J. Wind Eng. Ind. Aerodyn. 158 (2016) 61–68

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/indaer

Numerical analysis of smoke dispersion against the wind in a tunnel fire

Soufien Gannouni*, Rejeb Ben Maad

Faculty of Sciences of Tunis, Department of Physics, Laboratory of Energizing and Thermal and Mass Transfer, ElManar, 2092 Tunis, Tunisia

ARTICLE INFO

Keywords:
Tunnel fire
Smoke dispersion
Ventilation, Backlayering
FDS

ABSTRACT

In a tunnel fire, the dispersion of smoke in the opposite direction to the longitudinal imposed air flow remains the principal factors prejudicial to users which are blocked upstream of the fire. This phenomenon, usually referred to as “backlayering”, is a key parameter that one must better understand to ensure proper evacuation of users and emergency intervention. This paper carried out full-scale numerical simulation using Fire Dynamic simulator (FDS) to analyze the behavior of backlayering flow in a longitudinally ventilated tunnel fire. FDS predicted backlayering lengths are compared to calculated values using the Hu model. A reasonably good agreement has been obtained. Furthermore, the predicted maximum smoke temperatures are compared to those given by the Kurioka model. CFD results show that the backlayering flow takes CO towards the tunnel entry, which can be deadly to users immediately even at low concentrations. The inertia and buoyancy forces produced by ventilation and fire, respectively, affect the interface height. The backlayering arrival time increases with longitudinal ventilation velocity while it decreases with heat release rate of the fire. An equation is developed to predict the backlayering front arrival time against the distance to fire.

1. Introduction

In tunnel, fires have been identified among the most feared risks. The main hazards are in number three: smoke opacity which leads to loss of visibility, toxicity of produced gases, and high temperatures that can cause structural collapse of the infrastructure (Zhao et al., 2015). The presence of all these physical phenomena complicates safety operations and may indeed threaten the health of users and even firefighters.

Since 1950, the study of tunnel fires has improved due to the real need to achieve technical objectives in the area of safety while preserving some pragmatic limits imposed by technical specifications. Studies of fires can involve various techniques. Full-scale tests, which often require expensive costs and important means of investigations. Small-scale experimental models of two types: i.e. cold models where the heat source is represented by an injection at ambient temperature of a fluid with different density from that of the surrounding fluid and hot models where the fire is represented by a heat source such as a gas burner or a thermal resistor. Unfortunately, full similarity cannot be completely reproduced for all parameters for small-scale models. Finally, a third technique is by CFD numerical modeling. This technique which is based on numerical models can be more advantageous than the experimental methods in many ways such as the richness of the quantitative results, low cost and rapid turnaround time. However, such numerical models require validation due to alternate choices for turbulence modules, combustion alternatives, etc.

During tunnel fires, the critical velocity is one of the main criteria for the design of longitudinal ventilation systems (Tsai et al., 2010). Below the value of a critical velocity, a smoke layer phenomenon called backlayering results in spreading in fumes and combustion moving against the direction associated with the current airflow. This phenomenon which will form upstream of the fire can endanger persons that may already be blocked by the heat source. Generally, the governing parameters for the backlayering length are the heat release rate of fire, the longitudinal ventilation velocity, the tunnel geometry, the air density, the ambient temperature, the thermal capacity of air, and the gravitational acceleration. To take account of all these settings, Li et al. (2010) demonstrated a relationship between the ratio of ventilation velocity to the critical velocity at the same heat release rate and the dimensionless backlayering length which follows an exponential law. However, a simpler equation for predicting the backlayering length was reported by Vantelon et al. (1991) and Hu et al. (2008):

$$\frac{L_b}{H_{fc}} \approx C_b \cdot \left[\frac{gQ}{\rho_a C_p T_a V^3 H_{fc}}\right]^{0.3} \tag{1}$$

where H_{fc} is the vertical distance from the bottom of the fire source to the tunnel ceiling.

By assuming the exponent in Eq. (1) is 1/3 instead of 0.3, Ingason (2008) determined the proportionality constant based on a scale model

* Corresponding author.
E-mail address: gannounisoufien@gmail.com (S. Gannouni).

http://dx.doi.org/10.1016/j.jweia.2016.09.009
Received 10 February 2016; Received in revised form 15 May 2016; Accepted 12 September 2016

Nomenclature

A	cross sectional area, m^2
C_b	constant defined in Eq. (1), dimensionless
C_p	specific heat capacity, kJ kg^{-1} K^{-1}
C_s	smagorinsky constant, dimensionless
D	diffusivity coefficient, m^2 s^{-1}
D'	characteristic fire diameter, m
Fr	Froude number, dimensionless
g	gravitational acceleration, m s^{-2}
H	tunnel height, m
H_{fc}	vertical distance from the bottom of the fire source to the tunnel ceiling, m
h_{in}	interface height, m
h'_{in}	dimensionless interface height, dimensionless
k_t	turbulent thermal conductivity, W m^{-1} K^{-1}
L_b	backlayering length, m
Pr_t	turbulent Prandtl number, dimensionless
Q	dimensionless heat release rate, dimensionless
Q^*	dimensionless heat release rate, dimensionless
Sc_t	turbulent Schmidt number, dimensionless
S_{ij}	symmetric rate of strain tensor, s^{-1}
T	temperature, °C
T_a	ambient temperature, °C
T_{bf}	backlayering front arrival time, s
T'_{bf}	dimensionless backlayering front arrival time, dimensionless
T_s	simulation time, s
ΔT_{max}	Maximum excess smoke temperature, °C
$\mathbf{u}$	velocity vector, m.s^{-1}
V	longitudinal ventilation velocity, m.s^{-1}
V_m	mesh volume, m^3
X	longitudinal coordinate, m
X'	dimensionless longitudinal coordinate, dimensionless
Z	mixture fraction, dimensionless

Greek symbols

δx	grid size in direction X, m
μ_t	turbulent viscosity, kg m^{-1} s^{-1}
ρ	density, kg m^{-3}
ρ_O	ambient density, kg m^{-3}
Δ	filter width in LES, m

tests. Ingason (2008) proposed a value of C_b varying between 0.6 and 2.2 with an average value of 1.4. Hu et al. (2008) conduct full-scale tunnel fire tests to examine the sensitivity of the backlayering length to the ventilation velocity and fire size as well as to the tunnel height. Their results indicated that the backlayering length decreases with the increase in longitudinal ventilation velocity, and increases with fire size. Furthermore, Hu et al. (2008) suggested that the values of C_b deduced from scale models were much lower. Based on their full-scale results, they proposed an average value for C_b equal to 7.4. In their experimental study, Yang et al. (2006) showed that a decrease of ventilation velocity results in an increase in the speed of the smoke flow or backlayering velocity and an enlargement of the layer thickness and its length. Wang (2009) indicated that the backlayering spreads along the ceiling against the air current by carrying the CO and soot production towards the tunnel entry.

According to such earlier studies, a better understanding of the backlayering behavior are still needed to perform a reliable estimation of the arrival time of the backlayering front. Thus, numerical simulation tests using Fire Dynamic Simulator program are presented her for estimating the effects of longitudinal ventilation velocity and heat release rate of fire on backlayering arrival time.

2. CFD modeling

The numerical simulations are performed using the fire dynamic simulator (FDS) software in a tunnel of 300 m of length as shown in Fig. 1. The tunnel walls limit the numerical domain. The cross section of tunnel is rectangular with a 10 m of width and 7 m of height. The tunnel model is made of “concrete”. The physical properties of this material (thermal conductivity, density and specific heat) are specified in the FDS model by the “MATL” command. The values used for the calculation are a thermal conductivity of 1.65 W/m K, density of 2500 kg/m^3 and specific heat of 0.88 kJ/kg K. The tunnel surfaces (walls, ceiling and floor) are thermally thick and smooth. The default velocity condition at the wall surface provided by FDS was accepted. The boundary conditions for the velocity components at the grid points closest to the wall are related with the wall shear stress components according to Werner–Wengler model (Werner and Wengle, 1991; McGrattan et al., 2010a). The two surfaces of the tunnel extremities are both open to the external ambient environment, but the tunnel entry is specified as an airflow inlet for longitudinal ventilation. The longitudinal ventilation velocity is set up by a supply air condition at ambient temperature introduced in the tunnel entry surface. The ambient temperature of the tunnel domain is prescribed via the TMPA parameter provided by FDS with values set about 20 °C in the series of tests simulations. The fire is simulated by a rectangular heat source having a section area of 2 m×2 m. This source is placed at the tunnel center and in the middle of the two sidewalls, with its top surface being set at floor level. The heat release rate of the fire is set for alternate calculations among into three values 4, 10 and 20 MW. To produce simulated smoke, a reaction type “CRUDE-OIL” is applied.

The fire dynamic simulator (FDS) software is a CFD field code developed by NIST (National Institute of Standards and Technology, USA) (McGrattan et al., 2010a). FDS presents a simple turbulence model based on the technique of large eddy simulation (LES) which is built on a good approximation of equations for the low Mach number. This approximation is largely sufficient for a fire-induced flow as the air velocity is lower than 10 m/s and the ventilation velocity is less than 2 m/s (Gao et al., 2004). The model defines a subgrid turbulent viscosity using the subgrid model of Smagorinsky type in the following way (McGrattan et al., 2010a):

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

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

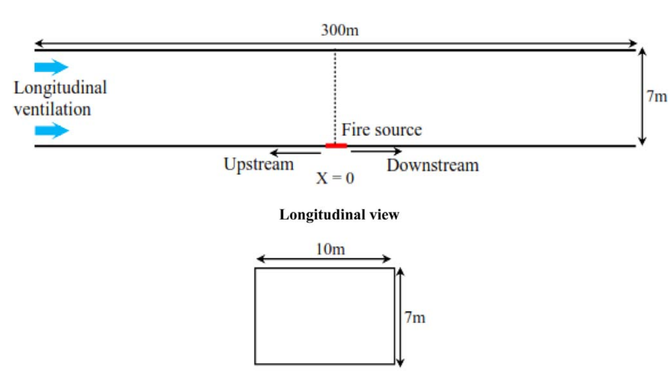

Fig. 1. Schematic view and cross section of the numerical model.