International Journal of Mechanics and Energy (IJME)
Vol. 2, Issue 4, 2014, ISSN: 2286-5845

CFD simulations on CO production in a longitudinally ventilated tunnel fires for different aspect ratio

Soufien Gannouni *, Rejeb Ben Maad
Faculty of Sciences of Tunis, Department of Physics, Laboratory of Energizing and Thermal and Mass Transfer, 2092 ElManar, Tunis, TUNISIA

Abstract: In a tunnel fire, the production of smoke and toxic gases remains the principal factors prejudicial to users. More than 80 % of deaths are in direct relation to the toxicity and the opacity of smoke. In this paper, the smoke propagation and CO production in a longitudinally ventilated tunnel fires for different aspect ratio is analyzed numerically with large eddy simulation (LES) by Fire dynamic simulator (FDS). The evolution of the CO concentration longitudinally along the tunnel will be studied in this work. The tunnel slope's effect on the CO concentration will also be studied. The FDS Numerical results are firstly verified by comparison with the experimental measurements of Li et al. [15-16], in terms of the critical velocity and the maximum smoke temperature under the ceiling. A relatively good agreement has been obtained. Results show that the CO concentration increases with the aspect ratio while this latter remains lower than 1 and then decreases as the aspect ratio increases for an aspect ratio greater than 1.

Key words: Tunnel fire, CO concentration, aspect ratio, CFD.

1. Introduction

The road and railway underground transport systems have undergone an important evolution to facilitate the passage of the mountainous regions and to preserve the environment in urban zones. Managers and manufacturers are faced with not only the question of the people and property security but also to public confidence in the tunnels as sure means of transport.

The fires which develop in these underground transport infrastructures have attracted a particular attention because of their disastrous consequences, such as the calamity of the Mont Blanc tunnel (39 deaths on March 24th, 1999) or the sinister criminal of the subway of Daegu Korea (198 deaths on February, 2003) [1]. Smoke is the most mortal factor in fire case where a huge quantity of toxic gases is released as a result of an incomplete combustion [2]. Moreover, the augmentation of temperature may trigger serious damage on the tunnel structure what entails long and expensive work for the reconstruction, besides its deleterious impacts on the economics of the entire regions.

Numerous studies and investigations have concentrated on the analysis of the smoke dynamics and the physical phenomena involved in the tunnel fire situation. The Studies of tunnel fires can be produced using different techniques: full-scale tests, experimental models on reduced scales and CFD numerical modeling. The diversity of investigation methods of tunnel fire problems and the richness of research are conducted to better understand the fire behavior and improve existing safety systems. Indeed, fire safety in the underground infrastructure to use public has become an important issue at the same time human and political.

The ventilation is the key element to guarantee a correct evacuation of users, contain the fire and control the dispersion of contaminants in tunnel [3]. In normal operating conditions, it serves to reduce the levels of pollutants in tunnel. During a fire, it helps to limit the smoke dispersion in tunnel. The strategy of emergency planning and smoke control most currently used in

*** Corresponding author:** Soufien Gannouni
E-mail: gannounisoufien@gmail.com.

majority of tunnels is based on the longitudinal ventilation produced mechanically [4]. The minimum ventilation velocity, generally designated by the name of critical velocity is simply the velocity to contain the smoke from one side. Below this velocity, a smoke layer goes against the current. This layer is known as backlayering. One of the foremost criteria for the design of longitudinal ventilation system of tunnel is the critical velocity value [5]. Currently, several different formulas to predict the critical velocity were obtained through on the theory based on the Froude number preservation combined with some experimental data of small or full scale [6-8].

However, the principal dangers to the users in a fire case are the toxicity and the opacity of the smoke. L.H. Hu et al [9] performed a full scale experiments in an 88 m long corridor and a large eddy simulation (LES) by fire dynamic simulator (FDS). They showed that the concentration of carbon monoxide increases linearly with height above the floor and decreases exponentially with distance away from the fire source. By performing a numerical study by using a large eddy simulation in a longitudinally ventilated tunnel fire, HY Wang [10] noted, for a ventilation velocity below a critical velocity, the back layer flow against the ventilation airflow carries the CO and soot production towards the tunnel entry. An inverse dependence of the backflow length to ventilation velocity is predicted. J. Zhang et al [11] have studied the tunnel's inclination effect of the CO concentration and the smoke movement by carrying in a reduced scale experiments in an inclined tunnel with the inclination angles varying to five different degrees, 0°, ± 5° and ± 10°. They found that the CO concentration is negatively correlated with the inclination angle and the thickness of the smoke layer and the smoke outflow rate are both positively correlated with the tunnel slope.

In this work, a three-dimensional numerical simulation of a tunnel model fire is performed using an internationally recognized CFD code of the fire simulation (FDS). The purpose of this study is to numerically analyze the influence of aspect ratio, A_r, on smoke propagation and toxic gas generation. The FDS results are firstly checked by comparison with experimental data available in the literature. This validation work is developed to study the effect of the tunnel geometry on the production and the distribution of the carbon monoxide concentration, longitudinally, in the tunnel. The effect of the tunnel slope, S, on the CO concentration will be also studied. The CFD results are presented and analyzed.

2. Problem formulation and CFD simulation

2.1 Problem Formulation and numerical model simulation

The problem considered consists of the study of a fire in a 12 m long horizontal tunnel with different cross-sectional geometry. The cross section of tunnel is rectangular, with a different aspect ratio. The aspect ratio is defined as tunnel height divided by its width. Five aspect ratios are considered in this study: A_r = 0.5 (tunnel A); 0.67 (tunnel B); 1 (tunnel C); 1.5 (tunnel D) and 2 (tunnel E) and the tunnel slope, *S*, is set at 0, 2.5 and 5%. To simulate the fire, a rectangular heat source having a section area of 8.1cm x 9.7 cm is used. This heat source is placed at the centre of the domain and in the middle of the two sidewalls, with its top surface set at floor level. At the tunnel entrance, a longitudinal air flow is assigned. A schematic representation of considered geometry is shown in Fig. 1. A series of measuring stations is placed at 1 cm below the tunnel ceiling to monitor the temperature and the CO concentration variation. The positions of the measuring points of temperature and CO concentration are shown in Fig. 2.

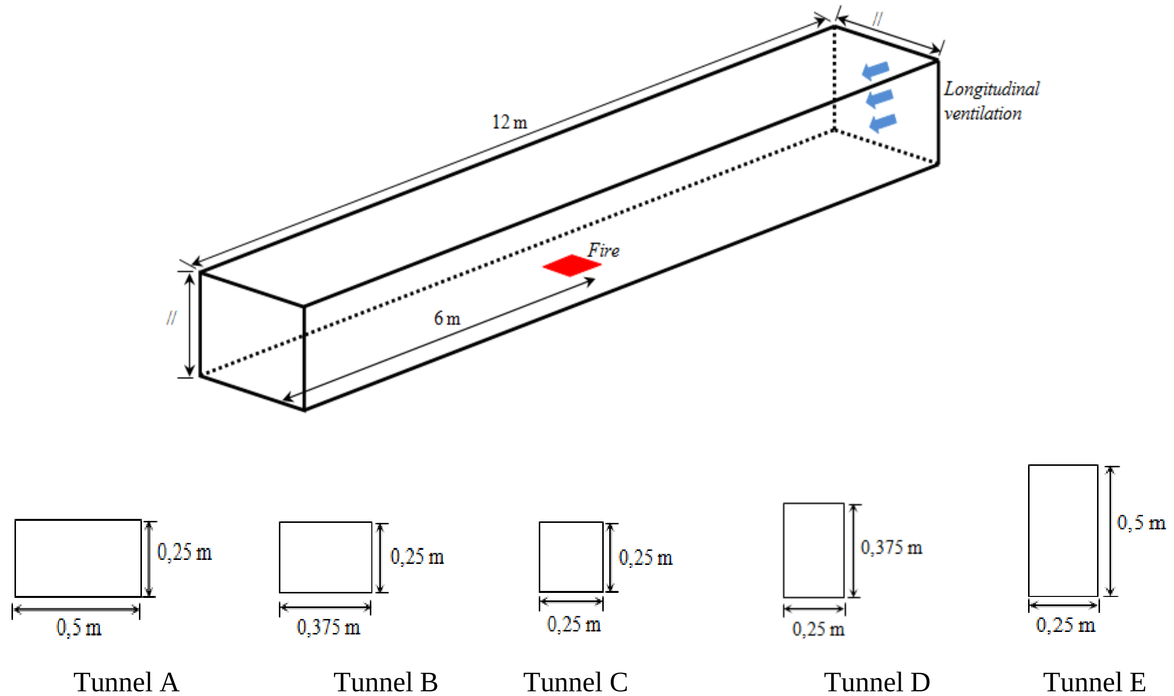

Fig. 1 Schematic of the tunnel model and the cross-sections

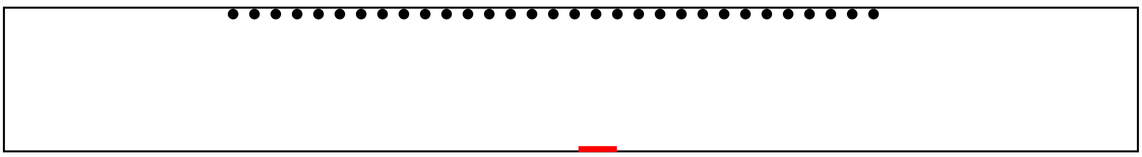

Fig. 2 Positions of the measuring points of temperature and CO concentration

2.2 CFD Simulation

2.2.1 FDS code

A CFD field code developed by NIST (National Institute of Standards and Technology, USA) [12-13] called FDS (Fire Dynamic Simulator) is used to simulate the same process. This tool is a computer program dedicated to study and to simulate the dynamics and combustion of fires in domestic or industrial confined buildings and to evaluate the efficiency and performance of the protection means against problems related to fire, such as the distribution of temperature and soot concentration. FDS solves numerically a form of the Navier-Stokes equations appropriate for low-speed, thermally-driven flow with an emphasis on smoke and heat transport from fires [12].

These equations used in this study are given here [12]:

Mass equations

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

Momentum equations

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}\right) + \nabla p = \rho g + \mathbf{f} + \nabla \cdot \tau \tag{2}$$

Energy equations

$$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot \rho h \mathbf{u} = \dot{q}''' - \nabla \cdot q_r + \frac{Dp}{Dt} + \nabla \cdot k \nabla T + \nabla \cdot \sum_l h_l (\rho D)_l \nabla Y_l \tag{3}$$

Species equations

$$\frac{\partial(\rho Y_l)}{\partial t} + \nabla \cdot \rho Y_l \mathbf{u} = \nabla \cdot (\rho D)_l \nabla Y_l + \dot{m}_l''' \quad (4)$$

Equation of state

$$p = \frac{\rho RT}{\overline{W}} \quad (5)$$

With $\boldsymbol{u}$ is the velocity vector, ρ is the density, p is the pressure, $\boldsymbol{f}$ is the external force vector, g is the acceleration of gravity, τ is the stress tensor, h is the specific enthalpy, is the heat release rate per unit volume, q_r is the radiative heat flux vector, T is the temperature, k is the thermal conductivity, Y_l is the mass fraction of lth species, **"!!"** is the production rate of lth species per unit mass, R is the universal gas constant and **"!!"** is the average molecular weight.

These equations are discretized on a series of connected recti-linear meshes by using the centered finite difference method of second order and in time using an explicit predictor-corrector second-order Runge-Kutta scheme. The turbulence is modeled by default by the technique of Large Eddy Simulation (LES). It is possible that the user can perform a Direct Numerical Simulations (DNS).

LES model with Smagorinsky subgrid type are retained in this study. The approach of the large eddy simulation is to calculate the large scale eddies those which are carrying of the most part of the turbulent kinetic energy and model the small scales dissipative processes (viscosity, thermal conductivity, and material diffusivity) than those that are explicity resolved on the numerical grid [12]. The modeling is based on the assumption of the turbulent viscosity or subgrid viscosity. The Smagorinsky model assumes that the subgrid viscosity can be considered as a dissipative phenomenon and modeled as tel. This model defines the subgrid turbulent viscosity, μ_t, in the following way:

$$\mu_t = \rho (C_s \Delta)^2 \left(2\overline{\mathbf{S}}_{ij} : \overline{\mathbf{S}}_{ij} - \frac{2}{3} (\nabla \cdot \overline{\mathbf{u}})^2 \right)^{\frac{1}{2}} \quad (6)$$

Where $\overline{S}_{ij}$ is the symmetric rate of strain tensor which is defined by:

$$\overline{S}_{ij} = \frac{1}{2} \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \quad (7)$$

And Δ is the filter width in LES and C_s is the Smagorinsky constant and taken equal to 0.2 the default value used in FDS.

The other diffusive parameters, the turbulent thermal conductivity k_t and material diffusivity $D_{l,t}$, are related to the turbulent viscosity by:

$$k_t = \frac{\mu_t Cp}{\Pr_t} \quad (8)$$

$$(\rho D)_{l,t} = \frac{\mu_t}{Sc_t} \quad (9)$$

Where Pr_t is the turbulent Prandtl number and Sc_t is the turbulent Schmidt number both assumed constant with the default value in FDS is equal to 0.5.

In FDS, a combustion model based on a mixture fraction concept is used for large-eddy simulation. This model assumes that the combustion reaction depends on the stoechiometric coefficients specified of the mixture of fuel, oxygen and yields of soot. The fraction of fuel mass converted into carbon monoxide, y_{CO}, is linked to the soot yield, y_s, via the correlation developed by par Koylu et Faeth [14]. In this model a radiation model is added based on the radiative transfer equation (RTE) and a pyrolysis model obeying a simple Arrhenius law. The RTE is solved using a finite volume technique (FVM).

2.2.3 Mesh and numerical stability

The most critical point of a large eddy simulation is probably related to the size and quality of calculation grid. This is a key parameter to be considered carefully at first since it plays an important role in the precision of the FDS simulation results. With a grid size chosen carefully, LES gives reasonable results. Typically, tests of grid sensitivity must be carried out until we do not see significant differences in our results. Specifically, the determination of the grid size is important for the field near to an important gradient place as for example near the site of the fire.

Table 1 Summary of conditions for all simulations tests

Test n°	Heat release rate (kW)	Ambient temperature (°C)	Longitudinal ventilation velocity (m/s)
1	4.3	20.4	0.70
2	4.3	20.4	0.59
3	4.3	20.4	0.36
4	4.3	20.4	0.08
5	5.2	20.8	0.69
6	5.2	20.8	0.59
7	5.2	20.8	0.34
8	6.0	20.8	0.72
9	6.0	20.8	0.66
10	6.0	20.8	0.58
11	6.0	20.8	0.48
12	1.9	20.3	0.13

McGrattan et al. [13] have suggested that for simulations implying buoyant plumes, how a measure of the field flow is resolved, is given by the dimensionless expression $D^*/\delta x$ where D^* is a characteristic fire diameter given by

$$D^* = \left(\frac{Q}{\rho_0 C_p T_0 \sqrt{g}} \right)^{2/5} \quad (10)$$

With Q is the heat release rate, ρ_0 is the density at ambient temperature, C_p is the specific heat capacity and T_0 is the ambient temperature.

McGrattan et al. [13] proved that a mesh size of about 10% of the characteristic diameter is acceptable to guarantee a reliable operation of FDS. In our study, the computation field is limited by the tunnel walls. The mesh size is chosen after performing grid sensitivity tests and the results showed that the grid distributions were sufficient. A total grid number of 500000 cells are finally used. The mesh is unstructured and is refined in the longitudinal direction near the heat source in the region covering the zone from 0.5 m upstream to 0.5 m downstream of the fire source.

To justify the stability of numerical scheme and to estimate the convergence calculation, the courant-Friedrichs-Lewy (*CFL*) criterion is used in FDS. This criterion is related to the convective terms of the governing equation. It is very important for the large-scale calculations where convective transport dominates diffusive. The estimated velocities are tested at each time step, δt , to ensure that the *CFL* condition is satisfied:

$$\delta t \max \left(\frac{|u_{ijk}|}{\delta x}, \frac{|v_{ijk}|}{\delta y}, \frac{|w_{ijk}|}{\delta z} \right) < 1 \quad (11)$$

The initial time step size is specified automatically in FDS by dividing the size of a mesh cell by the characteristic velocity of the flow. The default value of time step is:

$$\frac{5(\delta x \delta y \delta z)^{\frac{1}{3}}}{\sqrt{gH}} \quad (12)$$

During the calculation, the time step change and constrained by the convective and diffusive transport speeds by ensuring that the CFL condition is satisfied at each time step. The time step will eventually change into a quasi-stationary value when the fire environment has reached a quasi-steady state. The CFL condition which requires that the CFL number is less than or equal to 1 is satisfied with the CFL number varied generally in a range from 0.74 to 0.99 for all simulations. The simulation is performed up to 400 s. The time step size is adjusted by the solver itself. The time step in the simulations was in a range between 0.002 and 0.03 s with an average value of around 0.003 s.

2.2 Boundary conditions

A heat release rate per unit area is applied to the source surface (command "HRRPUA" provided in FDS) and is treated as an average value. The heat release rate of the fire is changed into three different values extend from 4.3 to 6.0 kW. Table 1 lists the summary of conditions of the numerical simulations tests. To produce smoke, a reaction type "Propane" was indicated according to FDS reaction data base. The longitudinal air flow is represented by a profile speed established in the tunnel entrance and in exit; a pressure condition is applied by the "OPEN" condition provided by FDS. The ambient temperature away from fire is approximately about 20 °C. The tunnel walls are considered smooth and thermally thick. The material in the FDS model is made of "stainless steel". The walls are heated by the radiative and convective heat transfer from the surrounding gas. Heat transfer to the wall is calculated in FDS by indicating the physical properties of the material (thermal conductivity, density and specific heat) by the command "MATL".

3. Results and discussion

3.1 Critical velocity and maximum smoke temperature under the ceiling

Numerical simulations of a longitudinally ventilated tunnel fires are carried out by using the 5.5 version of FDS. Firstly and to verify the use of FDS in tunnel fire problems and to validate our numerical simulations, our results obtained in the case of the tunnel of cubic cross-section are compared with the experimental results of Li et al. [15-16]. Li et al [15-16] carried out small-scale experimental tests series in a stainless steel model tunnel of 12 m of long, 0.25 m of width and 0.25 m of height. A circular porous bed burner was located at the tunnel center line, with its top surface set at floor level. Propane was used as fuel and a ventilation flow rate was metered at the entrance of the tunnel by a vortex flow meter for enabling a longitudinal ventilation velocity. In this study, to simulate the same process, a rectangular heat source having a section area similar to the circular heat source area employed in the experiments of Li et al. [15-16] is used. Scenarios of tunnel fires similar to those reported in ref [15-16] are simulated with FDS. In Fig. 3, the dimensionless critical ventilation velocity is plotted against the dimensionless heat release rate. These two dimensionless parameters were defined by the equations (13) and (14), respectively, by using the tunnel height as the characteristic length.

$$V_{cr}^* = \frac{V_{cr}}{\sqrt{gH}} \tag{13}$$

$$Q^* = \frac{Q}{\rho_0 C_p T_0 g^{1/2} H^{5/2}} \tag{14}$$

We can see that the results predicted by FDS and of the experiment of Li et al [15] seem reasonably in acceptable agreement. The predicted results are a little higher than that of the experimental results obtained by Li et al. [15]. This can be probably attributed to the difference in the configurations of heat source fire used in current CFD simulation and experiment of Li et al. [15]. The circular heat source used in the experiments is modeled as a square source.

The maximum smoke temperatures predicted by FDS under the tunnel ceiling are compared with those measured experimentally by Li et al [16] as shown in Fig. 4. We can also see that there is a good agreement between the results, suggesting that the FDS code is applicable for simulations of tunnel fire and confirming the possibility to use this code in tunnel applications.

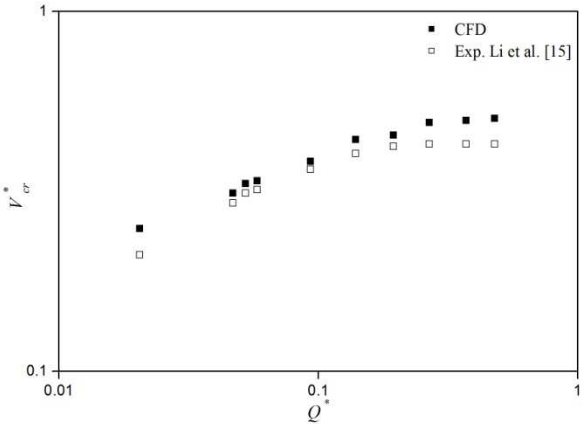

Fig. 3 Comparison of critical velocity predicted by FDS with that measured by Li et al [15] (A_r=1)

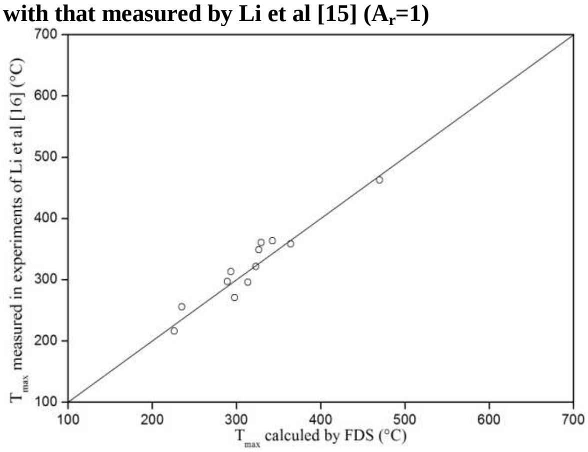

Fig. 4 Comparison of maximum temperature calculated by FDS with those measured in experiment of Li et al [16] (A_r=1)

2.4 Carbon monoxide concentration

During a fire in tunnel, the majority of deaths are in relation primarily with the products released in smokes where an enormous quantity of toxic gases (for example CO) is generated because of an incomplete combustion. In this section, the effect of the aspect ratio of tunnel on the CO concentration will be studied. As illustration, the profiles of the CO concentration as a function of the longitudinal distance to the 1.9 kW fire is presented in the tunnel median plane for different aspect ratios in Fig. 5. It can be seen that the profiles are very similar in shape. The profiles show high values of the CO concentration in the zone close to the heat source and as one move away from the fire, the profiles flatten and CO concentration undergoes a decrease in upstream and downstream of the heat source and it is maintained practically at an almost constant value along the tunnel length. It can be clearly seen in Fig. 5 that the CO concentration is sensitive to the aspect ratio variations. We recall that the aspect ratio is defined as the ratio of the tunnel height to its width. An increase in the CO concentration is observed when the aspect ratio increases while remaining below 1. However, for an aspect ratio greater than 1, it is observed that the CO concentration decreases as the aspect ratio increases. This is probably due to the action of the buoyancy force which entrains an increase in the disruption of the smoke layer with fresh air having as a result more fresh air into the smoke layer and thus lowers the CO concentration that will be oxidized to CO_2. Indeed, during the development of the fire in the tunnel, the fluid particles and the resulting combustion products having a decreasing density with temperature, become lighter and begin to rise up toward the ceiling under the effect of the buoyancy force. They cause a significant entrainment of ambient air by the flow smoke plume. At positions away from the heat source, the levels of CO decreases under a continuous dilution effect causing the oxidation of CO to CO_2.

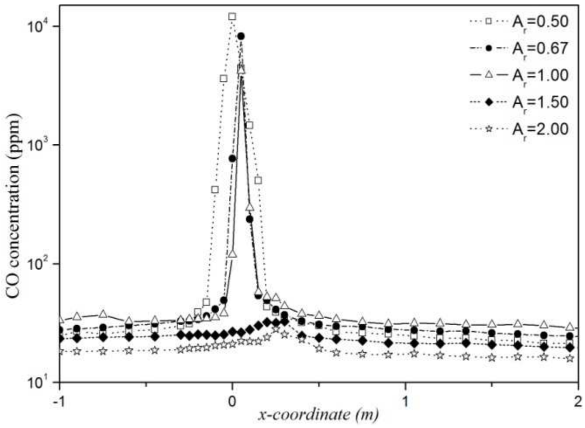

Fig. 5 Comparison of CO concentration with aspect ratio (test n°12)

Fig. 6 shows the CO concentrations measured under ceiling for the 4.3 kW fire for different ventilation velocity for the tunnel C (test n°1-4). This figure shows that an increase in ventilation velocity is correlated to a decrease of CO concentration. Indeed, a significant ventilation velocity having for result an increase in the oxygen concentration and thus improving the mixing between the oxygen and the fuel which facilitates the combustion process which leads to a decrease in the CO concentration.

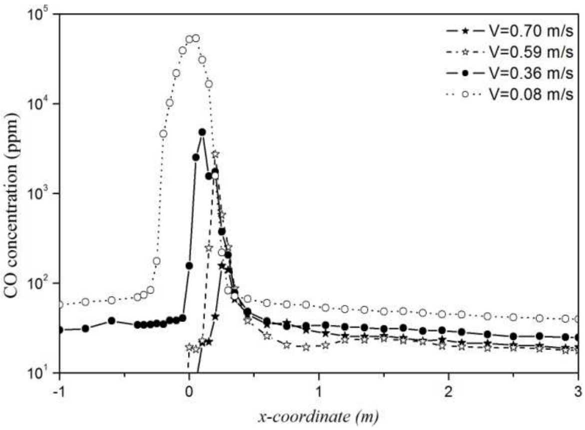

Fig. 6 Comparison of CO concentration with ventilation velocity (A_r=1, test n°1-4)

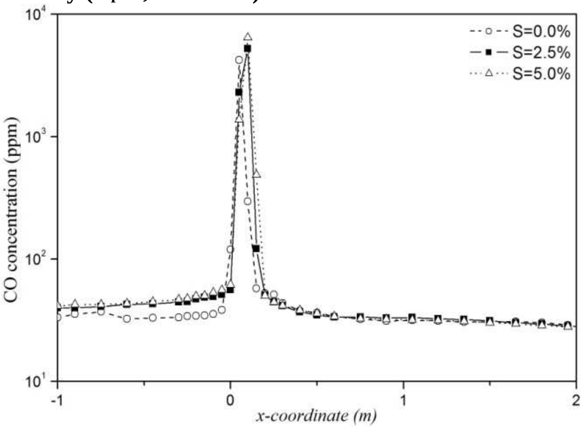

Fig. 7 Comparison of CO concentration with tunnel slope (A_r=1, test n°12)

The slope of tunnel has a significant influence on the smoke dispersion. Because of buoyancy, smoke tends to move in the upward direction: it is the chimney effect. In Fig. 7 is presented the profiles of the evolution of the CO concentration as a function of the distance to the source for different slopes for a heat release rate of 1.9 kW (test n°12). The comparisons of profiles show that they are very similar in shape. These profiles show relatively high levels of CO concentration near the heat source. At positions further away from the fire, the CO concentration decreases and is stabilized at a value almost constant along the length of the tunnel. Fig. 7 shows that the effect of the tunnel slope on the CO concentration usually results in an increase in the CO concentration near the heat source as the slope increases.

4. Conclusions

In this study, CFD simulations were carried out to study the effect of the aspect ratio on the smoke propagation and the CO production in longitudinally ventilated tunnel fires. The CFD simulations results were firstly verified by experimental results of Li et al [15-16]. The critical velocity and the maximum smoke temperature under the ceiling seem reasonably in good agreement with the experimental values. Results predicted by numerical simulations show that the increase in the aspect ratio, the ventilation velocity or of the tunnel slope is correlated with a reduction in the CO concentration.

The tunnel cross-sectional geometry affects the CO production and an increase aspect ratio correlates with a decrease in the CO concentration for an aspect ratio greater than 1. For an aspect ratio lower than 1, the CO concentration increases as the aspect ratio increases. Also, the results show that the CO concentration is negatively correlated with the longitudinal ventilation velocity. In addition, there are also an influence of the tunnel slope on the CO concentration.

Nomenclature

Ar *Aspec ratio*, –
H *tunnel height*
Q *heat release rate*, kW
Q^* dim*ensionless heat release rate*, –
S *tunnel slope*, %
V *longitudinal ventilation velocity*, $m.s^{-1}$
V_{cr} *critical velocity*, $m.s^{-1}$
V_{cr}^* dim*ensionless critical velocity*, –

References

[1] A. Mos, Modèles phénoménologiques pour la simulation d'incendies en tunnel routiers, Thèse de Doctorat, Université Claude Bernard, Lyon I, 2005.

[2] L.H. Hu, R. Huo, Y.Z. Li, H.B. Wang, W.K. Chow, Full-scale burning tests on studying smoke temperature and velocity along a corridor. Tunn. Undergr. Sp. Technol., 20 (2005) 223-229.

[3] K.C. Tsai, H.H. Chen, S.K. Lee, Critical ventilation velocity for multi-source tunnel fires. J. Wind Eng. Ind. Aerodyn., 98 (2010) 650-660.

[4] K. Kang, Characteristic length scale of critical ventilation velocity in tunnel smoke control. Tunn. Undergr. Sp. Technol., 25 (2010) 205-211.

[5] K.C. Tsai, Y.P. Lee, S.K. Lee, Critical ventilation velocity for tunnel fires occurring near tunnel exits. Fire Saf. J., 46 (2011) 556-557.

[6] P. Thomas, The movement of smoke in horizontal passages against an air flow. F. R. S. Note No. 723, Fire Research Station, 1968.

[7] Y. Oka, G.T. Atkinson, Control of Smoke Flow in Tunnel Fires. Fire Saf. J., 25 (1995) 305-322.

[8] Y. Wu, M.Z.A. Bakar, Control of smoke flow in tunnel fires using longitudinal ventilation systems - a study of the critical velocity. Fire Saf. J., 35 (2000) 363-390.

[9] L.H. Hu, N.K. Fong, L.Z. Yang, W.K. Chow, Y.Z. Li, R. Huo, Modeling fire-induced smoke spread and carbon monoxide transportation in a long channel: Fire Dynamics Simulator comparisons with measured data, J. of Hazardous Materials, 140 (2007) 293–298.

[10] H.Y.Wang, Prediction of soot and carbon monoxide production in a ventilated tunnel fire by using a computer simulation, Fire Saf. J., 44 (2009) 394–406.

[11] J. Zhang, X. Zhou, Q. Xu, L. Yang, The inclination effect on CO generation and smoke movement in an inclined tunnel fire, Tunn. and Undergr. Sp. Technol., 29 (2012) 78–84.

[12] K. McGrattan, S. Hostikka, J. Floyd, H. Baum, R. Rehm, W. Mell, R. McDermott, Fire Dynamics Simulator (Version 5): Technical Reference Guide. NIST Special Publication 1018-5, p.124, 2010.

[13] K. McGrattan, R. McDermott, S. Hostikka, J. Floyd, Fire Dynamics Simulator (Version 5): User's Guide. NIST Special Publication 1019-5, p.246, 2010.

[14] U.O. Koylu, G.M. Faeth, Carbon monoxide and soot emissions from liquid-fueled buoyant turbulent diffusion flames, Combust. Flame, 87 (1991) 61–76.

[15] Y.Z. Li, B. Lei, H. Ingason, Study of critical velocity and backlayering length in longitudinally ventilated tunnel fires. Fire Saf. J. 45 (2010) 361-370.

[16] Y.Z. Li, B. Lei, H. Ingason, The maximum temperature of buoyancy-driven smoke flow beneath the ceiling in tunnel fires. Fire Saf. J., 46 (2011) 204-210.