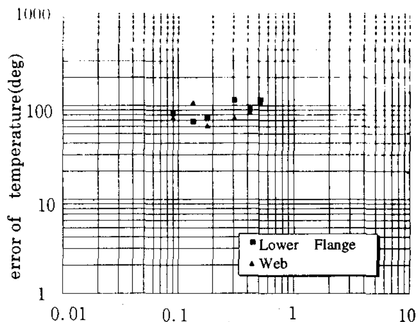

FIGURE.29 Relation between QH_B* and error of temperature.

REFERENCE

1) Yokobayashi,Y.,Hasemi,Y.,Wakamatsu, T., Wakamatsu, T., "Experimental Study on the Heating Mechanism and Thermal Response of a Steel Beam Under Ceiling Exposed to a Localized Fires ," Journal of Structural and Construction Engineering, Transactions of AJI , No .498 (in Japanese)
2) Wakamatsu,T.,Hasemi,Y., Yokobayashi, Y.,A.V.Ptchelintsev,"Heating Mechanism of Building Components Exposed to a Localized Fire - FEM Thermal and Structural Analysis of a Steel Beam Under Ceiling," Proceedings, OMAE '97 Yokohama 1997
3) Wakamatsu, T.,Hasemi, Y., "Heating Mechanism of Building Components Exposed to a Localized Fire- FDM Thermal Analysis of a Steel Beam Under Ceiling -," Fire Science and Technology - Proceedings of the Third Asia-Oceania Symposium, Singapore pp335-346,1998.06
4) Hara, T., Yokoi, M., Morikawa, Y., " Analysis of Thermal Plumes with Numerical Simulation and Experimental Results Basic Plume Analyses for the CFD Performance of the Standard k-e model. Annual Meeting," Architectural Institute of Japan , 1998(in Japanese)
5) Yoshie, R.,WEI Ran, " CFD Analysis of Fire Plume - (Part1) Influence of Computational Domain and Mesh Division on Numerical solution," Annual Meeting, Architectural Institute of Japan , 1998(in Japanese)
6) Yoshie,R. ,and WEI Ran "CFD Analysis of Fire Plume - (Part2) Influence of Numerical scheme for convection terms and model for fire source," Annual Meeting, Architectural Institute of Japan , 1998(in Japanese)
7) Welch, S.,P.Rubini "SOFIE Users Manual" School of Mechanical Engineering, Cranfield University, 1996
8) A.V.Ptchelintsev, M.Nikolaenko, Hayashi, Y., Wakamatsu, T. ,"Sofie Computation of the BRI Room Corner Fire Interim Report", SOFIE meeting, 1998
9) Rodi, W. "Turbulent buoyant jets and plumes", Pergamon, 1982
10) Bressloff, N.W., Moss, J.B., Rubini, P.A."Assessment of a differential total absorptivity solution to the radiative transfer equation as applied in the discrete transfer radiation model", Numerical Heat Transfer, Part B: Fundamentals, vol. 29, part 3, pp. 381-397, May, 1996
11) Bressloff, N.W., Moss, J.B., Rubini, P.A. "CFD prediction of coupled radiation heat transfer and soot production in turbulent flames," Twenty-sixth Symposium on Combustion, The Combustion Institute, pp. 2379-2386, 1996
12) "Fire safety design method, comprehensive fire preventing procedures, ed. by Japan Institute of Construction Engineering," vol.4, April, 1988 (in Japanese)
13) S.Welch,A.Ptchelintsev., "Numerical Prediction of Heat Transfer to a Steel Beam in a Localized Fire," Proceedings of the Second International SeminarFire-and Explosion Hazard of Substances and Venting of Deflagrations,11-15 August, 1997, Moscow, Russia, pp. 299-313.

Fires in Tunnels: Three-Dimensional Numerical Simulation and Comparison with the Experiment

ALEXEY V. KARPOV, DMITRY V. MAKAROV, VLADIMIR V MOLKOV and ALEXEY M. RYZHOV
All-Russian Research Institute for Fire Protection,
VNIIPO 12, Balashikha-3, Moscow region, 143900 Russia

ABSTRACT

The computational fluid dynamics code SOFIE has been used for three-dimensional unsteady simulation of experimental fire in the tunnel with sizes 21x1.5x1.6 m performed previously by Japanese researchers. The steady and unsteady burning rate approaches were used for numerical modeling. It has been shown that the computed flow velocities and temperature distribution for unsteady, growing in time, burning rate are in better correspondence to the experimental data. It is shown that an approach to the description of burning rate influences significantly on the magnitude and the distribution of heat fluxes in the enclosure, that is important in particular for solving the fire resistance problem.

KEYWORDS

Fire, tunnel, unsteady CFD modelling, variable burning rate, temperature, velosity, heat flux

INTRODUCTION

The computational fluid dynamics (CFD) is employed more and more often for fire phenomena investigation during last decades. CFD modeling is the essential tool for performance based approach to fire safety engineering. It enables to realize reliable cost-effective fire safety design of the premises, minimize expenses on testing, study regularities of fire phenomena in numerical experiments.

The CFD technique of fire modeling is the most informative because it calculates time and space distribution of thermo- and hydrodynamic parameters. Advanced CFD codes become more complicated and take into account new physical aspects of fire phenomenon. On the one hand it gives new parameters which being varied can influence on numerical solution. On the other hand the calculation results begin to depend not only on the quality of the code, but on the correct mathematical realization and physical formulation of a problem. This peculiarity appears more dramatically when the information about physical phenomena needed for the calculations is insufficient or the acceptable models for them are absent at the present time (e.g. burning rate, combustion incompleteness, etc).

The influence of mass and heat transfer in compartment on the processes in fire source and particularly on liquid fuel burning rate is one of the problems which are not investigated well enough. The huge number of experimental studies were devoted to steady maximum burning rate determination, while it is a variable value during some time after ignition. It depends not only on fuel properties, but also on incident heat flux, flow velocity, oxygen concentration, etc.

The work [1,2] presents apparently the most fundamental study devoted to the burning rate of pool fires of different diameters. Mainly burning rates for open fires are considered. It is well known that burning rate can vary in wide range for the same fuel under different fire conditions. So the burning rate of n-heptane for the open large diameter pool fire is 8 mm/min according data [3] where the method [4] was used, but it is equal to 1.2 - 2.0 mm/min only in 50 mm diameter pool according data [5].

Usually for the modeling of liquid combustion researchers use burning rate and its dependence on time obtained from experimental data or their own estimations. The constant burning rate of liquid fuel [2,7] was used in the paper [6] for the modeling of fire in tunnel. In study [8] the simulation of fire in the turbine hall of power plant was performed. The hall sizes were 40.0x55.5x29.6 m, the leakage of oil took place on the area 100 m^2. The rate of fuel evaporation was also supposed to be constant and equal to 3 mm/min, that produced the flow of fuel vapours 4.25 kg/s and heat release rate of 180 MW. The unsteady increasing in time burning rate for closed fire was used for example in the study [9] for the simulation of ethanol combustion in the pool with sizes 0.44x0.44 m. The experimental burning rate transition to steady state (stabilisation time) was 4 min [10].

In paper [11] the scenario with transient burning rate was used also. The experiment on methanol combustion on the area 3.24 m^2 in the centre of the room with sizes 24x30x26 m [12] was simulated. The stabilisation time was accepted equal to 4 min. The similar approach was used in study [13].

Obviously the stabilization time and maximum burning rate should affect on the numerical solution. Unfortunately, this problem is not illuminated sufficiently in studies on CFD fire modeling. The reproduction of fire source parameters supposed to be the key factor for fire dynamics. From publications it is known that burning rate of liquid hydrocarbons for open flames stabilizes from 2-4 min [5,23] up to 30 min [24] and is determined, in general, by radiation and convective heat fluxes from the flame. In case of closed fires the radiation flux to liquid pool changes while the values of temperature of combustion products and enclosure walls increase. Since the heating of concrete walls may last for a long time, the burning rate can increase continuously.

The present study has been undertaken to compare available experimental data with computational results obtained with advanced CFD code SOFIE for two different burning rate dependencies: steady burning rate (averaged by the experimental data) and unsteady increasing in time burning rate.

DESCRIPTION OF THE EXPERIMENT

In our study the experimental data obtained in National Research Institute of Fire and Disaster of Japan (NRIFD) were used [14]. The experimental setup represented the tunnel with semicircular ceiling and following sizes: length - 21 m, width - 1.5 m, ceiling height - 1.6 m, cross sectional area - 2.01 m^2, wall thickness - 0.1 m. The tunnel was made of heat-proof concrete (λ=0.75 W/(m·K)). A 0.5 m square steel tray with n-heptane was used as a fire source. During the experiment the temperature profiles were measured along the tunnel center line directly under the ceiling and 0.2 m below it.[1] Besides that the values of flow velocity under the ceiling and near the floor at the distance of 3 m from the tunnel exit cross section are known to be 3 and 0.5 m/s respectively and that the temperature of combustion products at the exit achieved 300 °C by the end of the experiment.

MATHEMATICAL MODEL

The CFD code SOFIE (version 2.06.01) was used for fire modeling [15]. This code was developed by consortium of six fire research organizations and universities from four European countries. The mathematical model realized using this code consists of the following equation set:

- continuity equation and momentum conservation equations

$$\frac{\partial \rho}{\partial \tau} + \frac{\partial \rho u_j}{\partial x_j} = 0, \quad (1)$$

$$\frac{\partial \rho u_i}{\partial \tau} + \frac{\partial \rho u_j u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\mu + \mu_t) \frac{\partial u_i}{\partial x_j} \right) + \frac{\partial}{\partial x_j}\left(\mu_t \frac{\partial u_j}{\partial x_i} \right) + \rho g_i, \quad (2)$$

- equations for k-ε turbulence model of Launder-Spalding [16] with buoyancy corrections [17]

$$\frac{\partial \rho k}{\partial \tau} + \frac{\partial \rho u_j k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G_K + G_B - \rho\varepsilon , \quad (3)$$

$$\frac{\partial \rho \varepsilon}{\partial \tau} + \frac{\partial \rho u_j \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k}\left[c_1 (G_K + G_B) \right] - c_2 \rho \frac{\varepsilon^2}{k} , \quad (4)$$

where $\mu_t = c_\mu \rho \frac{k^2}{\varepsilon}$,

$$G_K = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} ,$$

[1] The experimental data are kindly presented by NRIFD staff member Dr. S. Miyazaki

$$G_B = -\beta g \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial x_j},$$

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p,$$

- energy conservation equation

$$\frac{\partial \rho h}{\partial \tau} + \frac{\partial \rho u_j h}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\frac{\mu}{\sigma_h} + \frac{\mu_t}{\sigma_t}\right)\frac{\partial h}{\partial x_j}\right) + S_{h,rad} + S_{h,comb}, \quad (5)$$

- equations for species

$$\frac{\partial \rho Y_i}{\partial \tau} + \frac{\partial \rho u_j Y_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\frac{\mu}{Sc} + \frac{\mu_t}{Sc_t}\right)\frac{\partial Y_i}{\partial x_j}\right) + S_i. \quad (6)$$

The combustion was modeled by Magnussen and Hjertager's Eddy Break-Up model [18]:

$$S_f = -C\rho\frac{\varepsilon}{k}\min\left\{Y_f, \frac{Y_{ox}}{s}, \frac{Y_{pr}}{(1+s)}\right\}. \quad (7)$$

For radiation calculations the discrete transfer radiation model [19] was used. The radiative properties were calculated by weighted sum of gray gases model (WSGG) [20] with Truelove coefficients [21]. This model enables taking into account the influence of soot, the concentration of which was determined by Tesner's model [22].

The calculation domain included a quarter of real tunnel, confined by symmetry planes and outside region with length of 2 m and height of 2.1 m (by 0.4 m higher than outer surface of the tunnel), which was located behind the exit from the tunnel. Such external domain is known to improve the numerical convergence of natural convection problems with boundary conditions of the kind $\partial p/\partial n = 0$. The general view of calculation domain is given in Fig.1.

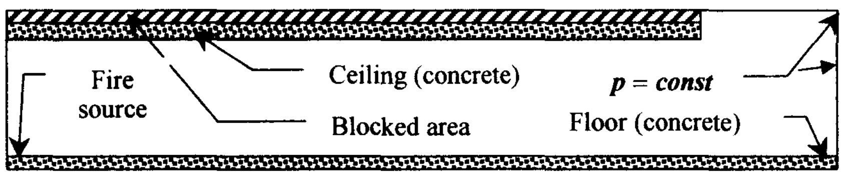

FIGURE 1. Computational domain

The semicircular ceiling of the tunnel was approximated by rectangular control volumes. The cross section area for such discretization was equal to 2.03 m^2 (the relative difference from real value – 1%). The approximation of the ceiling and computational grid in the cross section are presented in Fig. 2. No-slip conditions on solid surfaces and $\partial p/\partial n = 0$ in the outside re-

gion were used as the boundary conditions for the momentum equation. At the symmetry planes $\partial\Phi/\partial n = 0$ conditions were used for all, except of V-momentum, equations. The emissivity of the concrete surface was supposed to be 0.9.

Preliminary the sensitivity of results to computational grid was tested on the same problem in the steady state mode. Two grids were used: LxHxW=35x31x19=20615 and 47x40x24=45120 control volumes respectively. The discrepancy between the solutions was found to be small. The values of typical parameters (maximum velocity, temperature, convective and radiation fluxes) have changed less than by 6%. Henceforth for all calculations the computational grid 35x31x19 was used.

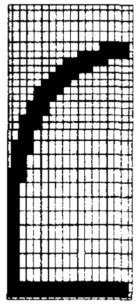

FIGURE 2. Tunnel cross-section area and computational grid

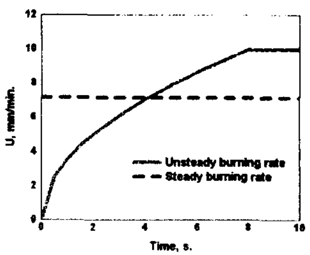

FIGURE 3. Steady and unsteady burning rate

In this study two approximations for burning rate were used: with steady burning rate (similar to calculations in original study of Japanese researchers [14]) and with unsteady burning rate continuously increasing during 8 min. At last case the burning rate was approximated by square root time dependence $u = A\sqrt{\tau}$ correlated with experimental data, in particular [5]. The diagrams of burning rate dependencies are shown in Fig.3. In both cases it was supposed that total time of combustion was 10 min, the amount of the heptane burned – 18 liters, that corresponded to experimental data.

RESULTS OF SIMULATION

The dynamics of calculated velocity and temperature profiles for unsteady burning rate is shown in Fig.4. One can see that by the moment of 30 s the combustion products have already achieved the exit from the tunnel and the circulation type flow has settled. For visualization of the temperature field the value 343 K was used as the low boundary of the painted range. It is easy to see that the velocity in tunnel and the maximum temperature of combustion products over the fire source increase with time.

The diagrams of calculated and experimental temperature distributions for moments 2, 3, 4 and 8 min are given in Fig.5.

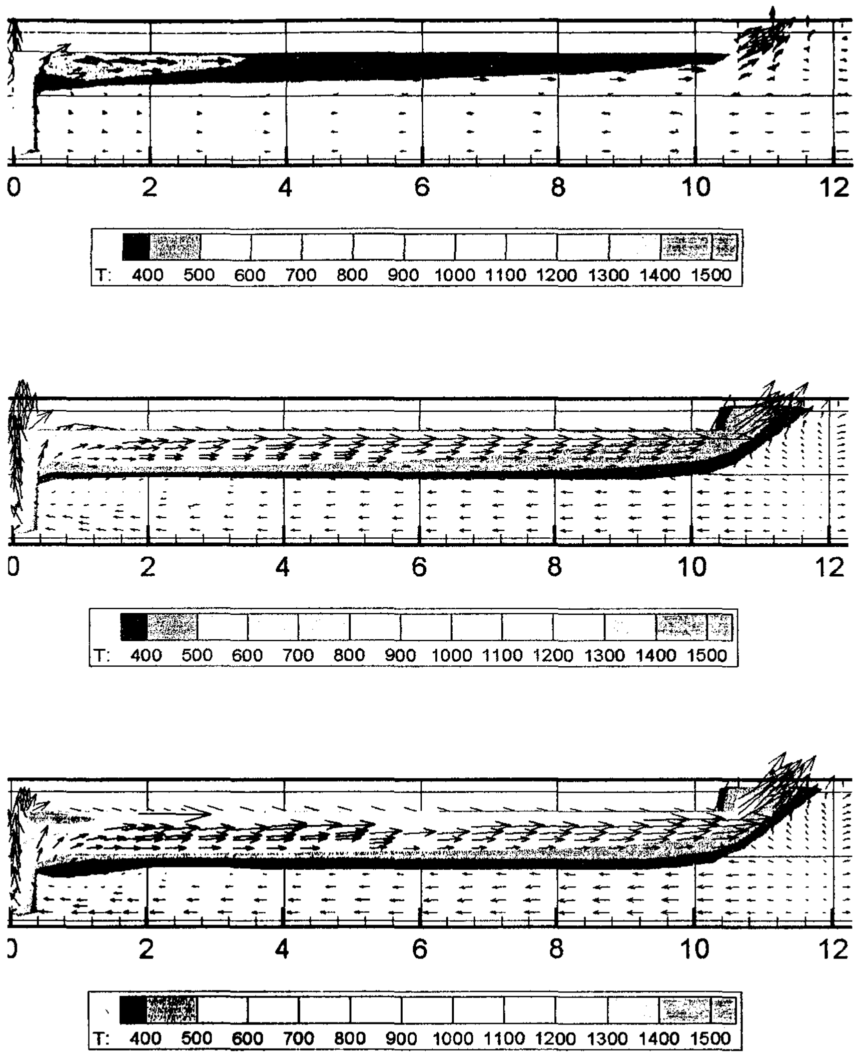

FIGURE 4. Velocity and temperature distributions in tunnel. Time 30c, 120c, 480c

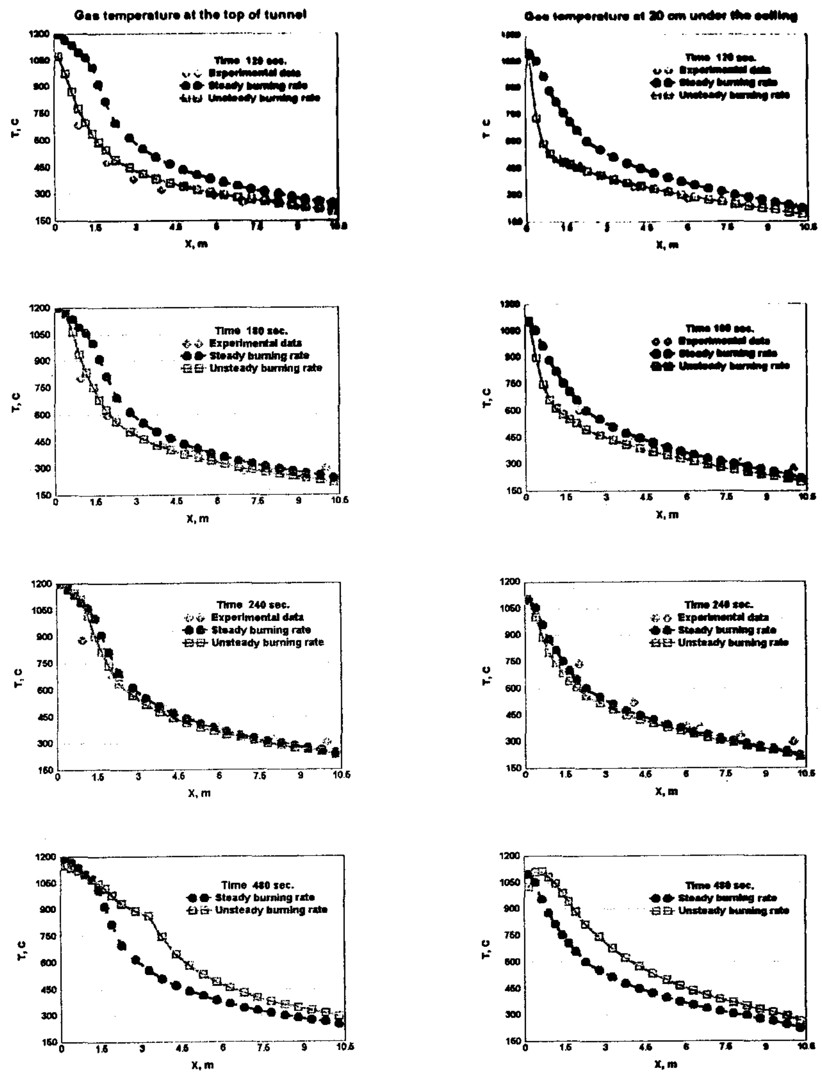

FIGURE 5. Combustion product temperature profiles along the tunnel center line

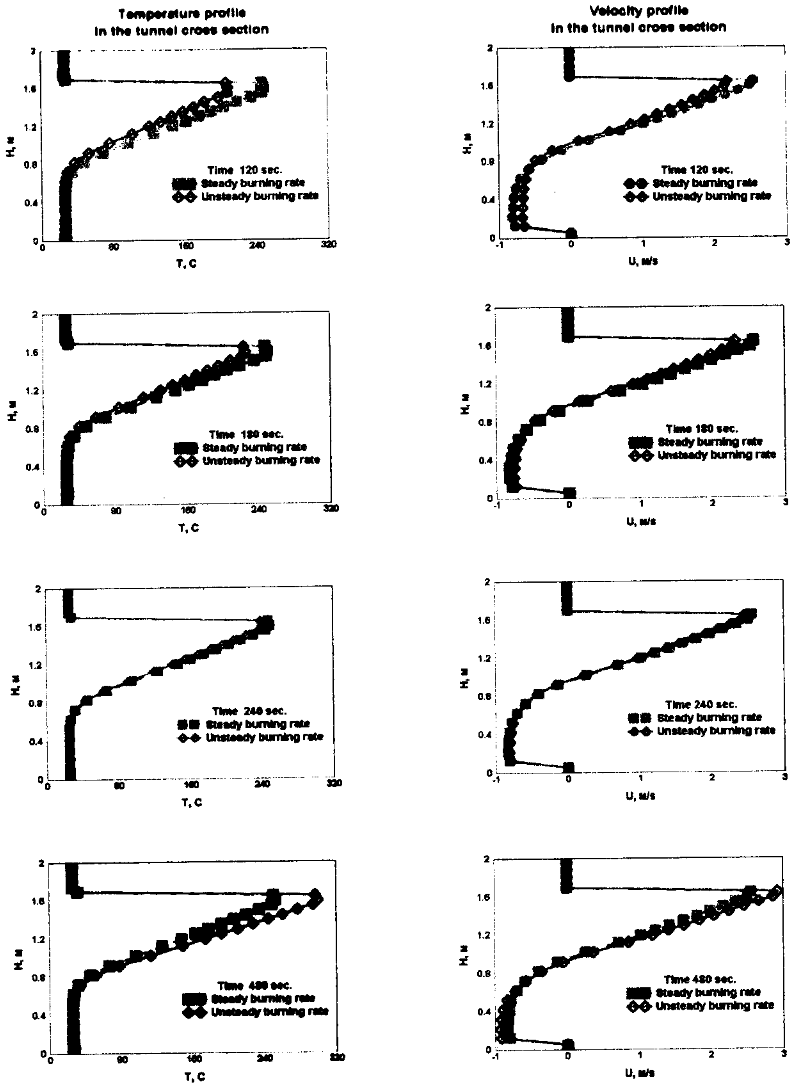

FIGURE 6 Combustion products temperature and velocity profiles in the cross section at the distance of 3 m from the tunnel exit

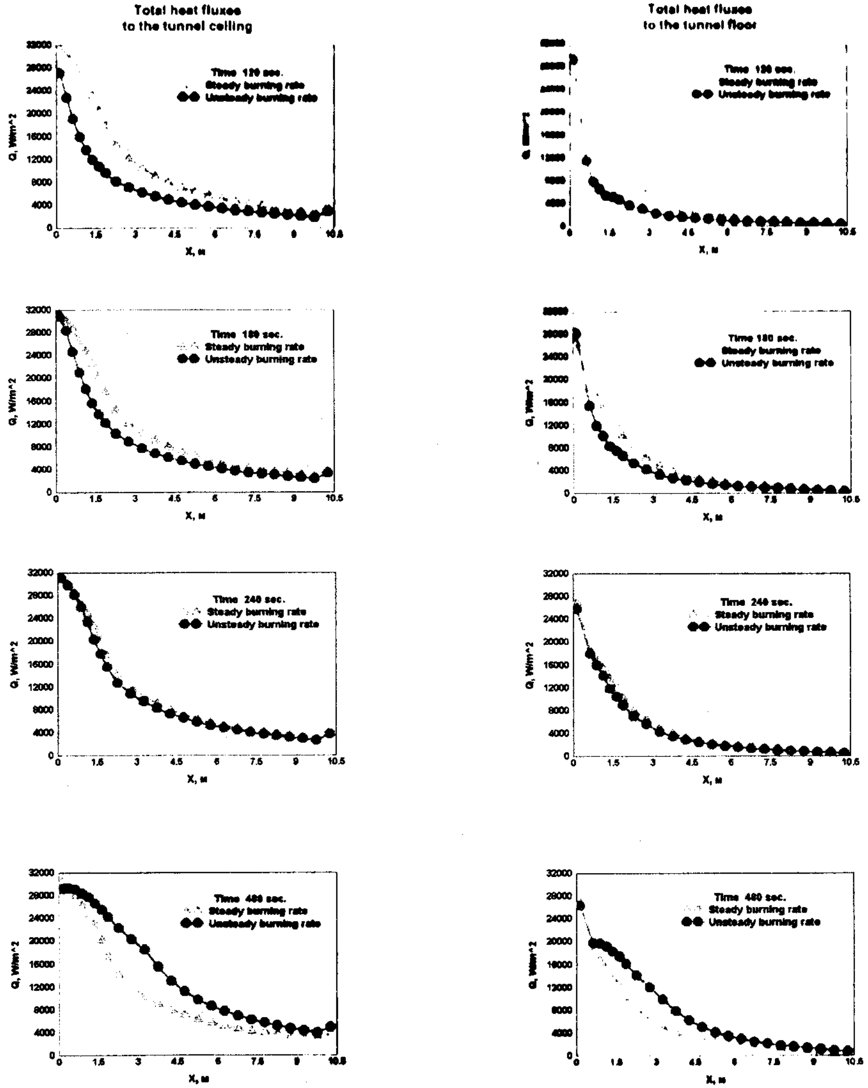

FIGURE 7. Total heat fluxes along the tunnel center line

One can see that temperature profiles for the case of unsteady burning rate are closer to the experimental values especially for the third minute. By the fourth minute these profiles slightly differ from each other. However then, as it is shown for the moment 8 min, the temperatures for the unsteady burning rate case become higher than for the steady one. It is also necessary to underline that the temperature profiles along the tunnel for the steady and unsteady burning rate are qualitatively different at this time.

The temperature and velocity profiles in the tunnel cross section at the distance of 3 m from the tunnel exit are given in Fig.6. The diagrams show the similar form of the velocity and temperature profiles for steady and unsteady burning rate. But the absolute values of these parameters change with time in a different way. In case of steady burning rate after two minutes of fire temperatures and velocities are almost constant, while in case of unsteady one they are cotinuously increasing. So the experimental values of combustion products velocity 3 m/s and temperature 300°C at the tunnel exit are achieved during numerical modeling only at the 8^{th} min of fire and only for unsteady burning rate.

The heat flux is one of the main hazardous factors of fire which is important in particular for solving fire resistance and tactics of fire extinguishment problems. Unfortunately there are no data on heat fluxes in tunnel during experiment. The calculated total heat fluxes to the floor along the tunnel center-line for moments 2, 3, 4 and 8 min are given in Fig.7. This figure shows the significant difference between the heat fluxes for cases of steady and unsteady burning rates especially during first 2-3 min. By the way it is rather interesting that for the unsteady burning rate not only the value of the heat flux changes but the form of its distribution also. The heat flux significantly increases in the central part of the tunnel. At the beginning of fire the heat flux for the unsteady burning rate was lower in 1.5 times (in the central part of the tunnel even in 2 times) than for steady one, but at the 8^{th} min it became 1.5 times higher.

One of the reasons of such phenomena is the fact that the wall surface temperature depends on the whole fire development process, while the temperature of combustion products in a plume mainly on current burning rate. In the case of unsteady burning rate the heat release during the initial period was small, so the wall temperatures are less than in the case of steady burning rate. Therefore relatively cold enclosure walls interact with becoming more and more hot combustion products and the heat flux value is high. Apparently this difference of heat fluxes may become even more significant for other fire conditions.

The obtained results demonstrated that the use of unsteady burning rate for the simulation of liquid combustion in experimental tunnel gives more correct prediction. It is no wonder because such approach better cohere to the physics of the processes. But a problem appears connected with the determination of both the maximum burning rate and its stabilization time. At the moment there are no reliable technique for calculation of these parameters. The solution of this important for practice problem can be achieved through the fundamental research, both experimental and numerical, including the conjugate problem of heat transfer in enclosure and the processes of heat transfer, evaporation and maybe boiling in the fuel tray (pool, tank, etc.).

CONCLUSIONS

The 3D unsteady CFD modeling of fire in the experimental tunnel was performed. The dependence of temperature, heat flux and velocity distribution in space and time on the character of burning rate (steady and unsteady) was studied.

It was found that calculated temperature fields for unsteady burning rate are qualitatively and quantitatively closer to the experimental data than those one for the time averaged steady burning rate. The difference between two calculated maximum temperatures at the tunnel exit is 15 %, the difference inside the tunnel reaches the value 25-30 % by the end of numerical simulation.

The calculated velocity profiles for the steady and unsteady burning rate differ slightly, but the rate of their change with time is qualitatively different. The maximum difference between two profiles is 15 % and it is reached near the tunnel ceiling by the 8 min of fire.

For the unsteady burning rate not only the value of the heat flux changes but the form of its distribution also. The heat flux significantly increases in the central part of the tunnel. At the beginning of fire the heat flux for the unsteady burning rate was lower in 1.5 times (in the central part of the tunnel even in 2 times) than for steady one, but at the 8^{th} min it became 1.5 times higher.

REFERENCES

1. Blinov V.I., Khudyakov G.N. Diffusion Combustion of Liquids. Moscow: Academy of Science, USSR, 1961. (In Russian).
2. Blinov V.I., Khudyakov G.N. On certain regularities on diffusion burning of liquids. Report of USSR Academy of Science (In translation), 113, 1957, pp.241 - 244.
3. Fire and Explosion Hazard of Substances and Materials and Their Extinguishing Agents. Handbook in 2 books. Khimiya, Moscow, 1990. (In Russian).
4. Monakhov V.T. Methods for Investigation of Fire Hazard of Substances. - Moscow: Khimiya, 1979. - 423 p. (In Russian).
5. Hayasaka H. Unsteady Burning Rates of Small Pool Fires // *Proc. of the Fifth Int. Symp. "Fire Safety Science"*, 3-7 March 1997, Melbourne, Australia. - pp.499 – 510.
6. Cox G., Kumar S. Mathematical Modelling of Fires in Road Tunnels // *Proc. of Fifth Int. Symp. "Aerodynamics and Ventilation of Vehicle Tunnels"*, 20-22 May, 1985, Lille, France. – pp.61-76.
7. Yumoto T. Radiation from Oil Tank Fire // *J. Japan Society of Fire Safety Engg.*, V.10, N3, 1971.
8. Huhtanen, R. Numerical Fire Modelling of a Turbine Hall // *Proc. of Second Int. Symp. "Fire Safety Science"* - pp.439-450.
9. Ryzhov A.M. The Basics of Differential (Field) Fire Modelling // *Proc. of Twelfth All Russia Scientific Conference on Fire Safety Problems, «Scientific and Technical Providing of Emergency and Rescue Works»*. - Moscow: VNIIPO, 1993. - pp.320-322. (In Russian)

10. Draft ISO Standard for Fire Test Fires for Fire Detection and Alarm Systems // ISO / TC/ 21 / SC3 / WG1. -1986. - June.- 10.- p.45.

11. Ryzhov A.M. Computer Modelling of 2D Convective Heat Transfer Processes in Compartment Fires // Deposited in the Main Information Centre of USSR Ministry of Interior,. – Moscow, 1988. - N338D – 46 p.

12. Yamana T., Tanaka T. Smoke Control in a Large Scale Spaces // *Fire Science and Technology*, 1985 - Vol.5, N 1 - pp. 41-54.

13. Ryzhov A.M. The Dynamics of Convective flows in Vented Tunnel Fires // *Abstracts of Fourteenth All Russia Scientific Conference on Fire Safety Problems*, V.2. – Moscow: VNIIPO, 1997. – pp.193-195.

14. Satoh K., Miyazaki S. A Numerical Study of Large Fires in Tunnels // Report Of Fire Research Institute of Japan, 1989. – No.68 - pp.19-34.

15. Welch S, Rubini P., SOFIE, Simulations of Fires in Enclosures, User Guide, Cranfield University, 1996

16. Launder B.E., Spalding D.B. The Numerical Computation of Turbulent Flow // *Comp. Meth. Appl. Mech. Eng.* - 1974. - N3. - pp.269-289.

17. Hossain M.S., Rodi W. A Turbulence Model for Buoyant Flows and Its Application for Vertical Buoyant Jets // *"Turbulent Buoyant Jets and Plums"* (Rodi W. ed.), HMT Series: Oxford, England. – 1982. - V.6. – pp.121-172.

18. Magnussen, B.F. and Hjertager, B.H. On Mathematical Modelling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion // *Sixteenth Symp. (Int.) Combust.* The Combustion Institute, Pittsburgh, PA. - 1976. – pp.719-729.

19. Lockwood, F.C. and Shah, N.G. A New Radiation Solution Method for Incorporation in General Combustion Prediction Procedures // *Eighteenth Symp. (Int.) Combust.* The Combustion Institute, Pittsburgh, PA. – 1981. – pp.1405-1414.

20. Bressloff N.W., Moss J.B., Rubini P.A. Assessment of a Total Absorptivity Solution to the Radiative Transfer Equation as Applied in the Discrete Transfer Radiation Model // *Numerical Heat Transfer, Part B.* - 1996. - V.29. – pp.381-397.

21. Truelove J.S. HTSF DR33, AERE, Harwell, Oxon, England. 1975.

22. Tesner P.A. Soot Formation During Fires // *Combustion, Explosion and Shock Waves.* – 1979. - N2. – P.2. (In Russian)

23. Kim M.B., Han Y.S., Yoon M.O. Convective Smoke Spread in a Corridor // *Proc. of Third Asia-Oceania Symposium «Fire Science and Technology»*, Singapore, June 10-12, 1998, pp 439-449.

24. Romanenko P.N., Koshmarov Y.A., Bashkirtsev M.P. Thermodynamics and Heat Transfer in Fire Engineering. Moscow: Higher Engineering Fire Technical School, Ministry of Interior, USSR, 1977.

Comparison between Experiments and Numerical Simulations of Fire Whirls Due to a Single Flame in a Vertical Square Channel with Symmetrical Corner Gaps

KOHYU SATOH
National Research Institute of Fire and Disaster
Mitaka, Tokyo 181, Japan

K.T. YANG
Department of Aerospace and Mechanical Engineering
University of Notre Dame
Notre Dame, IN 46556, USA

KUNIO KUWAHARA
The Institute of Space and Astronautical Science
Yunodai, Sagamihara, Kanagawa, Japan

ABSTRACT

Although fire whirls in large urban fires are known to cause many fire victims and extensive property damages, detailed mechanisms and physical effects of fire whirls still remain largely unknown, since real big fire whirls are relatively rare and their studies are too difficult to carry out. However, such large-scale fire whirls, until recently, can be conveniently studied by computer-based numerical simulation techniques. On the other hand, computer simulations for large-scale phenomena are only realistic if they can be validated in advance by the results of scale-modeled experiments. Despite recent studies in this regard, uncertainties of the whirling-fire phenomena are still abound. The purpose of this study was to compare quantitatively the fire-whirl temperature and velocity fields of the numerical simulations with those from laboratory experiments for the case of fire whirls generated with a single flame located in a vertical square channel with symmetrical corner gaps. Successful results of such comparisons have been obtained.

KEYWORDS

Fire Whirl, Urban Fire, Numerical Simulation, Vertical Channel, High Speed Motion Video

INTRODUCTION

It is well-known that fire whirls in large urban fires, such as those due to gigantic earthquakes in Tokyo, 1923, are known to claim many fire victims and cause extensive property damages, primarily because of their extreme destructive power. In addition, it is also known recently that such violent fire