Turkish J. Eng. Env. Sci.
31 (2007) , 391 – 402.
© TÜBİTAK

Tunnel-Curvature's Influence on the Propagation of the Consequences of Large-Scale Accidental Fire - a CFD-Investigation

M. MUHASILOVIC, M. O. DEVILLE
Ecole Polytechnique Fédérale de Lausanne STI – ISE – LIN, Station 9,
CH-1015 Lausanne-SWITZERLAND
e-mail: muhasilovic@gmx.net

Received 03.12.2007

Abstract

There is growing evidence for influence of irradiative heat-flux in the large scale confined fires onto the fire growth rate and subsequently there is a need to understand mentioned relationship regarding the development of the effective fire-suppression methods. In this study, the influence of the traffic tunnel's curvature onto the consequences of the fire on both the will be investigated. Heat radiation of the flame and the gaseous products of such non premixed combustion (NPC) depend on the geometrical position of the inflammable objects as well as on the propagation of the gaseous combustion products in such type of enclosure. In a traffic tunnel the position of the inflammable objects alone will determine the development of this large scale NPC, during this planed investigation. The simulations based on computer fluid dynamics (CFD) with characteristic conditions - real-geometry factors - will be performed.

This investigation; for the given geometrical characteristics of the investigated operating road tunnel; will establish an important crosstalk between the large-scale confined combustion of inflammable hydro-carbons and its propagation in an curved underground traffic facility.

Key words: CFD, Large-scale fires, RANS, k-ε, Tunnel-fires.

Introduction

Due to the increasing accidental fires in enclosures, especially in that area of modern society where our freedom is mostly expressed – traffic and tourism – there is an on-growing need to undertake the scientific research, aiming at better understanding of these reactive flow phenomena. One of the direct benefits coming out of such research is implication of optimal methods for fire suppression, that certainly must get along with world-wide statistical data for one tunnel-fire on each 107^{th} kilometre of covered (tunnel) road (Holmsted et al., 1996; Miles et al., 1999).

In spite of evaluations (Heins et al., 1990) and sophisticated changes (Charters et al., 1994), the zone-model approach for exploring large-scale reactive flows was overruled by the field models for CFD-research of fires in enclosures. The latter programme codes do not divide the area of interest into very few smaller control volumes only, but are based on the full solution of the fundamental physical laws of conservation. Here, the computational domain is divided in thousands of much smaller control volumes -cells- , where mathematical models, after their discretisation are "translated" into a programme code for combustion and radiation, turbulence. So between mid '80-s and mid '90-s, hardware sources supported computational domains with few thousands cells (Kumar et al., 1985; Kumar et al., 1988). But in spite of these limiting conditions very satisfying results were accomplished in attempts of both validating (Chasse 1993, Briollay et al., 1994; Kumar et al., 1986; Tuovinen 1994; Kunsch 2002) of software tools and aimed CFD-prognoses for particular explored cases of fluid phenomena (Holmsted et al., 1996; Beard et al., 1993).

Modern field-model codes (Zhang 2002) that are engaged in CFD-research within the last decade, supported by powerful hardware, can cope with domains made out of several hundred thousands cells. So the JASMINE (Miles 2006), applied for research on fires in tunnels, is mostly used for mitigation studies assuming simple one-step chemistry for combustion modelling, as the local reaction rate is calculated from a modified version of the eddy break-up mixing model (Magnusen et al., 1976). This approach is suitable for turbulent diffusion flames – one of a characteristic of large-scale fires, where the rate of reaction is controlled by the comparatively slow mixing of fuel with oxidant (air). JASMINE solves the time-averaged (Miles et al., 1999), turbulence-modelled conservation equations for mass, momentum, energy and species, employing a k-ε turbulence model with extra source terms accounting for the effect of buoyancy in turbulent mixing. The integro-differential equations, together with the equations of state for an ideal gas, form here a closed set of coupled equations. These are again discretised and solved on a three-dimensional, finite-volume Cartesian mesh. Identical principles of functioning has TUNFIRE (Miles et al., 2004) as this code was verified by Miles et al. against the experimental data from tests held by Massachusetts Highway Department (Mc Grattan et al., 2002). In the last phase of these tests, based on a general purpose computer program for analysis of fluid flow – the COMPACT-3D, a new CFD-tool was developed (www.tunnelfire.com 2006), the SOLVENT – that can handle both steady and transient flows, where combustion is not modelled, but represented as time-dependent source of heat and mass, since the turbulence is "treated" through buoyancy-augmented k-ε model including variety of boundary conditions and wall functions with modifications for wall roughness.

All of these research attempts (Zhang et al., 2002) that have been brought up into the CFD-community (Leupi 2005), do report on good capability (Westbrook et al., 2006) of the numerical approaches (Jojo et al., 2003) used in handling the reactive flows (Markatos et al., 1982) in straight enclosed traffic objects. Besides the slight denivelation of a few percent, the geometry of the arbitrary tunnels was relative a simple one.

Therefore the aim of the study performed, is the exploration of the (accidental) large-scale fires in such a tunnel that, for sure, turn up as an element of modern traffic road-communications and would have a different geometrical characteristics, different than tunnels explored in mentioned studies.

So, firstly we performed a validation of the used CFD-tool (that applies, mathematical model, and additional numerical step) against the data of the distinguished experiments run (Wehlan 1995). Subsequent, we run the simulation of large-scale fire in a newly constructed tunnel, where possible accidental scenario was presumed. Both results from validation and investigation's outcome found the additional agreement with other experimental approaches as well.

Mathematical model

In this study the flow phenomena are computed by the Reynolds Averaged Navier-Stokes (RANS) equations, with the turbulence k-ε model (Neophytou et al., 2005), representing the major characteristic of the applied CFD-investigation-tool of the FLUENT; and handling the buoyancy by applying the Boussinesq approximation. This approach, is not affected by fluctuation of initial conditions, offering more accurate presentation of the time dependent flow – particularly the distribution of the gaseous combustion products (Gao et al., 2004). Since the investigations have shown that the Mach Number was of the order of 0.022, such a flow can be assumed as incompressible (Peric et al., 2001). So assumed as incompressible, the fluid while crossing the reaction front, doesn't undergo thermal-caused expansion and the reaction makes no impact onto flow-velocity. Further assumption, to have a planar propagation front of combustion in a motionless fluid, leads to the application of the Boussinesq approximation (Baum 1995) without external forces (Vladimirova 2006). Here, the flow velocity obeys the incompressible Navier-Stokes equation with a temperature-dependent force term (Vladimirova 2006).

Numerical approach

For transient simulations (a CFD-mode that was applied in this study) the governing equations must be discretised in both space and time (www.fluent.com 2006, Versteeg et al., 1995). In choosing the numerical method we rely on the standard of the finite volumes (Peric et al., 2001, Versteeg et al., 1995, Hirsch 1988). The spatial discretisation of time-dependent equations employed a segregated solution method. The linearised equations result in a system of linear equations for each cell in the computational domain, containing the unknown variable at the cell centre as well as the unknown values in surrounding neighbour cells.

The estimation of the boundary conditions in this CFD-based investigation was supported by the experience of some previous studies (http://lin.epfl.ch 2005, Muhasilovic 2007). So were tunnel-entrance and exit characterised as open (pressure) boundaries with minor pressure increase or pressure-drop of 10 Pa, respectively. The fuel "pool" – the simulated fire-place, has been determined by the constant max flux rate of 0.101 kg/m^{2}s (Babrauskas 1983) having hydraulic diameter of 4.41 m for a 50 MW-heptane fire (Vela et al., 2006).

Tunnel-entrance as open (pressure) boundary, was used for initializing computational values for the velocity and pressure in the domain since the global temperature was set to the 293 K.

The tunnel housing, tunnel road and tunnel walls as well, were presumed to be adiabatic barriers in the area where the objects of interest (the investigated tunnels) are built (http://lmr.epfl.ch 2007). This decision was based on some previous research experience, but the reality-oriented investigation on modern tunnel-construction knows for the thermal conductivity of a rock where through a tunnel was built. Particularly for the Orgus tunnel, that was built in such area, in the Croatian midlands, where the specific thermal conductivity of limestone (λ=2.3 W/mK) (http://lmr.epfl.ch 2007, www.also-natursteine.de 2007) must be configured in the boundary conditions in further investigations.

Procedure of Investigation

The explored tunnel

The shapes of cross-section of a tunnel can be generally separated into two groups: ones of a shape of a "horse-shoe" and ones of the rectangular shape – according to the final use of this traffic steady-object. What is however important for an investigation of the movement of the air, air-pollutants and gaseous products of an accidental combustion is the aspect ratio, defined as the ration between the width and the height of a tunnel, A$_p$. Most common tunnels that have been built world-wide, have an aspect-ratio of 1 to 1.5. The tunnel, explored in this study belongs to that category.

The tunnel which was object of interest is an active, new-built tunnel binding the midlands with the Croatian coast at the Adriatic Sea near town of Split. It was chosen because of the interesting geometry, made in the road-curve of radius of 2500 m and having its cross section (As=1.24) made out of two incomplete swiped intersecting circles in a shape of a "horse-shoe" (Figure 1). In its length of 354 m, this tunnel is having the longitudinal elevation of 3,2% as well as the achieved lateral torsion of the tunnel tube of 2.5%.

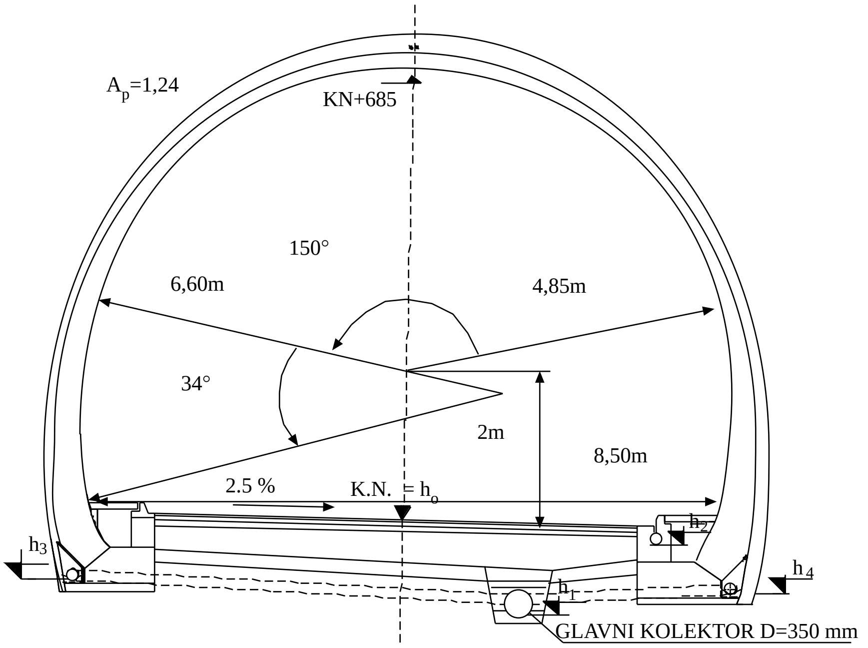

Figure 1. The cross-section of the tunnel on the road D-1 near town of Split in Croatia.

Here we positioned the fire-place (within the computational domain) at the fourth of the tunnel-length, having the heptane in the pool as inflammable goods for simulating the fire of 50 MW-heat-release-rate. As fire source of such heat release served a pool (6.4 m x 3.84 m) of hydraulic diameter of 4.41 m filled with heptane (in computational approach) positioned one and the half meter above the tunnel-road.

The computational domain

The area, in which the computation with applied mathematical model approach and additional numerical discretisation was performed, is the very volume that a fluid can take (an air, or in case of accidental fire, the combustion gaseous products), without the physical walls, i.e. housing of the tunnel. Therefore the computational domain, the Orgus tunnel, the newly built tunnel connecting the Croatian midlands with the Adriatic coast near town of Split is formally called here the Croatian tunnel.

The Croatian tunnel

The mesh of this computational domain (Figure 2) is characterised through hexahedral cells of a random structure. In this case a denser grid was applied in the area around the fire pool, having so more grid-points to support the major fluid-mechanic and thermodynamic occurrences. Such unstructured hexahedral mesh-shape was installed in whole computational domain. However, the following parts of the 354 m long tunnel with surface of the cross-section of 56.80 m^2 are also meshed with unstructured hexahedral cells in the explained way having subsequent increased cells size to 400 mm, 800 mm and 1200 mm as distance from fire pool was growing towards the tunnel-exit and entrance.

The pool with the fuel had a flux rate of 0.101 kg/m^2s^{-1} as well, and the pool-surface temperature is set also to 393 K. Middle-plane of the Tunnel was going throughout the domain following the curvature of this cavity.

The "Croatian" tunnel-housing and the tunnel-road was in the computational domain defined as adiabatic walls as well and fluid-domain is air, with the ambient conditions and no fluid-movement. The entrance as well as the tunnel-exit, are in this case also defined as open-pressure boundaries.

Discussion of the observed phenomena – results from Croatian tunnel

Distinguishing two major groups of the flame-shapes in enclosure: the ones, that impinge on the tunnel roof and the others that, influenced by the longitudinal (even natural) (air)stream, do not; during the

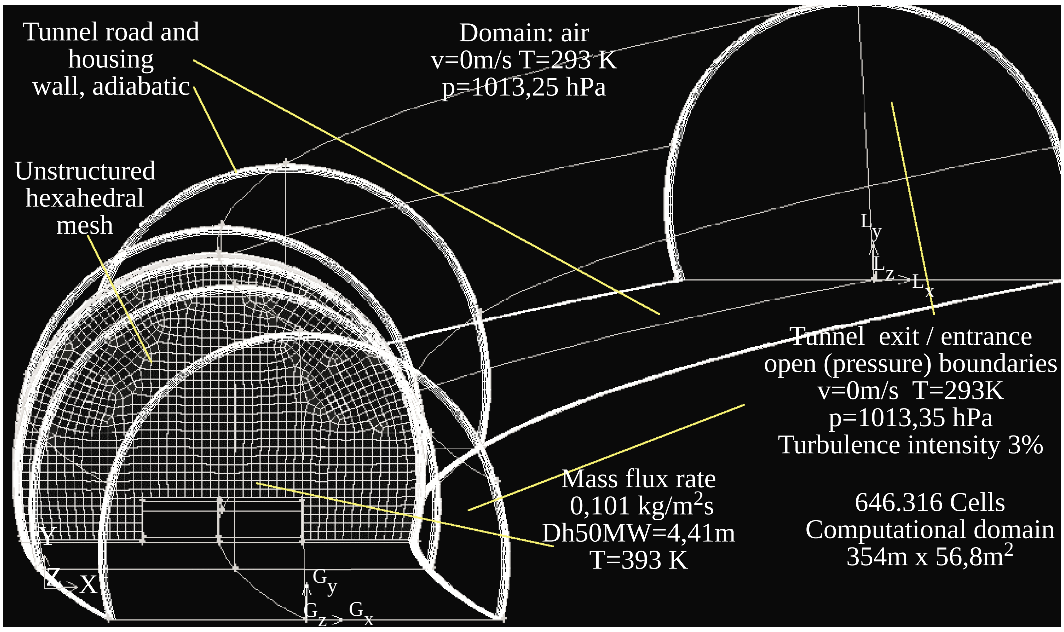

Figure 2. The unstructured hexahedral mesh is applied in the "Croatian" tunnel – here, at one fourth of its length, around the zone of the fuel pool.

CFD-investigations in the Croatian tunnel we recognise the first ones in the results of our research, that cause far stronger damageable impacts on the material.

Defining this phenomena (impinging the tunnel-ceiling), using the definition for the flame tilt angle θ, we apply the suggested approach (Kurioka et al., 2001) onto the results from the Croatian tunnel.

In the first 44 s of the numerically performed investigation, we noticed that the behaviour both of the flame and of the gaseous products of combustion, that was proven experimentally where axis of the flame is pushed away, downstream, because of the relative weak buoyancy, compared to the longitudinal (natural) tunnel ventilation (Koseki et al., 1988; Kurioka et al., 2003). As one moves further of the fire-source, along the distance X_1, the buoyancy gains on the strength due to the increased temperature, as we observe a rising of the flame under the angle θ. Further interesting moment was the reaching of the combustion-time of one minute. The additional agreement with the experimental approaches (Kurioka et al., 2001; Ingason 2007) was observed in this recorded moment of the CFD-simulation as well: after the flame and/or hot current impinged on the tunnel ceiling, heading towards tunnel's higher geodetic position, from the impinged position, the stream of the hot gases is loosing the temperature through the convection (Figure 3). The buoyant effect is becoming weaker and the "head" of this hot gaseous current is falling down and "rolling", finally being mixed with the cooler far-fire-place air (Figure 4) impacting in this way the tunnel road as well (Figure 5).

Additional size of the impact of irradiance and temperature offers a view (seeing from below) of the tunnel ceiling and tunnel-central-plane above the fire-place (Figure 6). On the places with the most developed irradiance one can notice the places with the highest temperature (hotspots), occurred, due to the fluctuations in the combustion.

And this especially is case of constructive interference of the buoyancy with the slow longitudinal natural flow (towards the higher geodetic position, according to the tunnel-slope of 3.2%), wherever the buoyant forces are stronger than convection of the fire plume and the hot gas-stream, which is above the fire-place and near-fire-area. These occurrences do present in this phase of the fire-development already a dangerous point for the construction of the tunnel body (Figure 7).

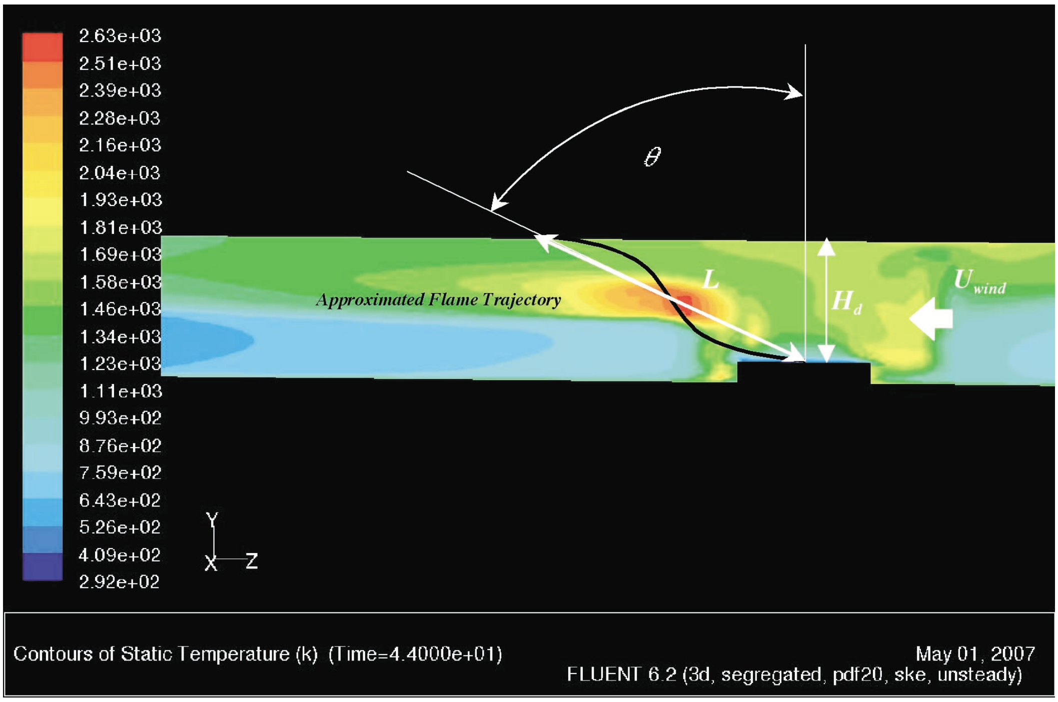

Figure 3. Already on this sketch, recorded at the time t= 44 s is easy to recognise another characteristic phenomenon of the confined large-scale combustion, noticed by Kurioka et al., 2001 as flame impinges the tunnel roof under fire tilt angle, defined as $\cos(\theta) = H_d / L$ (41).

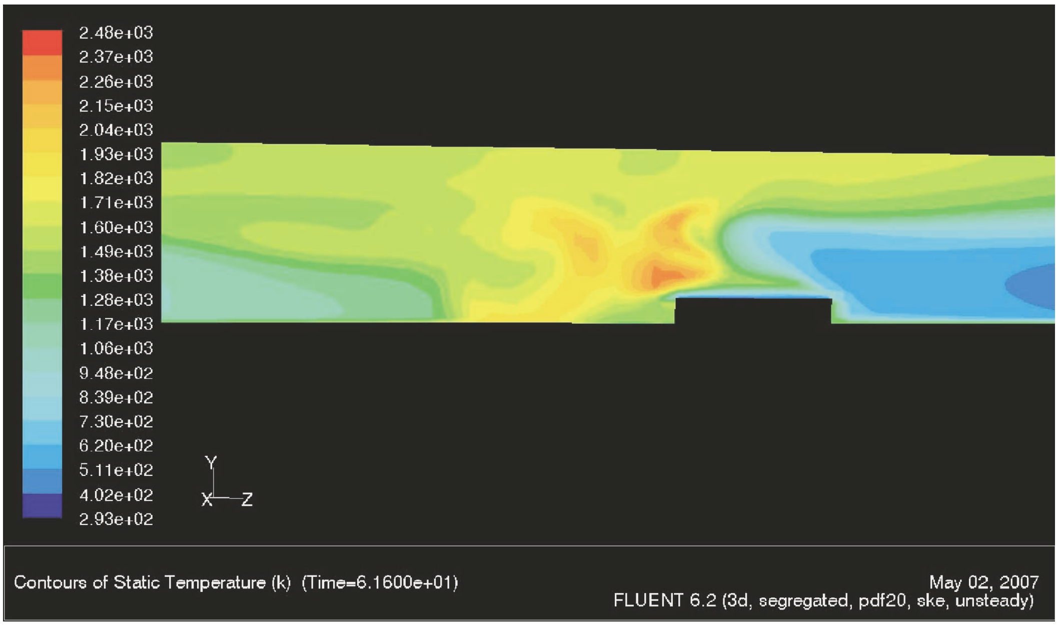

Figure 4. After the transmission of its energy to the surroundings by the convection and irradiance, the hot current of the gaseous products is "rolling" down from the ceiling and finally getting mixed with the cooler fluid (air).

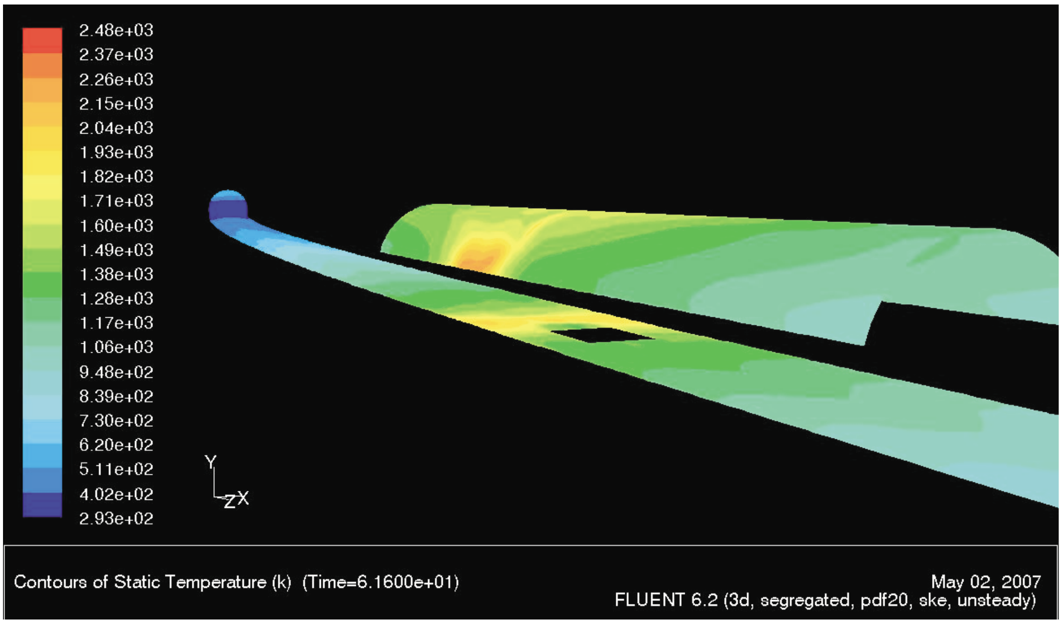

Figure 5. Temperature field as impact on the road due to the effect of "rolling" of the hot gaseous combustion products. On this sketch we see the tunnel entrance with lower geodetic position and section of tunnel ceiling above fire place.

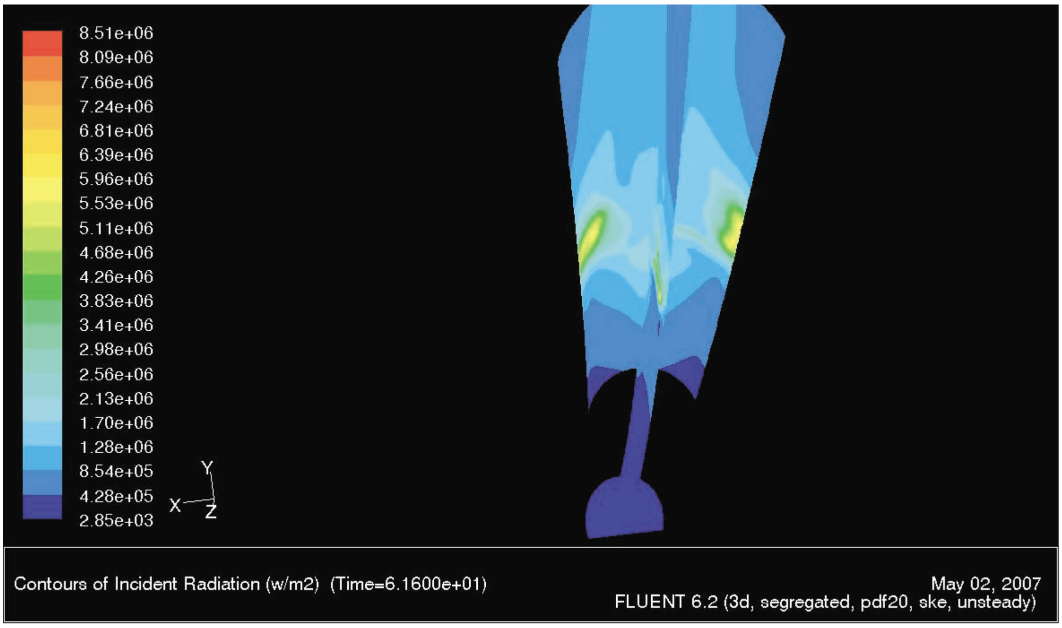

Figure 6. "Hotspots" are visible on the tunnel-central-plane, above the fire-place after first 61 s of the 50 MW-heptane-fire in Croatian tunnel.

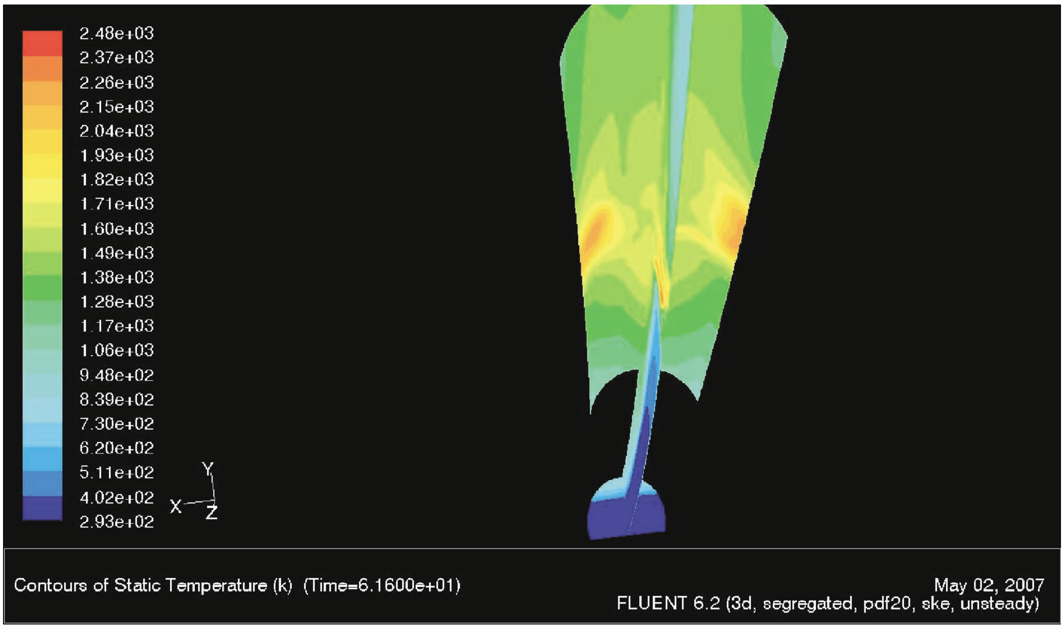

Figure 7. On the bottom of the temperature-field, the upper shape of the tunnel exit is visible.

As one of the characteristics of the large-scale combustion, the shape of the soot-distribution in this recorded moment is complying with the temperature and irradiance fields in the Croatian tunnel (Figure 8).

The curvature of Orgus tunnel did not influence the turbulence as expected where distribution of the turbulent kinetic energy stayed in alignment with the tunnel central plane (Figure 9).

The observation of the heat-radiation impact (Figure 10) has offered the same conclusion, holding no proof for interaction between the cavity curvature and the distribution of the large-scale fire consequences.

The asymmetric cuts (regarding the central tunnel plane) have proven that the homogeneous distribution of the large-scale-fire's consequences hold throughout the "Orgus" tunnel as one is consequently "crossing" tunnel transversally from the left wall to the right one. This occurrence is to be recognised in the temperature field, for instance (Figure 11).

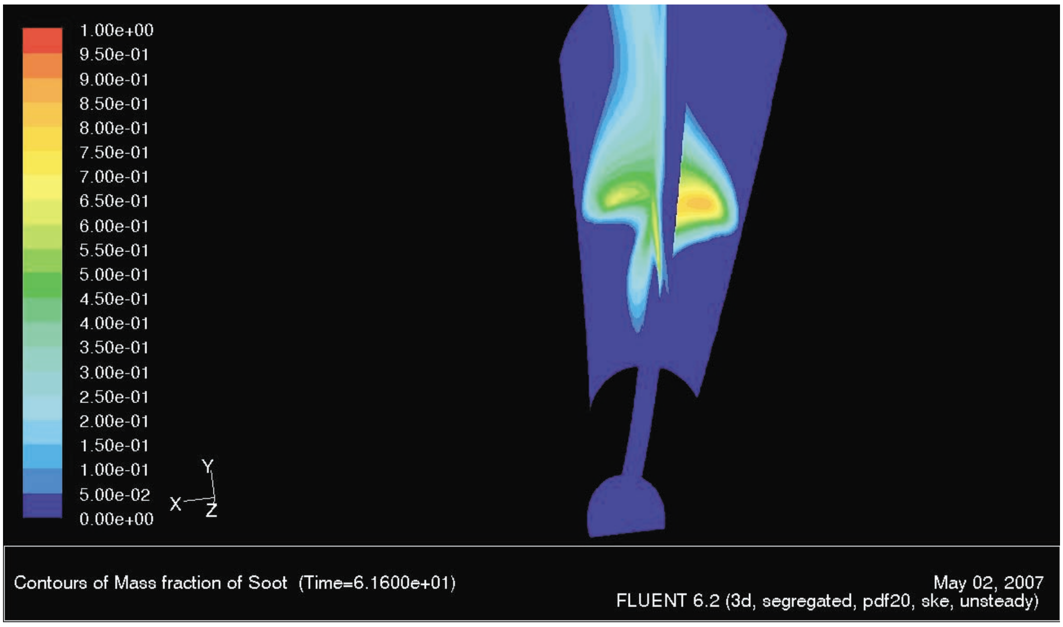

Figure 8. The generation of the soot corresponds to the impacts of the irradiance and the temperature above the fireplace.

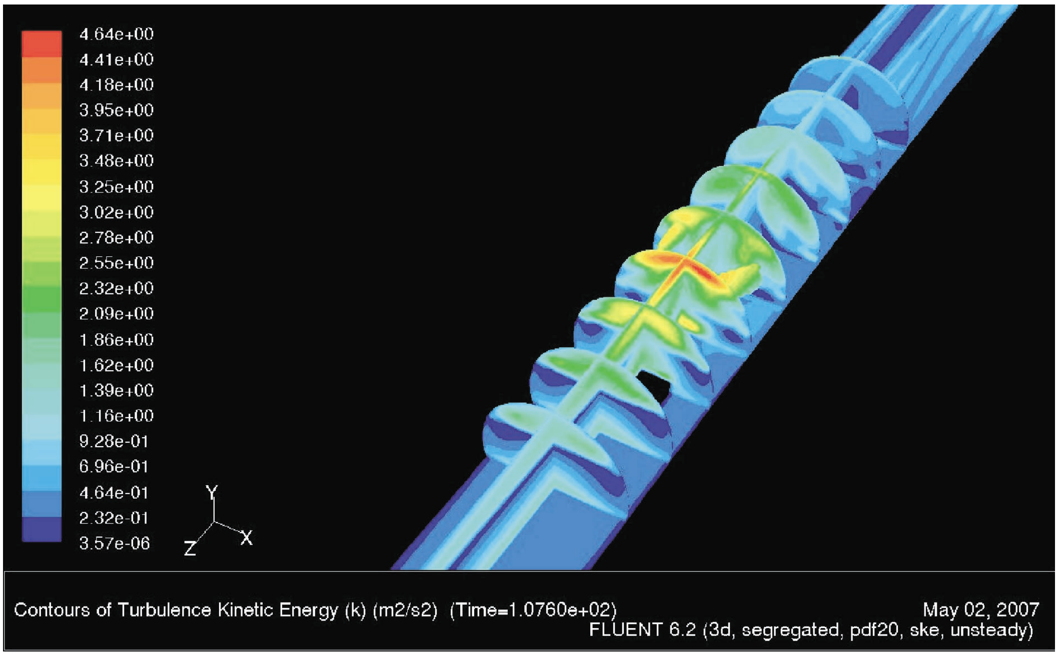

Figure 9. The turbulent kinetic energy in "Orgus" tunnel, sketched on the tunnel cross-sections at the time t=107 s of simulated large-scale accidental combustion around the fire place.

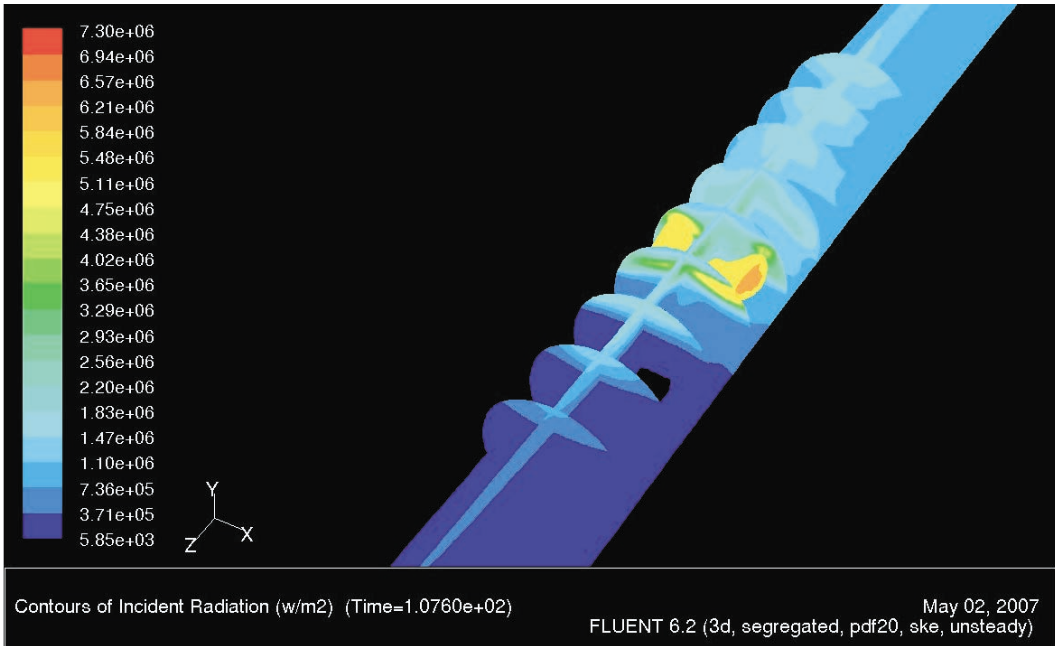

Figure 10. Combined cross-sectional plots with iso-surfaces for temperature (of 1750 °C) and radiation (of 5.84 MW/m^2) pointing to good symmetric distribution, in spite of the curvature.

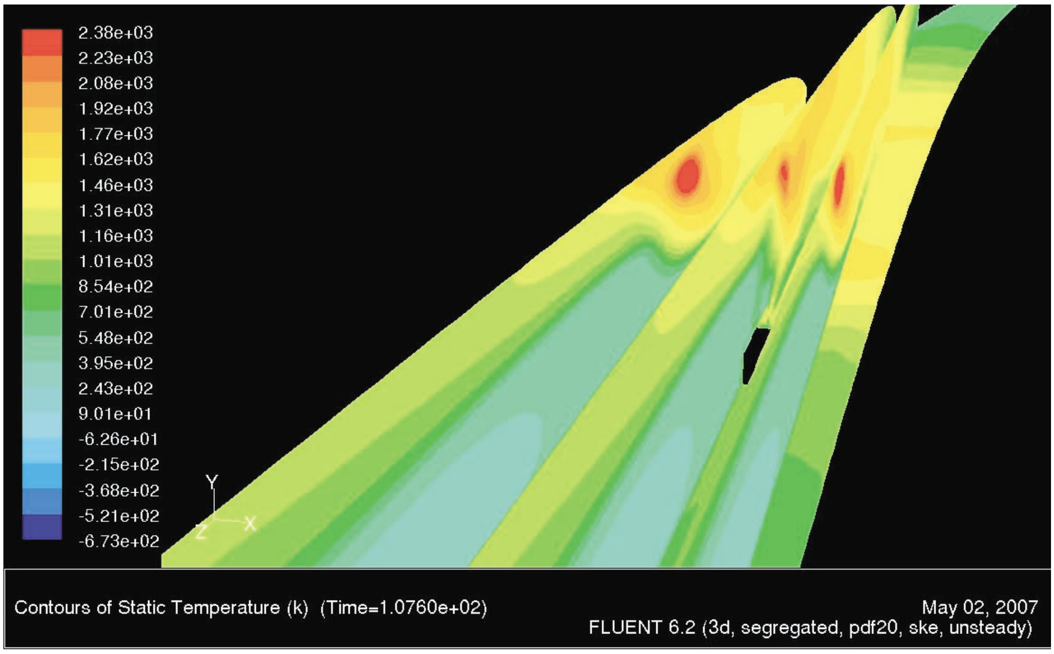

Figure 11. In the appearance of the "hotspots" that occur due to random constructive interference of buoyant flame and natural flow, symmetry is also visible –here, in the temperature fields.

By performing this study on only one object of interest of the given geometric characteristics, the CFD-based investigation on the accidental fire in natural ventilated tunnel "Orgus" (Croatian tunnel) were conducted according to the both standard and novel experimental (Megret 2000, Modic 2003, Vauquelin et al., 2002; Ingason 2007) and computer-aided (Vauquelin et al., 2006; Woodburn et al., 1996; Grant et al., 1998) research (Lee et al., 2006, Carvel et al., 2001). Giving the small mosaic-stone to the entire urge in the community which is researching on large-scale fires, with the results of this research-attempt, we intend to address also the civil-engineering sector (Kodur et al., 2006; Wald 2006) and enlarge data-base for the medical health-protection (Lestari et al., 2006) as well.

The specific geometry of the object of interest –this traffic road-object built in Croatia– was a "provocation enough" to conduct this research, expecting new answers due to the possible impact of a reality-oriented enclosure (computational domain) onto large-scale fire and escorting occurrences.

However, the curvature of the Orgus tunnel was not "strong enough" to influence the propagation of the combustion consequences in the first minute and a half of numerical investigations. So the propagation of the gaseous products has "followed" the tunnel line and expected major asymmetric differences in distribution of the soot, temperature or irradiance (due to the curved cavity), over the cross-section were not noticed.

Acknowledgements

Medzid Muhasilovic thanks to Miss Aida Karahasanovic, Dipl.-Ing. and Mr. Davor Bojanic, Dipl.-Ing. for providing the "blue prints" of the Croatian tunnel "Orgus" that was used in this computational investigation. Mr. Muhasilovic uses this opportunity to thank to his institute-collegues Mr. Mark Soley and Marc-Antoine Habisreutinger for constant support and scientific advices. The gratitude of Mr. Muhasilovic goes in this way to Mr. Michael Wehlan for the accurate and rich information on "West Virginia Memorial Tunnel Tests". Last but definitely not least, M. Muhasilovic's special thank goes to his "sponsor" the Heinrich Hertz Foundation, for understanding, patience, trust and advices during this research endeavour.

References

Babrauskas, V., "Estimating Large Pool Fire Burning Rates". Fire Technology, 251, 1983.

Baum, H. R., et. al. "Gravity-Current Transport in Buildings Fires", Proceedings of International Conference on Fire Research and Engineering 18, 1995.

Beard, A., Drysdale, D., Holborn P. and Bishop S., "Cofiguration Factor for Radiation in a Tunnel as Partial Cylinder". Fire Technology,. 29, 1993.

Briollay, H. And Chasse P., "Validating and Optimazing 2D and 3D Computer Simulations for the Offenegg Tunnel Fire Test", Full report of Centre d'Etudes des Tunnels: Bron Cedex, 1994.

Carvel, R. O., Beard A. N. and Jowitt P. W., "The influence of longitudinal ventilation systems on fires in tunnels" Tunneling and Underground Space Technology, 3, 2001.

Charters, D. A., Gray, W. A. and MacIntosh A. C., "A Computer Model to Assess Fire Hazards in Tunnels (FASIT)", Fire Technology, 143, 1994.

Chasse, P. "Sensitivity Study of Different Modelling Techniques for the Computer Simulation of Tunnel Fire: Comparison with Experimantal Measures". Proceedings of First CFDS International User Conference. 14, 1993.

Gao, P. Z., Liu, S. L., Chow, W. K. and Fong, N. K., "Large Eddy Simulations for Studying Tunnel Smoke Ventilation", Tunneling and Underground Space Technology, 577, 2004.

Gatski, T. B. and Jongen, T., "Nonlinear Eddy Viscosity and Algebraic Stress Models for Solving Complex Turbulent Flows", Progress in Aerospace Sciences, 655, 2000.

Grant, G. B., Jagger, S. F. and Lea, C. J., "Fires in Tunnels", Philosophical Transactions: Methematical, Physical, Engineering Sciences, 2873, 1998.

Heins, T. and Kordina, K., ,,Untersuchungen über die Brand- und Rauchentwicklung in Unterirdischen Verkehrsanlagen - Katastrophenschutz in Verkehrstunneln" Full report Österreichisches Verkehrsministerium, 1990.

Hirsch, C., "Numerical Computation of Internal and External Flows" Manuscript Chichester Brisbane Toronto New York: John Wiley & Sons. 515, 1988.

Holmstedt, G., Bengston, S. and Tuovinen, H., "Sensitivity Calculations of Tunnel fires Using CFD", Fire Safety Journal, 99, 1996.

http://www.also-natursteine.de, Internet site, 2006

http://www.fluent.com, Internet site, 2006

http://lin.epfl.ch, Internet site, 2006

http://lmr.epfl.ch, Internet site, 2007

http://www.tunnelfire.com, Internet site, 2006

Ingason, H., "Correlation Between Temperatures and Oxygen Measurements in a Tunnel Flow", Fire Safety Journal, 75, 2007.

Jojo, S. M. and Chow, W. K., “Numerical Studies on Performance Evaluation of Tunnel Ventilation Safety Systems”, Tunneling and Underground Space Technology, 435, 2003.

Kodur, V. K. R., Bisby L. A. and Green, M. F., “Experimental Evaluation of the Fire Behaviour of Insulated Fibre-Reinforced-Polymer-Strengthened Reinforced Concrete Columns”, Fire Safety Journal, 547, 2006.

Koseki, H. and Yumoto, T., “Air Entrainment and Thermal Radiation from Heptane Pool-Fires”, Fire Technology, 33, 1988.

Kumar, S. and Cox, G., “Mathematical Modelling of Fires in Tunnels” Proceedings of 5th International Symposium on the Aerodynamics & Ventilation of Vehicle Tunnels. 23, 1985.

Kumar, S. and Cox, G., “Mathematical Modelling of Fires in Tunnels - Validation of JASMINE” Full report, Transport and Research Laboratory - Crowthorn, UK, 1986.

Kumar, S. and Cox, G., “Radiation and Surface Roughness Effects in the Numerical Modelling of Enclosure Fires”, Proceedings of Fire Safety Science - 2nd International Conference, 32, 1988.

Kunsch, J. P. “Simple Model for Control of Fire Gases in a Ventilated Tunnel”, Fire Safety Journal, 67, 2002.

Kurioka, H., Oka, Y., Satoh, H., Kuwanna, H. and Sugawa, O., “Properties of the Plume and Near Fire Source in horizontally long and narrow spaces”, Journal of Construction Engineering, 151, 2001.

Kurioka, H., Oka, Y., Satoh, H. and Sugawa, O., “Fire Properties in Near Field of Square Fire Source with Longitudinal Ventilation in Tunnels”, Fire Safety Journal, 319, 2003.

Lee, S. R. and Ryou, H. S., “A numerical Study on Smoke Movement in Longitudinal Ventilation Tunnel Fires for Different Aspect Ratio”, Building and Environment, 719, 2006.

Lestari, F., et al., “An alternative Method for Fire Smoke Toxicity Assessment Using Human Lung Cells”, Fire Safety Journal, 605, 2006.

Leupi, C., “Numerical Modelling of Cohesive Sediment Transport and Bed Morphology in Estuaries”, Full Report, La Faculte Sciences at Techniques de l’ Ingenieur, Ecole Polytechnique Federale de Lausanne: Lausanne, 2005.

Magnussen, B. F. and Hjertager B. H., ”On Mathematical Modelling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion”, Proceedings of 16th International Symposium on Combustion, 76, 1976.

Markatos, N. C. and Malin, M. R., “Mathematical Modelling of Buoyancy-Induced Smoke flow in Enclosures”, International Journal of Heat Mass Transfer, 63, 1982.

McGrattan, K. B. and Hamins, A., “Numerical Simulation of the Howard Street Tunnel Fire” Full Report, NIST, 2002.

Megret, O. and Vauquelin, O, “A Model to Evaluate Tunnel Fire Characteristics”, Fire Safety Journal, 393, 2000.

Miles, S. D., “About JASMINE”, Personal communication, 2006.

Miles, S. D. and Kumar, S., “Computer Modelling to Assess the Benefits of Road Tunnel Fire Safety Measures” Proceedings of Inflam 51, 2004.

Miles, S. D., Kumar, S. and Andrews, R. D., “Validation of a CFD Model for Fires in the Memorial Tunnel”, Proceedings of First International Conference on Tunnel Fires, 43, 1999.

Modic, J., “Fire Simulation in Road Tunnels”, Tunneling and Underground Space Technology, 525, 2003.

Muhasilovic, M., “CFD-Approach in Investigation of Consequences of Accidental Large-Scale Fires in Road Tunnels with Natural Ventilation” Full Report, EPFL: Lausanne, Switzerland, 2007.

Neophytou, M. K.-A. and Britter, R. E., “A Simple Model for the Movement of Fire Smoke in a Confined Tunnel”, Pure and Applied Geophysics, 1941. 2005.

Peric, M. and Ferziger, J. H., “Computational Methods for Fluid Mechanics”, Manuscript, , Berlin Springer Verlag, 423, 2001.

Tuovinen, H., “Validation of Ceiling Jet Flows in a Large Corridor with Vents Using the CFD Code JASMINE”, Fire Technology, 32, 1994.

Vauquelin, O. and Megret, O., “Smoke Extraction Experiments in Case of Fire in a Tunnel”, Fire Safety Journal, 525, 2002.

Vauquelin, O. and Wu, Y., “Influence of Tunnel Width on Longitudinal Smoke Control”, Fire Safety Journal, 420, 2006.

Vela, I., et al., “Scale Adaptive Simulation (SAS) of Heat Radiation and Soot Amount in a Large-Scale Turbulent JP-4 Pool Fire” Proceedings of DECHEMA, 11, 2006.

Versteeg, H. K. and Malalasekera, W., “An Introduction to Computational Fluid Dynamics”, Full manuscript, London Longman Group Ltd. 1995.

Vladimirova, N., “Model Flames in the Boussinesq Limit”, Full report, ASC / Flash Center, Dept. of Astronomy and Astrophysics, The University of Chicago, IL 60637: Chicago, USA. 2006.

Wald, F., et al., “Experimental Behaviour of a Steel Structure Under Natural Fire”, Fire Safety Journal, 509, 2006.

Wehlan, M., "Memorial Tunnel Experiments" , Personal Communication, 2006.

Westbrook, C. K., Pitz, W. J. and Curran H. J., “Chemical Kinetic Modelling Study of the Effects of Oxygenated Hydrocarbons on Soot Emissions from Diesel Engines”, Journal of Physical Chemistry, 6912, 2006.

Woodburn, P. J. and Britter, R. E., “CFD-Simulations of a Tunnel Fire - Part One”, Fire Safety Journal, 35, 1996.

Zhang, W., et al., “Turbulence Statistics in a Fire Room Model by Large Eddy Simulation”, Fire Safety Journal, 721, 2002.