Wind and Structures, *Vol. 19, No. 2 (2014) 185-197*
DOI: http://dx.doi.org/10.12989/was.2014.19.2.185

Computational study of road tunnel exposure to severe wind conditions

Simon Muhič[*1,2] and Mitja Mazej[1]

[1]School of Technologies and Systems, Na Loko2, SI-8000 Novo mesto, Slovenia
[2]SimTec, Dr. Simon Muhič s. p., Stična 113, SI-1295 Ivančna Gorica, Slovenia

(Received December 26, 2013, Revised June 16, 2014, Accepted June 28, 2014)

Abstract. Ventilation and fire safety design in road tunnels are one of the most complex issues that need to be carefully considered and analysed in the designing stage of any tunnel project, as well as in case of any potential upgrade of ventilation and other fire safety systems in tunnels. Placement of road tunnels in space has an important influence on fire safety, especially when considering the effect of adverse wind conditions that significantly influence ventilation characteristics. The appropriate analysis of fire and smoke control is almost impossible without the use of modern simulation tools (e.g., CFD) due to a large number of influential parameters and consequently extensive data. The impact of the strong wind is briefly presented in this paper in the case of a longitudinally ventilated road tunnel Kastelec, which is exposed to various severe wind conditions that significantly influence its fire safety. The possibility of using CFD simulations in the analysis of the tunnel placement in space in terms of negative effect of wind influence on the tunnel ventilation is clearly indicated.

Keywords: CFD (computational fluid dynamics); wind loads; simulations; tunnel safety; longitudinal ventilation

1. Introduction

Ventilation plays a fundamental and extremely important role in road tunnel safety - both in case of normal operating conditions and in emergency conditions in case of fire. Several disasters in road tunnels occurred due to fire, like the crashes in Mont Blanc Tunnel and Tauern Tunnel in 1999 or Saint Gotard Tunnel in 2001. These accidents have confirmed that ventilation design is one of the main issues for the fire safety (Leitner 2001, Vuilleumier *et al*. 2002). While under normal operation ventilation has to keep the concentration of generated pollutants within safety levels and maintain good visibility, its role under emergency conditions - to extract smoke to enable fast and safe evacuation of passengers and safe firefighting activities - is even more significant. In some cases ventilation may be already provided by natural means or by the traffic-induced piston flow alone, however, mechanical ventilation systems are necessary in most cases and different types of them are commonly used in tunnels, depending on the tunnel

∗Corresponding author, Professor, E-mail: simon.muhic@guest.arnes.si
[a] Ph.D., E-mail: mitja.mazej@netsi.net

http://www.techno-press.org/?journal=was&subpage=8 ISSN: 1226-6116 (Print), 1598-6225 (Online)

characteristics, layouts and traffic directions (PIARC: Fire and Smoke Control in Road Tunnels, 1999). In modern road tunnels, transverse and longitudinal ventilation systems are two most commonly adopted designs to provide smoke control. Transverse ventilation systems are based on ducts for fresh air supply and extraction of waste air, located in the upper part of the tunnel cross-section. These systems are designed to assure smoke confinement only in a small section of the tunnel, keeping the air velocity low to enhance stratification and provide effective extraction of smoke. This is the reason why they are especially well suited for use in longer tunnels with bi-directional traffic. Longitudinal ventilation, based on the use of jet fans installed within the tunnel bore, is highly effective when used in unidirectional traffic tunnels by providing flow in the direction of traffic (Wu and Baker 2000). These ventilation systems are designed to assure longitudinal air velocity that is high enough to prevent spreading the smoke against the prevailing flow, commonly known as back-layering (Oka and Atkinson 1995, Kunsch 2002, Vaquelin and Wu 2006). Prevention of this effect can be achieved when longitudinal air velocity generated by the ventilation system is higher than the critical ventilation velocity (Hwang and Edwards 2005, Roh *et al*. 2007, Hu *et al*. 2008, Modic 2003). However, severity of tunnel fire is more critical in case of longitudinal ventilation systems due to the absence of the extraction system when its efficiency in smoke management depends only on airflow velocity provided by jet fans. Therefore these ventilation systems have to incorporate the operating manual with the airflow conditions necessary to control the smoke and the direction of flow clearly specified. Due to the inherent complexity of the problem, considering non-linear convection of combustion gases in the particular tunnel geometry, fire characteristics and variable weather boundary conditions at the tunnel portals (wind pressure), operating conditions are not trivial as they cannot be easily estimated. Moreover, considering the effect of adverse wind conditions on exposed tunnel portals that significantly influence ventilation characteristics mean that the placement of road tunnels in space has an important influence on the overall fire safety. Appropriate analysis of fire and smoke propagation in such cases is almost impossible without the use of numerical modelling, based on modern CFD simulation tools, which have been widely used to solve airflow both under the normal traffic conditions as well as in case of emergency conditions, providing detailed supplementary information to field measurements or experimental studies. While full scale tests are not possible in the designing stage of the tunnel, the only remaining experimental analysis may be performed on scaled models, based on dimensional analysis and similarity law. However, in comparison to this approach, numerical modelling has some important advantages, such as the ability to provide the performance of ventilation and to optimize efficiency of the method prior to the validation experiment.

Although many different numerical studies on ventilation characteristics in road tunnels have been performed (Van Maele and Merci 2008, Vega *et al*. 2008, Wang *et al*. 2010, Se *et al*. 2012, Tang *et al*. 2013), there are no known studies dealing with the aerodynamic influence of wind on tunnel portals. The main objective of the present pilot study was therefore to numerically analyse the influence of adverse wind conditions on ventilation and fire safety characteristics in the case of a longitudinally ventilated unidirectional road tunnel, using a commercial CFD package ANSYS Fluent. A detailed three-dimensional geometry of the terrain with two tunnel pipes was developed for the purpose of this study and then applied to simulate flow field characteristics in the tunnel during normal operation caused by different wind conditions, considering wind direction and velocity. Another objective of this study was to indicate the necessity of the tunnel placement analysis as a fundamental part of the design, especially in case of tunnel portals being exposed to wind due to specifics of surrounding terrain geometry, and the potential of using CFD tools for this purpose.

The present study was motivated by the need for the understanding of the aerodynamic behavior in Kastelec tunnel, which is one of the two highway tunnels on the A1 motorway section between Ljubljana and Koper in Slovenia. The tunnel consists of two separated one-way, two-lane traffic pipes, the left one 2278 m and the right one 2237 m long, with a minimum horizontal curve radius of approximately 1800 m for both pipes and cross-section area 54,5 m^2. Both portals of the tunnel on its northern side are directed towards east and therefore directly exposed to the strongest wind in the region, known as bora, which has distinctive north-east directions. Situations in which the bora wind is strong enough to establish flow direction in the left tunnel pipe, which is opposite to the traffic direction (negative velocity), are quite common and as a consequence the tunnel needs to be partially closed in direction towards Ljubljana in order to assure appropriate fire safety in the case of emergency.

2. Methods

2.1 About computational fluid dynamics

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations: Conservation of mass, Conservation of momentum, Conservation of energy, Conservation of species and Effects of body forces.

CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition. ANSYS Fluent solver is based on the finite volume method. Using this method domain is discretised into a finite set of control volumes. General conservation equations for mass, momentum, energy, species, etc. are solved on this set of control volumes. General conservation equation in integral form is defined by:

$$\frac{\partial}{\partial t}\left(\int_{CV}\rho\phi dV\right)+\int_{A}\mathbf{n}(\rho\phi\mathbf{u})dA=\int_{A}\mathbf{n}(\Gamma\,\mathrm{grad}\,\phi)dA+\int_{CV}S_{\phi}dV \tag{1}$$

Replacing the variable ϕ with 1, gives continuity equation, with *u*, *v* and *w*, X, Y and Z momentum equations, and with *h*, energy equation. Furthermore, turbulence kinetic energy *k* and its dissipation rate *ε* are in the case of RNG k-ε model solved from the following transport equations, respectively

$$\frac{D}{Dt}(\rho k)=\frac{\partial}{\partial x_j}\left(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon \tag{2}$$

$$\frac{D}{Dt}(\rho\varepsilon)=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon\mu_{eff}\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{3}$$

In Eqs. (2) and (3) α_k and α_ε are the inverse effective Prandtl numbers for *k* and *ε*, respectively, μ_{eff} the effective viscosity, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients and G_b the generation of turbulence kinetic energy due to buoyancy. At high Reynolds numbers effective viscosity transforms into the turbulent viscosity term

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

Partial differential equations are discretised into a system of algebraic equations and all algebraic equations are then solved numerically to render the solution field. Additionally, in numerical solution first-order upwind discretization scheme was used both for convective and diffusion terms, whereas the SIMPLE algorithm was used as a pressure-velocity coupling method. As a measure for reaching convergence criteria the sum of normalized absolute residuals in every control volume for all variables should drop below 10^{-3}, only for energy this was set to 10^{-6}.

2.2 About tunnel Kastelec

Considering the need for understanding the aerodynamic influence of adverse wind conditions in case of Kastelec tunnel it was necessary to develop detailed geometry of surrounding terrain, both tunnel portals and the tunnel pipe in order to appropriately place the tunnel in space. A Digital Elevation Model (DEM 12.5), obtained from the Slovenian surveying and mapping authority, was used in the numerical study to generate all important terrain characteristics for this purpose. This model is a part of the Digital Relief Model (DRM) of Slovenia and its geographical surroundings, which includes more than 25 types of elevation data from 1947 onwards, such as digital terrain models with a resolution from 10 to 600 m, digitized contour lines, roads and railway layers of different scales, surveying points, building cadastre, etc. DRM characteristics are appropriate in terms of the elevation model, which is homogenous, contains no rough errors and covers a wider area around Slovenia. This model is comprised of altogether more than 353 million points at the resolution of 12.5 m, resulting in the estimated terrain accuracies of 1.1 m for planes, 2.3 m for knolls, 3.8 m for hills and 7.0 m for mountains, with the overall model accuracy of 3.2 m. The DEM data can be issued in Arc/Info GRID format and the format yxz with the basic unit of the issuance given as the basic topographic map sheet at the scale of 1:5000. Basic topographic maps are made for the entire territory of Slovenia with the sheet size of 2250 × 3000 m. Altogether 3258 sheets cover the whole area. For the present tunnel placement analysis these data was used between coordinates (410000, 46000) and (416750, 52000), corresponding to the area of 6750 × 6000 m as given in Fig. 1.

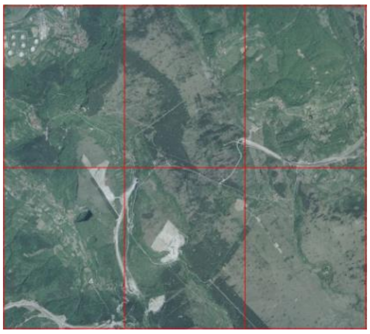

Fig. 1 Orto photo of the terrain analysed in the CFD model from public viewer of Ministry of Agriculture, Forestry and Food of the Republic of Slovenia

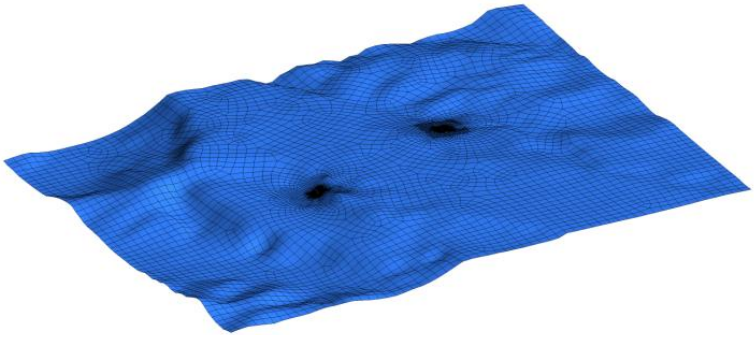

Fig. 2 3D numerical model of the terrain

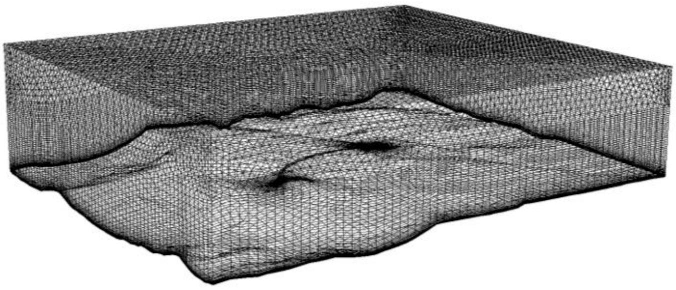

Fig. 3 3D view of the numerical domain with generated numerical mesh

In the pre-processing stage a preprocessor of the software package was used to generate the detailed terrain geometry. Appropriately treated spatial data of the DEM was imported into the preprocessor, where a high-resolution contour surface of the terrain was generated over the imported spatial points (Fig. 2). Based on the ortho-photo imagery of the terrain, locations of both left tunnel pipe portals were estimated and included into the model. Due to the terrain height differences the generated computational domain volume over the terrain surface had to be sufficiently high to account for boundary layer effects of the vertical wind velocity profile. Therefore the top of the domain was on height of 2000 m, which was at least by 1167 m above the terrain, and the flow at the upper part of the domain was under no influence.

2.3 Computational grid for CFD

Grid generation in the computational domain required high grid resolution in the region of complex flow interactions around the tunnel portals and in the tunnel itself, which is fundamental for the numerical stability and precision of the results. Therefore the whole domain was divided into four main sections, namely two rectangular volumes around the portals, tunnel volume and an outer region, covering the rest of the domain volume (Fig. 3).

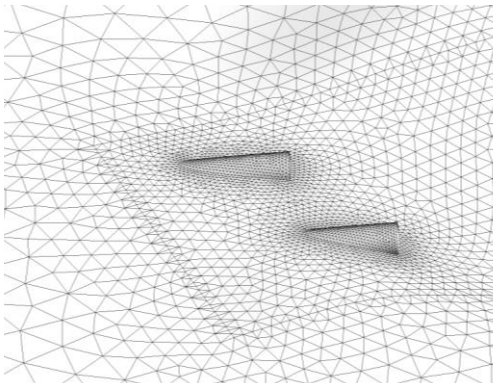

Fig. 4 Surface grid transition detail around the northern tunnel portal

With such division of the domain unstructured grid with a tetrahedral cell topology was used around the complex geometry of the portals, whereas a structured grid with hexahedral cell topology was used in the rest of the domain. This approach allowed optimized grid stretching from the portals towards the side of the domain and ensured higher accuracy for a given number of grid cells (Fig. 4).

Computational grid requirements have been assessed on different mesh sizes, including the chosen grid, comprised of 1338109 cells. Grid for both tunnel pipes was modelled with structured mesh with 101624 cells. Performed grid sensitivity and solution dependency test confirmed the chosen grid to be solution independent and the grid was therefore used for the simulations as it offered an acceptable compromise between the accuracy of simulation results and computational cost for pilot study.

2.4 Boundary conditions for CFD

For accurate simulation of the adverse wind conditions and its effect on the aerodynamic characteristics in the tunnel, the computational domain side boundary surfaces were treated as inlet, outlet and symmetry surfaces, depending on the simulated wind direction. The top surface of the domain was defined by the symmetry boundary condition, while the terrain and the tunnel surfaces were defined as the adiabatic walls, imposing no-slip condition. In homogeneity of the terrain characteristics across the simulated area was not taken into account due to effects of vegetation and was assumed to be smooth. Empty tunnel without any traffic or fire inside the tunnel and with inactive longitudinal ventilation was simulated in order to directly assess the influence of the wind under steady-state conditions. In the case of the fire in tunnel, air flow inside the tunnel and at portals is strongly influenced by the fire. Ambient temperature of -6.5°C at the height of 0 m was assumed in all simulated cases and used when comparing simulation results with measurements obtained on a winter day when the tunnel was closed due to the strong wind. On the basis of the available meteorological data on the strongest wind in the region, known as ‘bora’, this wind has distinctive north-east directions. Therefore in the present analysis the wind was described by the three main parameters, namely velocity, direction and temperature. In this way the wind velocity on the inlet boundary of the domain was defined by the velocity profile, given

$$\frac{u(z)}{u_r} = \left(\frac{z}{z_r}\right)^{\alpha} \tag{5}$$

where u_r is the velocity at the reference height z_r (m), which is usually 10 m above the ground, z (m) the height above the ground and α the wind profile parameter equal to 1/7 (Elliott 1979). The present analysis was performed for the wind reference velocities of 80, 100, 120 and 140 km/h. Furthermore, the wind temperature profile which depends on the altitude was considered and defined by:

$$T(z) = T_0 - \frac{6.5 \cdot z}{1000} \tag{6}$$

whereT_0 is the air temperature at the ground level and was in our case set to 0 °C and z (m) is the height above the ground. With this equation we take into account linear drop of air temperature of 6.5 K for every 1000 m of height. Altogether 7 different wind directions were simulated and analysed, covering the 90° angle between the north wind (0°) and the east wind (90°) by every 15°.

The flow in the computational domain under the given boundary conditions was assumed to be steady and incompressible, as the main objective of the present study was to analyse induced flow in the left tunnel pipe due to specific aerodynamic characteristics in case of adverse wind conditions without the occurrence of fire. The commercial CFD code ANSYS Fluent v14.0 was employed to solve the continuity, momentum and energy equations with the Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling approach. Although Large Eddy Simulation (LES) turbulence modelling has been recently applied in some tunnel fire safety and ventilation studies (Van Maele and Merci 2008, Hu *et al*. 2008), the huge computational costs prevent this approach from being used for large scale tunnel simulations such as in the present case. Therefore selection of RANS approach was a logical compromise regarding accuracy, computational stability and CPU time. From the range of RANS models the standard k-ε turbulence model has been mostly used in tunnel ventilation cases and has also been validated against the experimental data (Vega *et al*. 2008, Wang *et al*. 2010). However, the present study importantly differed from these cases as its main concern lies in the modelling of the wind flow over the terrain with all its specific features, such as separation and recirculation. Therefore in this particular study, the RNG k-ε turbulence model combined with the wall functions and the standard set of constants for the k-ε equations was adopted as it represents a better compromise in terms of accuracy and numerical stability in comparison to the standard k-ε model (Kim and Patel 2000, Stangroom 2004). In numerical solution first-order upwind discretization scheme was used for all parameters and SIMPLE algorithm was used as a pressure-velocity coupling method. As a measure for reaching convergence criteria the sum of normalized absolute residuals in every control volume for all variables should drop below 10^{-3}, only for energy this was set to 10^{-6}.

3. Results and discussion

With CFD simulation variables of the air flow through the numerical domain were calculated. Figs. 5 and 6 show for example air velocity at height 2 and 10 m above the terrain.

Results of the numerical analysis of the wind direction and velocity influence on the flow field characteristics in the left pipe of the Kastelec tunnel are given in Table 1. This pipe is problematic due to flow direction, which is opposite to the traffic direction (negative velocity). Flow characteristics are given by the volume integral of the velocity in the tunnel, representing the average speed of air flow that occurs in the whole tunnel. Positive velocity values represent the flow direction in the direction of the traffic, while negative values represent the flow direction which is opposite to the traffic direction and represents the situation which should be prevented by the longitudinal ventilation system in order to assure appropriate fire safety in case of emergency.

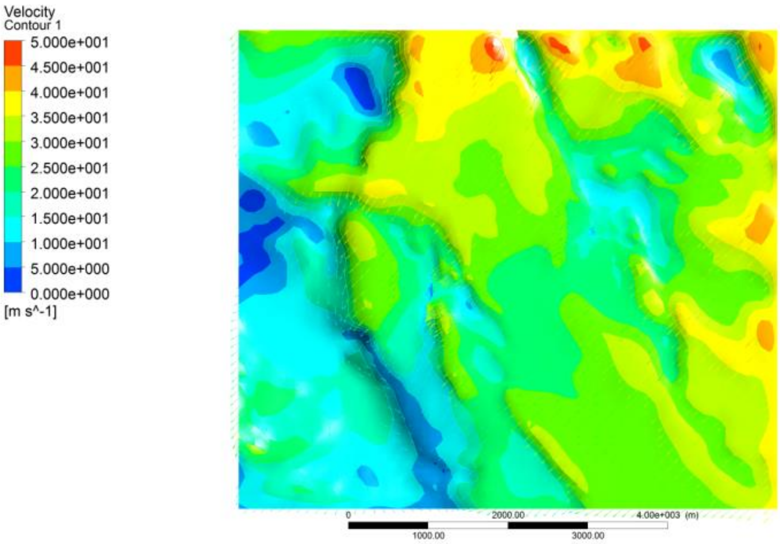

Fig. 5 Air velocity at height 2 m above the terrain

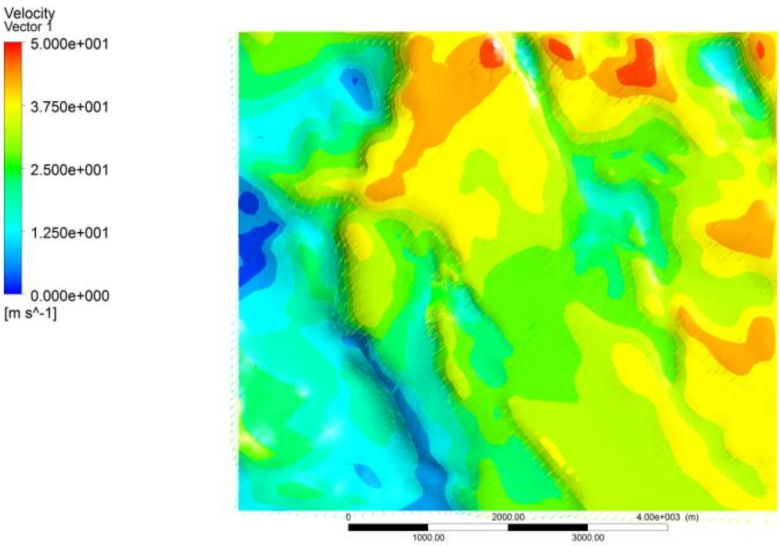

Fig. 6 Air velocity at height 10 m above the terrain

Table 1 Simulation results of the average air velocity in the left tunnel pipe with respect to the external wind velocity at height 10 m (negative velocity: flow opposite to the traffic direction)

	80 km/h	100 km/h	120 km/h	140 km/h
Winddirection from N (Deg)	Airvelocity in pipe (m/s)	Airvelocity in pipe (m/s)	Airvelocity in pipe (m/s)	Airvelocity in pipe (m/s)
0	2.92	3.86	4.65	5.32
15	2.94	3.51	4.13	4.71
30	0.72	0.62	-0.60	-0.65
45	-5.31	-6.82	-8.24	-9.96
60	-8.03	-10.33	-12.49	-15.01
75	-11.37	-14.41	-16.94	-20.86
90	-11.62	-14.40	-17.74	-20.81

Graphical representation of these results is given in Fig. 7. Obviously the flow field in the tunnel is strongly influenced by the east wind direction, with the highest negative velocities obtained at the directions of 75° and 90° from the north. Moreover, there is a significant influence of the wind velocity on the negative flow in the tunnel both in terms of its absolute values as well as in terms of its behaviour. There is a clear tendency of higher wind velocities having stronger influence already at the wind directions from the north (30°), while at both lower wind velocities negative flow occurs at the angle of 45° from the north.

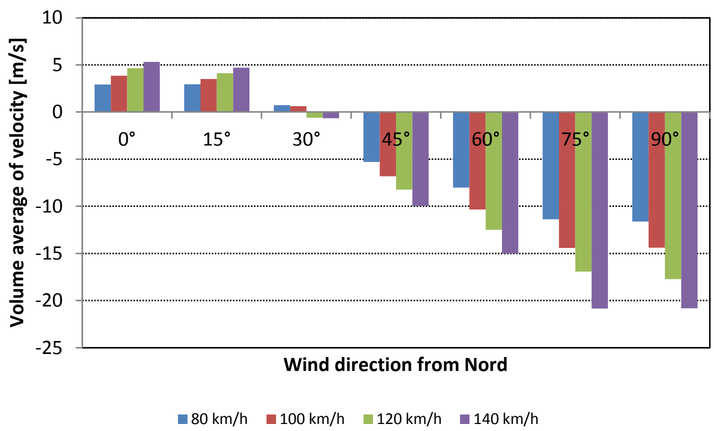

Fig. 7 Volume average of air velocity in the left tube of the tunnel for different reference air velocities at the reference height 10 m

For the purpose of verifying of the numerical model, the numerical simulation results were compared with the left tunnel pipe velocity data, measured on 23rd and 24th Jan 2006 when the tunnel was closed due to excessive wind velocity. Such simulation is hard to verify because the steady state problem was simulated in environment with stochastic wind and without any measurement device in numerical domain. But in order to perform at least partial verification of the numerical model the meteorological data from the closest two measuring meteorological stations of the Slovenian Environmental Agency (ARSO) were obtained for the same period of time. The available data from the two meteorological stations, one located in the Port of Koper and the other one in the Park of Škocjan caves, provided information on the scalar of an average velocity, an average wind direction, the maximum gust of wind in half an hour interval, the standard deviation of half an hour wind speed and the time interval wind velocity and direction measurements. Those two stations are located several kilometres from analysed area as shown in Fig. 8.

The left tunnel pipe air velocity data between the 23rd and 24th Jan 2006 in comparison to the average wind direction and average velocity data in 30 minutes interval from the two stations are presented in Fig. 9. From this data it can be seen that both wind velocities and the tunnel negative velocity are very well correlated, considering the time lag due to the distance between the three locations. It is evident that the highest negative velocity in the tunnel occurred in the period with the east and north-east wind direction, which is very much in agreement with the numerical results. Moreover, the highest negative velocities in the tunnel correspond to the simulated effect of the east wind at 80 km/h, which is in agreement with the maximum wind gust velocities measured during this period at the meteorological station Škocjan. In the meteorological data obtained from meteorological station Škocjan we could find data for 24th Jan at 2 AM where average air velocity in last 30 minutes was 10.7 m/s and the peak velocity was 20.5 m/s with average wind direction 91 degrees from Nord. A bit later, at 2:15 AM, maximum velocity 11.45 m/s occur in the tunnel.

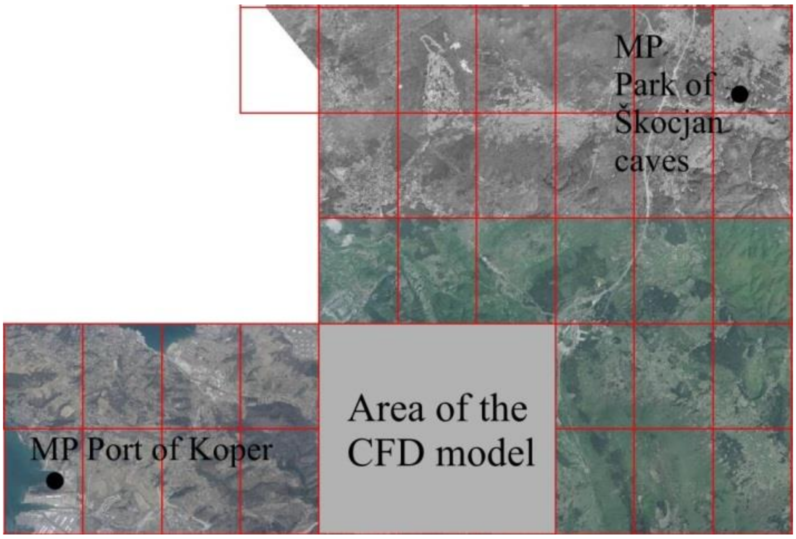

Fig. 8 Orto photo of the terrain, analysed in the CFD model with location of the closest two measuring meteorological stations of the Slovenian Environmental Agency from public viewer of Ministry of Agriculture, Forestry and Food of the Republic of Slovenia

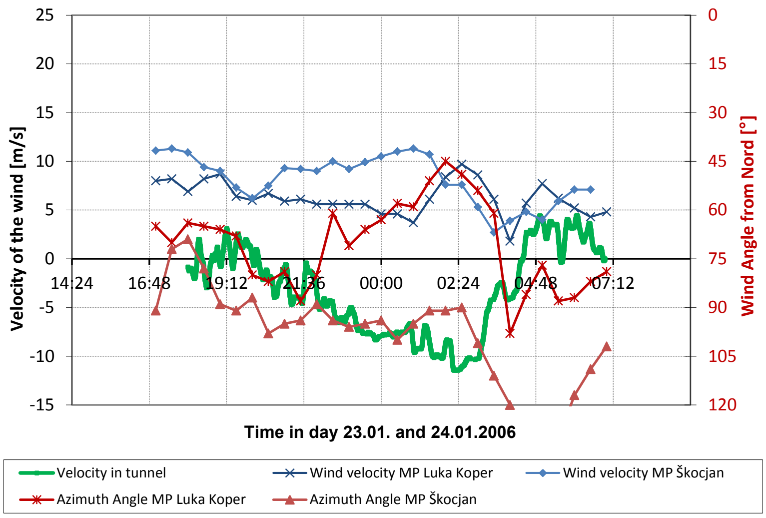

Fig. 9 Velocity measurements in the left tunnel pipe in comparison to the wind direction and velocity data (hour average velocity) from the two meteorological stations in the observed time period

4. Conclusions

The present study is a proof that meteorological parameters and appreciable wind influence are important issues that need to be carefully considered in the procedure of designing a road tunnel and its ventilation system. Performed numerical analysis of the adverse wind conditions shows important influence of typical weather conditions on operation of the longitudinally ventilated unidirectional road tunnel Kastelec. These conditions have important impact to ventilation and fire safety characteristics of the tunnel. This study clearly indicated the weaknesses of the existing road tunnel Kastelec in terms of its exposure to stronger winds of east to north-east directions that result in negative velocity in the left pipe and consequently reduced fire safety in the tunnel. Wind influence is obviously most pronounced at the northern portal of the tunnel Kastelec, where the static pressure is significantly higher than on the southern portal, where the static pressure is neutral or slightly negative. Furthermore, from the range of simulated wind directions, the highest exposure with respect to the total pressure difference between the pipe portals has been obtained for the north-east wind directions of 45° and 60° from north, which correspond to the strongest bora wind in the region. On the other hand the lowest exposure influence of the tunnel was observed in case of the north wind

Another important observation of this study has also been finding that the design parameter *k* may underestimate the wind influence if derived simply as a function of the angle between the direction of the wind and the direction of the air flow entering/exiting the tunnel, as proposed by PIARC (*PIARC, Fire and Smoke Control in Road Tunnels, 1999*). Within the recommendations of fire and smoke control in road tunnels given by PIARC, the effective wind resistance or thrust is defined simply as a function of the angle between the direction of the wind and the direction of the air flow entering/exiting the tunnel. However, this may be true for more common situations where micro location characteristics of the terrain do not have an influence on the flow direction, but the obtained results of the present study have shown that such an approach is not entirely correct, as it may underestimate the wind influence. Therefore, when evaluating a stated wind pressure, the wind reduction factor *k*, which depends on the configuration of the portals, should be considered very carefully as it is obviously the most influential parameter in the design process of the ventilation system performance. Therefore it is necessary that the wind reduction factor is defined more carefully and accurately in order to properly design tunnel and its ventilation systems for fire safety and to avoid exploitation problems of such important infrastructure objects due the impact of the “normal” weather conditions in the area where the object is located. This being a demanding issue, the situation may be improved by seeking the aerodynamic solutions in reducing the impact of the east wind on the exposed portals of the tunnel.

Last but not least, this study has clearly shown the importance of detailed wind analysis performed in an early stage of choosing the best geographical location for the tunnel and portholes in space. Among other important factors that influence construction characteristics of the tunnel, wind effects analysis on ventilation and fire safety is fully recommended for different tunnel variants being considered in this process in order to avoid potential inconveniences in operation due to strong winds.

References

Elliott, D.L. (1979), *Adjustment and analysis of data for regional wind energy assessments*, Workshop on Wind Climate.

Hu, C.H., Ohba, M. and Yoshie, R. (2008), “CFD modeling of unsteady cross ventilation flows using LES”. *J. Wind Eng. Ind. Aerod.*, **96**(10-11), 1692-1706.

Hu, L.H., Penga, W. and Huoa, R. (2008), “Critical wind velocity for arresting upwind gas and smoke dispersion induced by near-wall fire in a road tunnel”, *J. Hazard. Mater.*, **150**(1), 68-75.

Hwang, C.C. and Edwards, J.C. (2005), “The critical ventilation velocity in tunnel fires – a computer simulation”, *Fire Safety J.*, **40**(3), 213-244.

Kim, H.G. and Patel, V.C. (2000), “Test of turbulence models for wind flow over terrain with separation and recirculation”, *Earth Environ. Sci.*, **94**(1), 5-21.

Kunsch, J.P. (2002), “Simple model for control of fire gases in a ventilated tunnel”, *Fire Safety J.*, **37**(1), 67-81.

Leitner, A. (2001), “The fire catastrophe in the Tauern tunnel: experience and conclusions for the Austrian guidelines”, *Tunn. Undergr. Sp. Tech.*, **16**(3), 217-223.

Modic, J. (2003), “Fire simulation in road tunnels”, *Tunn. Undergr. Sp. Tech.*, **18**(5), 525-530.

Oka, Y. and Atkinson, G.T. (1995), “Control of smoke flow in tunnel fires”, *Fire Safety J.*, **25**(4), 305-322.

Roh, J.S., Ryou, H.S., Kim, D.H., Jung, W.S. and Jang, Y.J. (2007), “Critical velocity and burning rate in pool fire during longitudinal ventilation”, *Tunn. Undergr. Sp. Tech.*, **22**(3), 262-271.

Se, C.M.K., Lee, E.W.M. and Lai, A.C.K. (2012), “Impact of location of jet fan on airflow structure in tunnel fire”, *Tunn. Undergr. Sp. Tech.*, **27**(1), 30-40.

Stangroom, P. (2004), *CFD modeling of wind flow over terrain*, Ph.D. Thesis, Notingham, University of Notingham.

Tang, W., Hu, L.H. and Chen, L.F. (2013), “Effect of blockage-fire distance on buoyancy driven back-layering length and critical velocity in a tunnel: an experimental investigation and global correlations”, *Appl. Therm. Eng.*, **60**(1-2), 7-14.

Van maele, K. and Merci, B. (2008), “Application of RANS and LES field simulations to predict the critical ventilation velocity in longitudinally ventilated horizontal tunnels", *Fire Safety J.*, **43**(8), 598-609.

Vaquelin, O. and Wu, Y. (2006), "Influence of tunnel width on longitudinal smoke control", *Fire Safety J.*, **41**(6), 420-426.

Vega, M.G., Argüelles Díaz, K.M., Fernández Oro, J.M., Tajadura, R.B. and Morros, C.S. (2008), "Numerical 3D simulation of a longitudinal ventilation system: memorial tunnel case", *Tunn. Undergr. Sp. Tech.*, **23**(5), 539-551.

Vuilleumier, F., Weatherill, A. and Crausaz, B. (2002), "Safety aspects of railway and road tunnel: example of the Lötschberg railway tunnel and Mont-Blanc road tunnel", *Tunn. Undergr. Sp. Tech.*, **17**(2), 153-158.

Wang, F., Wnag, M., He, S., Zhang, J. and Deng, Y. (2010), "Computational study of effects of jet fans on the ventilation of a highway curved tunnel", *Tunn. Undergr. Sp. Tech.*, **25**(4), 382-390.

Wu, Y. and Baker, M.Z.A. (2000), "Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity", *Fire Safety J.*, **35**(4), 369-390.

CC