Journal of Naval Science and Engineering
2013, Vol.9, No.1, pp.1-18

THEORETICALLY INVESTIGATION OF SMOKE PRODUCTION IN A TUNNEL FIRE

Asst.Prof. Erinç DOBRUCALI, Naval Lt.

Department of Marine Architecture and Marine Engineering
İstanbul Technical University
edobrucali@yahoo.com

Abstract

Fire is the heat and light energy released during a chemical reaction, in particular a combustion reaction. Depending on the substances alight, and any impurities outside, the color of the flame and the fire's intensity might vary. Fire in its most common form can result in conflagration, and has the potential to cause physical damage through burning. Fires in buildings and transportation systems are a threat to human lives and also to buildings and cultural heritage. In a fire inferno, most of the victims die due to late detection of the fire and consequent inhalation of toxic smoke gases. In addition to the human consequences, the economic and cultural loss is significant.

BİR TÜNEL YANGININDA DUMAN OLUŞUMUNUN TEORİK OLARAK İNCELENMESİ

Özetçe

Yangın özellikle bir yanma reaksiyonunda kimyasal reaksiyon boyunca ısı ve enerji ortaya çıkmasıdır. Yanmış olan maddelere ve dışarıdaki kirliliğe bağlı olarak alevin rengi ve yangının şiddeti değişebilir. En yaygın formda alev/ateş büyük bir yangınla sonuçlanabilir ve büyük fiziksel zararlara sebep olabilecek potansiyeli vardır. Taşıma sistemlerindeki ve yapı/binalardaki yangınlar insan yaşamına olduğu gibi bina ve kültürel mirasada bir tehdit olmaktadır. Bir yangın cehenneminde, yangının geç tespiti dolayısıyla toksit duman gazlarının solunmasıyla dumana maruz kalanların çoğu ölür. Bu insan ölümlerine ilave olarak ekonomik ve kültürel kayıp da çok önemlidir.

Keywords: *Fire, Computational Fluid Dynamics(CFD), field modeling, tunnel fire.*
Anahtar Sözcükler: *Yangın, Hesaplamalı Akışkanlar Dinamiği (CFD), saha modellemesi, tünel yangınları.*

1. INTRODUCTION

1.1. Fire

Fire is the heat and light energy released during a chemical reaction, in particular a combustion reaction. Depending on the substances alight, and any impurities outside, the color of the flame and the fire's intensity might vary. Fire in its most common form can result in conflagration, and has the potential to cause physical damage through burning.

Also, fire, which is conventionally defined as uncontrolled flame spread, is arguably one of the most complex phenomena considered in combustion science. It embraces nearly all the effects found in subsonic chemically reacting flows.

Fluid dynamics, combustion, kinetics, radiation and in many cases multi-phase flow effects are linked together to provide an extremely complex physical and chemical phenomenon.

1.2. Natural Fire Models in Brief

Fire modeling has from time to time proved to be effective for convincing enforcing authorities that chosen solutions are fire safe. It is convenient to solely fix on such models, which can be used in modeling local fires. Nevertheless it is necessary to go over general information on the most common natural fire methods in use. The numerical models that describe fire can be divided in to three large groups:

Stochastic models: In stochastic or probabilistic models fire growth is generally treated as a series of sequential events or states. For instance, an ignition of a litter box impulsed by a burning cigarette followed by the ignition of household curtains caused by the flaming litter box et cetera. These models are sometimes referred to as "states transition" models. The results of stochastic models are estimates about the probabilities of ignition of objects or fire spread.

Empirical models: Empirical models are algebraic equations that estimate fire development based on experience. One of the most common empirical models is the ISO 834 time-temperature curve, which illustrates temperature development of the post-flashover fire.

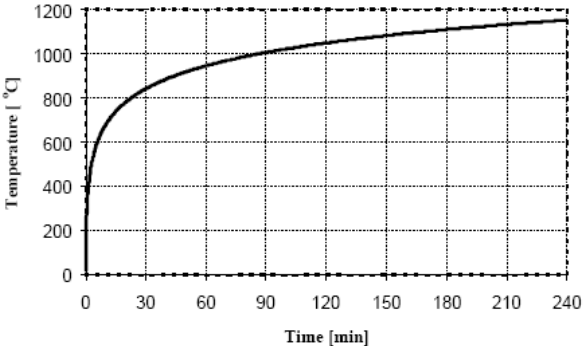


Figure 1.1. ISO 834 time-temperature curve

Deterministic models: Deterministic models are composed with algebraic equations. Solving the equations produces numerical values for the variables describing the fire (variables such as temperature, gas flow and concentration). Furthermore the deterministic models are divided into zone models, field models and network models. The latter is suitable for approximating the spread of fire in complex buildings. Therefore actually two different methods exist for modeling fire in enclosures: the zone model and the field model. They differ primarily in their treatments of the gas phase.

1.3. Fire Behavior

Basically, an enclosure fire may include some or all of the following phases of development. (See Figure 1.2)

Incipient phase: Heating of potential fuel is taking place through a variety of combustion processes such as smoldering, flaming or radiant.

Growth phase: (pre-flashover) Ignition is the beginning of fire development. At the initial growth phase, the fire will be normally small and localized in a compartment. An accumulation of smoke and combustion products in a layer beneath the ceiling will gradually form a hotter upper layer in the compartment, with a relatively cooler and cleaner layer at the bottom. With sufficient supplies of fuel and oxygen and without the interruption of fire fighting, the fire will grow larger and release more hot gases to the smoke layer. The smoke layer will descend as it becomes thicker.

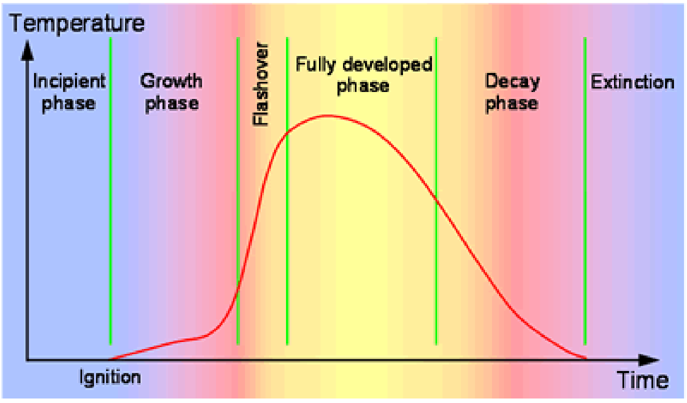


Figure 1.2. Fire Stages

Flashover: In case of fire developing into flashover, the radiation from the burning flame and the hot smoke layer may lead to an instant ignition of unburned combustible materials in the compartment. The whole compartment will be engulfed in fire and smoke.

Fully developed phase: (post-flashover) After the flashover, the fire enters a fully developed stage with the rate of heat release reaching the maximum and the burning rate remaining substantially steady. The fire may be ventilation or fuel controlled. Normally, this is the most critical stage that structural damage and fire spread may occur.

Decay phase: After a period of sustained burning, the rate of burning decreases as the combustible materials is consumed and the fire now enters the decay phase.

Extinction: The fire will eventually cease when all combustible materials have been consumed and there is no more energy being released.

1.4. Fire Modeling

Fire model	**Nominal fires**	**Time equivalences**	**Compartment fires**		**Zone Models**		**CFD / field models**
			Parametric	**Localized**	**One-zone**	**Two-zone**	
Complexity	Simple	Intermediate			Advanced		
Fire Behaviour	Post-flashover fires			Pre-flashover fires	Post-flashover fires	Pre-flashover / localized fires	Complete temperature-time relationships
Temperature distribution	Uniform in whole compartment			Non-uniform along plume	Uniform	Uniform in each layer	Time and space dependent
Input parameters	Fire type No physical parameters	Fire load Ventilation conditions Thermal properties of boundary Compartment size		Fire load & size Height of ceiling	Fire load Ventilation conditions Thermal properties of boundary Compartment size Detailed input for heat & mass balance of the system		Detailed input for solving the fundamental equations of the fluid flow
Design tools	BSEN1991-1-2				COMPF2 OZone SFIRE-4	CCFM CFAST OZone	FDS SMARTFIRE SOFIE
	PD7974-1		PD7974-1				
	Simple equations for hand calculations		Spreadsheet	Simple equations	Computer models		

Table 1. Options for fire modeling.

The models can be classified as either probabilistic or deterministic;
Deterministic models can roughly be divided into three categories:

Field Models , CFD
Zone Models
Hand-calculation Models

Probabilistic models can be divided into three basic categories:

A network model
Statistical modeling
Simulation models

1.4.1. Deterministic Models

Field Models (computational fluid dynamics technique)

The most sophisticated deterministic models for simulating enclosure fires are termed ‘field models’ or ‘CFD models’. Field models of fire development in structures involve directly the fully 3-D time-dependent partial different equations of the fundamental conservation laws, and are used in a wide range of engineering disciplines.

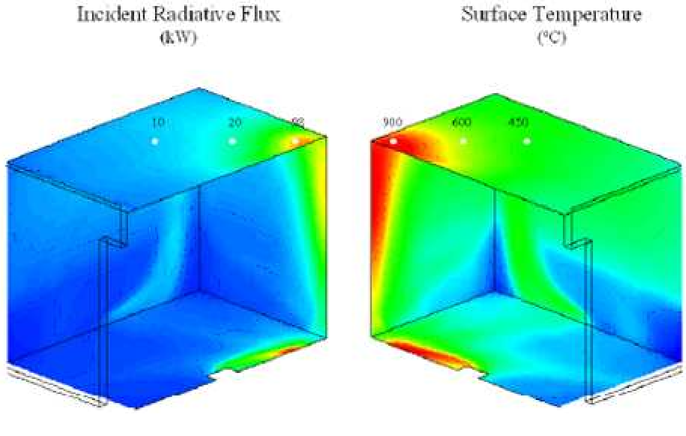


Figure 1.3. Post processed 3D graphics from CFD (field model) software

Equations are solved for mass, momentum, energy and species at each of the many points of a fine grid covering the entire volume of the interior of the structure. In this way, values are permitted to vary from point to point within each compartment, and from compartment to compartment. Thus, these models have the capability of very accurate solution of the equations, and hence accurate simulation of the events.

Zone models

The theoretical background of zone models is the conservation of mass and energy in fire compartments. Basically, the models take into account of rate of heat release of combustible materials, fire plume, mass flow, smoke movement and gas temperatures. They rely on some assumptions concerning the physics of fire behavior and smoke movement suggested by experimental observation of real fires in compartments. The zone models also model the fire compartments in more detail, compared to that for parametric fires and time equivalence methods. The geometry of compartments, as well as the dimensions and locations of openings, can be modeled easily.

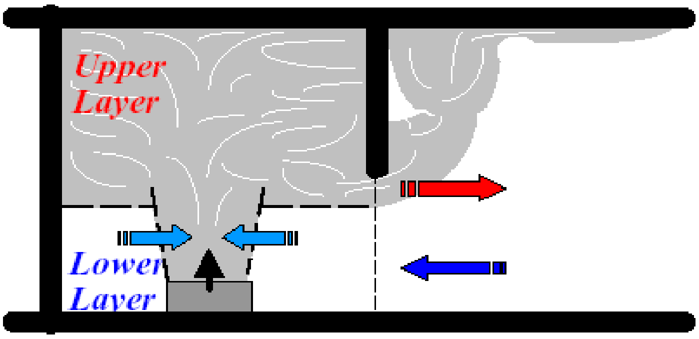


Figure 1.4. Zone Model

Zone models solve the conservation equations for distinct regions. A number of zone models exist, varying to some degree in the detailed treatment of fire phenomena. The dominant characteristics of this class of model are that it divides the compartment(s) into hot upper layers and cold lower layers. The basic assumption of a zone model is that properties can be approximated throughout the zone by some uniform function.

The uniform properties are temperature, smoke, and gas concentrations, which are assumed to be exactly same at every point in a zone. Experimental observations show that the uniform properties zone assumption yields good agreement.

1.5. Tunnel Fire

Fires in buildings and transportation systems are a threat to human lives and also to buildings and cultural heritage. In a fire inferno, most of the victims die due to late detection of the fire and consequent inhalation of toxic smoke gases. In addition to the human consequences, the economic and cultural loss is significant. Computational fluid dynamics (CFD) is nowadays widely used to simulate smoke spread and temperature distribution as well as ventilation measures in a fire scenario. It allows improving fire fighting strategies and precaution measures in order to suppress the fire as fast as possible and help people to evacuate.

In recent years, it has become apparent that computational fluid dynamics (CFD) can play an important and useful role in fire safety problems. The application of CFD to fire problems is also known as field modeling. Field modeling is based on the fundamental laws, which govern the fire phenomena. Therefore, it is a valuable alternative for experimental investigations and empirical correlations and can be applied as a predictive tool. In this paper, field modeling is applied to tunnel fires in order to predict the critical ventilation velocity.

A tunnel fire can be considered as a fire-taking place in a tube with two openings, one at each side of the tube. Typically, the cross-sectional dimensions are much smaller than the length of the enclosure. Above the fire source, the hot combustion products and smoke rise in a plume-like flow due to the effect of buoyancy. Eventually, these gases impinge onto the ceiling and spread out radially. Typically, the sidewalls are quickly reached and afterwards the smoke propagates beneath the ceiling in two opposite directions along the tunnel axis. At a certain distance from the fire source, the flow of hot gases becomes essentially one-dimensional. One of the possibilities to increase the safety of a tunnel is to provide a mechanical ventilation system that creates a longitudinal flow inside the tunnel.

In most tunnel fires, carbon monoxide from products of incomplete combustion represents the principal life threat, and soot formation strongly affects the extent of flame propagation and the associated heat flux. This work has focused on investigations applying CO and soot models in a full-scale wind-aided flame propagation.

2. GEOMETRICAL MODEL

Generally, tunnel geometry is like that;

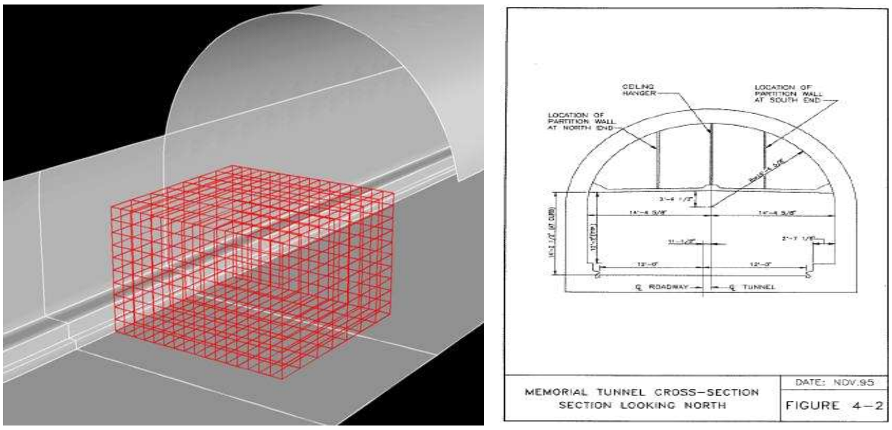
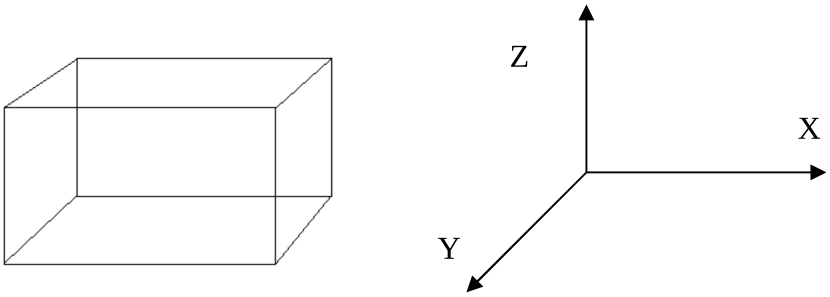


Figure 2.1. Schematic diagram of 3D tunnel.

In this study it is modeled this tunnel as a rectangular prism;



X=L=90m.
Y=W=5.4m.
Z=H=2.4m.

A schematic diagram of this 3D tunnel and its geometry are shown in Fig. 2.2. The walls of the tunnel are made of concrete. The tunnel was modeled as a rectangular prism with length L =90 m. , width W =5.4 m. and height H=2.4m.

(i) 3D
(ii) The walls are made of concrete
(iii)Octane fire (pool fire)
(iv)Uniform grid system
(v) Computational mesh with x: 190 cells, y: 50 cells, z: 34 cells

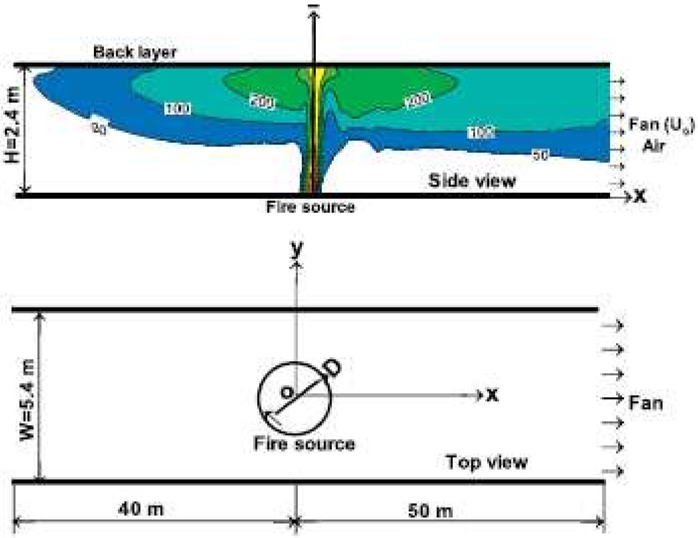


Figure 2.2. Full-scale tunnel fire, and the coordinate system for numerical simulation.

3. PHYSICAL MODEL

A physical model describes the relationship between the main parts which define with geometrical model. In other words, it defines the main physical Fundamentals between the parts that are defined in geometrical model and also it gives the required information about system behaviors by making some assumptions to obtain the mathematical model.

The turbulent combustion process is modeled by an eddy break-up concept by using two sequential, semi-global steps to CO prediction. The numerical model solves three- dimensional, time-dependent Navier–Stokes equations, coupled with sub models for soot formation and thermal radiation transfer. The smoke movement is modeled as an unsteady process, from the time of ignition until convergence to a quasi-steady state.

In this study, it can be separated physical model four different parts; fluid dynamic equations, combustion model, soot formation and its oxidation, radiation model.

3.1. Fluid Dynamic Equations

<u>The Mass Conservation Equation:</u>

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) - S_m \qquad \text{(i)}$$

$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \cdot u_j)}{\partial x_j} = 0$ S_m= 0 (The mass added to the continuous phase from the dispersed second phase)

<u>Momentum Conservation Equations:</u>

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \qquad \text{(ii)}$$

$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \rho g_i = \nabla \cdot \tau_{ij,SGS}$ F=0 External body Force (eg.That arise from interaction with the dispersed phase)

<u>Energy Equation:</u>

$$\frac{\partial \rho \bar{h}}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{h})}{\partial x_j} - \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{Pr_t}\frac{\partial \bar{h}}{\partial x_j}\right) = \dot{q}'''_c - \nabla \cdot q_r \qquad \text{(iii)}$$

q_c''' = Heat Release Rate per Unit volume (oxygen Consumption rate) (from combustion model)

$\nabla \cdot q_r$ = Radiant energy flux (from Radiation model)

3.2. Combustion Model

(i) $$1\ \mathrm{kg\_}C_xH_y + v_1^*\mathrm{kg\_O_2} \rightarrow (28/12)f_c\ \mathrm{kg\_CO} + (18/2)(1-f_c)\mathrm{kg\_H_2O}$$ ………

$$\dot{\omega}_{\mathrm{C_xH_y}} = \frac{\mathrm{d}\rho Y_{\mathrm{C_xH_y}}}{\mathrm{d}t} = -\rho\tau_{\mathrm{mix}}\min\left(Y_{\mathrm{C_xH_y}}, \frac{Y_{\mathrm{O_2}}}{v_1^*}\right)$$

Reactant

(ii) $$1\ \mathrm{kg\_CO} + v_2\ \mathrm{kg\_O_2} \rightarrow (1+v_2)\mathrm{kg\_CO_2}$$ ……..

$$\dot{\omega}_{\mathrm{CO}} = \frac{\mathrm{d}\rho Y_{\mathrm{CO}}}{\mathrm{d}t} = -\rho\tau_{\mathrm{mix}}\min\left(Y_{\mathrm{CO}}, \frac{Y_{\mathrm{O_2}}}{v_2}\right)$$

Products

$$\Downarrow$$

$$\dot{\omega}_{\mathrm{O_2}} = v_1^*\omega_{\mathrm{C_xH_y}} + v_2\dot{\omega}_{\mathrm{CO}} \quad \Rightarrow \quad \dot{q}'''_c = -H_0\dot{\omega}_{\mathrm{O_2}}$$

$\mathbf{H_0}$=13100kJ/kg

These are **eddy-dissipation models** : Reaction rates are assumed to be controlled by the turbulence, so expensive Arrhenius chemical kinetic calculations can be avoided. The model is computationally cheap, but, for realistic results, only one or two step heat-release mechanisms should be used.

Most fuels are fast burning, and the overall rate of reaction is controlled by turbulent mixing. In non-premixed flames, turbulence slowly convects/mixes fuel and oxidizer into the reaction zones where they burn quickly. In premixed flames, the turbulence slowly converts/mixes cold reactants and hot products into the reaction zones, where reaction occurs rapidly. In such cases, the combustion is said to be mixing-limited, and the complex and often unknown, chemical kinetic rates can be safely neglected.

An eddy-dissipation concept is used to formulate interaction between combustion, soot and turbulence. With such a model, it is possible to address the important issue of CO and soot production for the under ventilated tunnel fires. Large eddy simulation (LES) for the fluid dynamic equations of 3D elliptic, reacting flow is coupled with soot production and radiation models.

3.3. Soot formation and its oxidation

In a heavily sooting flame, soot particles generated from combustion processes have a significant impact on the radioactive heat transfer characteristics. Various detailed soot formation models would require accurate knowledge of fuel composition (C_2H_2, C_6H_5, C_6H_6, OH,etc.). However, coupling between LES and the detail chemistry to track the soot development stages in relation to its formation and destruction is not available due to the prohibitive computation cost for large-scale fire simulation. In this work, the soot formation and its oxidation are incorporated in to a turbulent flow calculation in two convection–diffusion equations for the precursor particle density, n, and soot concentration, C_s. This model assumes that soot is formed from a gaseous fuel in two stages, where the first stage represents the formation of radical nuclei, and the second stage represents soot particle formation from these nuclei.

The two-step Tesner model predicts the generation of radical nuclei and then computes the formation of soot on these nuclei. Fluent thus solves transport equations for two scalar quantities: the soot mass fraction and the normalized radical nuclei concentration:

(i) The first stage represents FORMATION of Radical Nuclei

$$\frac{\partial}{\partial t}(\rho b^*_{\mathrm{nuc}}) + \nabla \cdot (\rho \vec{v} b^*_{\mathrm{nuc}}) = \nabla \cdot \left(\frac{\mu_t}{\sigma_{\mathrm{nuc}}} \nabla b^*_{\mathrm{nuc}}\right) + \mathcal{R}^*_{\mathrm{nuc}}$$

PS: $b^{*}_{nuc} = \bar{n}$

↕

$$\mathcal{R}^{*}_{\mathrm{nuc}} = \mathcal{R}^{*}_{\mathrm{nuc,form}} - \mathcal{R}^{*}_{\mathrm{nuc,comb}}$$

$\mathcal{R}^{*}_{\mathrm{nuc,form}}$ = rate of nuclei formation (particlesx 10^{-15} /m ^{3}-s)

$\mathcal{R}^{*}_{\mathrm{nuc,comb}}$ = rate of nuclei combustion (particlesx 10^{-15} /m ^{3}-s)

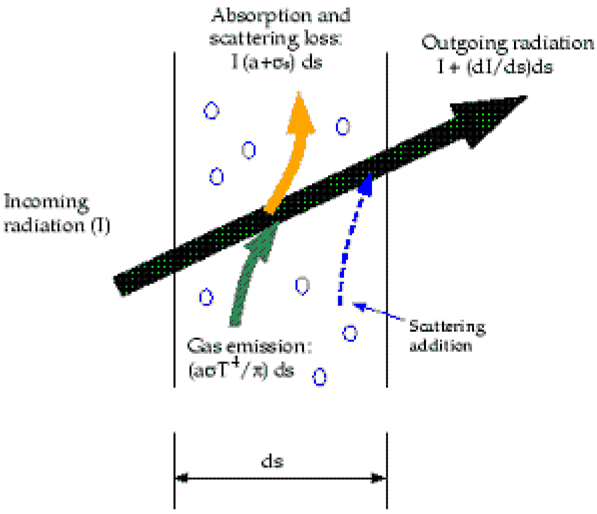
(ii) The second stage represents SOOT PARTICLE FORMATION from these nuclei.

$$\frac{\partial}{\partial t}(\rho b^{*}_{\mathrm{nuc}}) + \nabla \cdot (\rho \vec{v} b^{*}_{\mathrm{nuc}}) = \nabla \cdot \left(\frac{\mu_t}{\sigma_{\mathrm{nuc}}} \nabla b^{*}_{\mathrm{nuc}} \right) + R_{soot}$$

PS: $b^{*}_{nuc} = C_{s}$ ↕

$$\mathcal{R}_{\mathrm{soot}} = \mathcal{R}_{\mathrm{soot,form}} - \mathcal{R}_{\mathrm{soot,comb}}$$

3.4. Radiation Model



$$\frac{dI(\vec{r},\vec{s})}{ds} + (a+\sigma_s)I(\vec{r},\vec{s}) = an^2\frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\vec{r},\vec{s}\,')\,\Phi(\vec{s}\cdot\vec{s}\,')\,d\Omega'$$

↓ ↓ ↓

$\sigma_s = 0$ n=1 =0 Without scattering

▶ $\vec{\nabla}\cdot\vec{\Omega}I + \kappa I = \kappa\frac{\sigma T^4}{\pi}$

▶ $-\nabla\cdot q_r = -\int_{4\pi}\vec{\nabla}\cdot\vec{\Omega}I\,d\Omega \approx \kappa\left(\sum_{l=1}^{L} w^l I^l - 4\sigma T^4\right)$ It can be directly substituted into energy equation.

4. CONCLUSION

A tunnel fire can be considered as a fire-taking place in a tube with two openings, one at each side of the tube. Typically, the cross-sectional dimensions are much smaller than the length of the enclosure. Above the fire source, the hot combustion products and smoke rise in a plume-like flow due to the effect of buoyancy.

Eventually, these gases impinge onto the ceiling and spread out radially. Typically, the sidewalls are quickly reached and afterwards the smoke propagates beneath the ceiling in two opposite directions along the tunnel axis. At a certain distance from the fire source, the flow of hot gases becomes essentially one-dimensional. One of the possibilities to increase the safety of a tunnel is to provide a mechanical ventilation system that creates a longitudinal flow inside the tunnel.

In recent years, it has become apparent that computational fluid dynamics (CFD) can play an important and useful role in fire safety problems. The application of CFD to fire problems is also known as field modeling. Field modeling is based on the fundamental laws, which govern the fire phenomena. Therefore, it is a valuable alternative for experimental investigations and empirical correlations and can be applied as a predictive tool.

In this paper, smoke production in a tunnel fire is investigated theoretically. But smoke production must be studied practically. An application will be presented regarding investigation on prediction of smoke production in a tunnel fire by using a computer simulation.

REFERENCES

[1] Kevin B. McGrattan, Anthony Hamins, 2001, Numerical Simulation of the Howard Street Tunnel Fire, Fire Research Division Building and Fire Research Laboratory.

[2] A.R.Nilsen, T.Log, 2009, Results from three models compared to full-scale tunnel fires tests Fire Safety Journal 44 (2009) 33– 49.

[3] J. Collazo, J. Porteiro *, D. Patiño, J.L. Miguez, E. Granada, J. Moran, 2009, Simulation and experimental validation of a methanol burner, Fuel 88 (2009) 326–334.

[4] I.S. Lowndesa, S.A. Silvestera,, D. Giddingsb, S. Pickeringb, A. Hassanb, E. Lester, 2007, The computational modelling of flame spread along a conveyor belt, Fire Safety Journal 42 (2007) 51–67.

[5] F. Liu, J.X. Wen, 2002, The effect of turbulence modelling on the CFD simulation of buoyant diffusion flames, Fire Safety Journal 37 (2002) 125–150.

[6] D. Rusch, L. Blum, A. Moser, T. Roesgen, 2008, Turbulence model validation for fire simulation by CFD and experimental investigation of a hot jet in crossflow, Fire Safety Journal 43 (2008) 429–441.

[7] J.P. Kunsch, 2002, Simple model for control of fire gases in a ventilated tunnel, Fire Safety Journal 37 (2002) 67–81.

[8] C.C. Hwang_, J.C. Edwards, 2005, The critical ventilation velocity in tunnel fires—a computer simulation, Fire Safety Journal 40 (2005) 213–244.

[9] Y. Wu, M.Z.A. Bakar, 2000, Control of smoke Flow in tunnel fires using longitudinal ventilation systems a study of the critical velocity, Fire Safety Journal 35 (2000) 363}390.

[10] Sherman C.P. Cheunga, Richard K.K. Yuena,, G.H. Yeohb, Grace W.Y. Cheng, 2004, Contribution of soot particles on global radiative heat transfer in a two-compartment fire, Fire Safety Journal 39 (2004) 412–428.

[11] Karim Van Maele, Bart Merci, 2008, Application of RANS and LES field simulations to predict the critical ventilation velocity in longitudinally ventilated horizontal tunnels, Fire Safety Journal 43 (2008) 598–609.