Tunnelling and Underground Space Technology 42 (2014) 52–58

Contents lists available at ScienceDirect

Tunnelling and Underground Space Technology

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

Numerical comparison of performance between traditional and alternative jet fans in tiled tunnel in emergency ventilation

Marilena Musto *, Giuseppe Rotondo

Dipartimento di Ingegneria Industriale, Università di Napoli Federico II, Italy

ARTICLE INFO

Article history:
Received 4 July 2013
Received in revised form 30 December 2013
Accepted 2 February 2014
Available online 25 February 2014

Keywords:
Tunnel fire
Critical velocity
Tiled tunnel
Alternative jet fan
Computational fluid dynamic (CFD)

ABSTRACT

In this paper a computational study was carried out to evaluate the performance of longitudinal ventilation system equipped with an alternative jet fan with respect to traditional one in case of fire in tiled tunnel. The alternative jet fan is equipped with inclined silencers (pitch angle $\alpha = 6°$) in order to reduce the Coanda effect and consequently shear stress on the tunnel ceiling. The fire was simulated setting heat flux on HGV surface. Computational fluid dynamic analysis was applied to simulate the ventilation in the unidirectional tunnel through κ–ε model. The comparison conducted in terms of total thrust required to prevent back-layering phenomena and numerical results were provided in terms of thrust of jet fan values, average velocity values and temperature profiles, for different tunnel slope values. Furthermore the authors have compared the critical velocity provided by CFD analysis with critical velocity provided in the literature.

1. Introduction

Road tunnels, usually, are equipped with ventilation systems in order both to provide a good level of safety in ordinary service and to prevent the upstream smoke flow (back-layering) and to ensure safe evacuation in case of fire. Typically, tunnel ventilation systems are of three types: longitudinal, transverse and semi-transverse. The choice of the ventilation system mainly depends on the tunnel characteristics: length, traffic directions (one-way or bi-directional), traffic volume and interactions between smoke plume and tunnel ceil. Semi-transverse systems supply fresh air only and extract exhausted air only; air is supplied or extracted continuously along the tunnel length through numerous adjustable apertures in the ventilation plenum which runs parallel to the tunnel axis (usually either above a false ceiling or below floor level). Fully transverse systems permit the independent regulation of both air supply and extraction along the tunnel length via two separate plenums. The longitudinal systems offer the advantage that the running bore acts as the ventilation duct, overcoming the need to separate the ventilation bore.

This is the main reason why longitudinal ventilation systems are adopted in one-way road tunnel with a length typically less than 3.00 km.

In the past, studies of tunnel longitudinal ventilation systems were carried out by simple (one/bi-dimensional) mathematical models and reduced or full-scale experiments scale model. These studies have been essential to understand fundamental aspects of smoke movement in case of fire. Oka and Atkinson (1995) provided a simple correlation to calculate critical velocity for both high and small fire; the authors demonstrated that formula provided by Thomas (1968), fails when the flame height is equal or greater than tunnel height. Atkinson and Wu (1997) investigated the effect of slope on critical velocity value for different burning RHR and provided a simple correlation that was subsequently confirmed by experiments carried out by Vauquelin (2005) and Vauquelin and Wu (2006). The shape influence was investigated by Wu and Bakar (2000), they essentially confirmed results obtained by previous study (Oka and Atkinson, 1995; Atkinson and Wu, 1997) and fixed new critical value of dimensionless RHR.

As mentioned above, the experimental investigation was essential to obtain the basics of ventilation systems, as they omitted the presence of jet fans and always uniform velocity was assumed at entrance of tunnel. This assumption is only true if the airflow is fully develop before it reached the fire zone; therefore it hardly can be used to study, i.e., pollutant concentration in road tunnel or to optimize complex ventilation systems.

From 1990 the CFD analysis has been widely adopted. It can easily be used to model the complexity of geometrical characteristics of boundary (spatial and time) conditions and, at the same time, provide accurate prediction of velocity and temperature

* Corresponding author. Tel.: +39 081 7682290; fax: +39 081 2390364.
E-mail address: marilena.musto@unina.it (M. Musto).

http://dx.doi.org/10.1016/j.tust.2014.02.003
0886-7798/

Nomenclature

Symbol

A	tunnel cross-section (m^2)
$C_{1\varepsilon}$, $C_{2\varepsilon}$	model constants (–)
c_p	specific heat at constant pressure ($J\ kg^{-1}\ K^{-1}$)
g	gravity acceleration ($m\ s^{-2}$)
k	turbulent kinetic energy ($m^2\ s^{-2}$)
k_g	grade correction factor (–)
G_b	generation of turbulent kinetic energy due to the buoyancy ($kg\ m^{-1}\ s^{-3}$)
G_k	generation of turbulent kinetic energy due to the mean velocity gradient ($kg\ m^{-1}\ s^{-3}$)
Q	heat release rate (W)
T	temperature (K)
u, v, w	velocity components ($m\ s^{-1}$)
u_{av}	tunnel average volume velocity ($m\ s^{-1}$)
u_{ca}	analytical critical velocity ($m\ s^{-1}$)
u_{cn}	numerical critical velocity ($m\ s^{-1}$)
x,y,z	cartesian coordinates (m)

Greek letters

α	inclination angle (°)
ε	dissipation rate of turbulent kinetic energy ($m^2\ s^{-3}$)
μ	dynamic viscosity (Pa s)
μ_t	dynamic turbulent viscosity (Pa s)
ρ	air density ($kg\ m^{-3}$)
σ	turbulent Prandtl Number (–)
t	time (s)

Subscripts

α	tiled angle value (°)
i, j, l	indexes
k	turbulent kinetic energy
hg	hot gases
ε	dissipation rate of turbulent kinetic energy

fields and concentrations of smoke formed in the air in case of fire. The main advantage of the CFD approach has made it an almost ideal method for designing and optimizing ventilation and smoke extraction systems. Nevertheless, the velocity and temperature fields predicted by mathematical model using CFD, are affected by uncertainty in absence of validation by means of experimental full-scale data (Vauquelin, 2008).

Recent research has focused on the design and optimization of ventilation systems i.e. the pitch of jet fans, the swirl motion of jet fan and the jet fan position. To reduce the Coanda effect and improving the efficiency of the installation, an experimental system characterized by jet fans with inlet/outlet sections inclined slightly toward the center of the tunnel has been proposed by Martegani et al. (1997), Witt and Schtüze (2006) and Armstrong et al. (1994).

In order to increase the energy efficiency and improve the performance of a longitudinal ventilation system, Betta et al. (2010, 2009) have compared numerically both traditional and alternative jet fans for a road tunnel with and without traffic jam and they proposed to adopt a pitch angle equal to 2° and 6° respectively. Wang et al. (2012) were investigated the effects of deflected angles of jet fans on ventilation systems in a curved tunnel, obtaining an optimal angle value as a result of a maximum pressure rise. Camby (2012) provided a theoretical correlation between the distance of active jet fan group from the fire source and upstream velocity, for tiled tunnel.

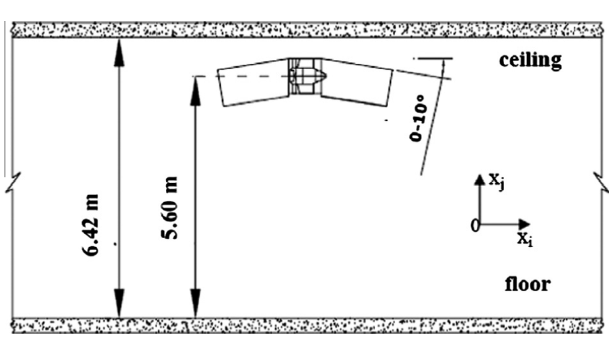

Fig. 1. Longitudinal tunnel sketch.

In this paper the authors compare numerically the performance of longitudinal ventilation systems for tiled road tunnel equipped with alternative jet fans (LAVS, Longitudinal Alternative Ventilation System) with respect to traditional one (LTVS, Longitudinal Traditional Ventilation System), in emergency. The performance was evaluated in terms of total thrust required to prevent the back-layering phenomena and numerical results in terms of axial average velocity and temperature profiles, for different slope values.

Furthermore, the authors have compared the critical velocity computed from CFD results with those provided in literature (Danziger and Kennedy, 1982; Lee et al., 1979; Kunsch, 2002).

2. Computational tunnel domain and CFD details

2.1. Computational tunnel domain

The investigated physical domain consists in an 800 m length one directional road tunnel equipped with four jet fans arranged in longitudinal ventilation system. The jet fans were all positioned at 5.60 m above tunnel floor (Fig. 1); the distance between the two successive fans was fixed in 200 m while the jet fans J_{f1} and J_{f4} (Fig. 2) were installed at a distance of 100 m from inlet and outlet sections of the tunnel, respectively. The thrust of jet fans and their spacing were chosen to guarantee that the local velocity value, in optimized system ventilation without fire, tends to the designed velocity value in the proximity of the subsequent inlet jet fan section (Betta et al., 2010).

In order to reduce the computational time, the physical domain was divided into three different zones (A, B and C) with different grid sizes as shown in Fig. 2. The need to discretize the domain in different areas depends on the high gradients of the air velocity and temperature that along the tunnel occur.

The B type zone was meshed by tetrahedral grid with step equal to 0.1 m. The A and C type zones were meshed with hexahedral grid with step equal 0.4 m obtained by previous mesh analysis (Betta et al., 2010). The tetrahedral grid and the very fine grid step choices are necessary, respectively, to adapt the different geometrical blocks shape, to simulate very high velocity gradients. The jet fan was simulated as momentum source divided into tetrahedral grids with step equal to 0.1 m (Betta et al., 2010), as shown in Fig. 2.