Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 45 (2014) 1146 – 1154

Energy Procedia

68th Conference of the Italian Thermal Machines Engineering Association, ATI2013

Numerical analysis for reduced-scale road tunnel model equipped with axial jet fan ventilation system

Antonio Costantino, Marilena Musto*, Giuseppe Rotondo, Alessandro Zullo

Dipartimento di Ingegneria Industriale - Università degli Studi di Napoli Federico II - P.leTecchio, 80 – 80125 – Napoli - Italy

Abstract

In this paper a preliminary tridimensional CFD analysis is carried out for a future realization of a reduced-scale road tunnel experimental apparatus of 1/50 equipped with different impulsive fans, traditional and alternative. The alternative jet fan is provided of inlet/outlet sections inclined at a fixed pitch angle (α=6°) toward the tunnel floor. Typically, in the experimental scale tunnel model the air flow induced by ventilation system is provided by an external fan and fully developed flow field is considered. In this paper, the authors have simulated a realistic full and reduced-scale tunnel in order to evaluate the influence of ceiling and floor roughness height on the velocity field to identify an appropriate material for a future experimental apparatus.
The jet fans are simulated as a simple momentum source. The fan is considered to be infinitely thin and the discontinuous pressure rise (pressure drop) across it is defined as a function of the air velocity through the fan. In order to create a reduced-scale model from a full scale, Froude method is applied to preserve geometrical, kinematical and dynamical similitude. The results, provided in terms of axial velocity profiles in different tunnel sections, show the overlapping between velocity profiles of full scale numerical model with those of the reduced-scale model, for the both ventilation systems.

Keywords: Longitudinal ventilation; Scaled tunnel; CFD.

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 and to ensure safe evacuation in case of fire. There are three kinds of ventilation systems whose choice is made considering the tunnel length, traffic directions (one-way or bi-directional), traffic volume, interaction between smoke plume and tunnel ceil: longitudinal, transverse and semi-transverse. With the Longitudinal Ventilation Systems (LVS) exhausted air is pushed away from the tunnel entrance

doi:10.1016/j.egypro.2014.01.120

and fresh air is pushed into the tunnel through the other tunnel side. This is possible by means impulsive fans (jet fan) installed to the tunnel's ceiling. These longitudinal ventilation systems are usually adopted in one-way road tunnel with a length typically less than 3.00 km equipped with traditional fans, with inlet and outlet sections parallel toward the tunnel floor. In the latest years, a new kind of jet-fan (alternative jet fan), equipped with inlet/outlet sections inclined at a fixed pitch angle (α) toward the tunnel's floor, has been introduced to obtain economic advantages and energy savings.

In order to study the alternative jet fan [1] and to compare its performance with that of the traditional fan, realistic ventilation systems have been simulated through the CFD analysis [2]. Then, to study the ventilation system, scaled model tunnels have been realized in order to reduce costs of the experiments. In literature there are a lot of example of scaled tunnel but in every scaled model there is the lack of a ventilation system made up of fans connected to the tunnel ceiling. Ingason et al. [3] effected 12 tests on a model 1/23, 10 m long and 0.4 m width; in the model, which had a variable height, the flow was produced by an electric fan placed in correspondence of the inlet section of the model and contiguous to a plywood box useful to realize a uniform flow at the entrance. Other models have been realized using external fan [4; 5], compressed air [6] or wind tunnel [7].

In this paper a preliminary tridimensional CFD analysis was carried out for the realization of an experimental set up of 1/50 reduced-scale road tunnel model equipped with different impulsive fans, traditional and alternative. The alternative jet fan was equipped with inlet/outlet sections inclined at a fixed pitch angle (α = 6°) toward the tunnel floor. Froude method was applied to preserve geometrical, kinematical and dynamical similitude. Due to the presence of impulsive fans in the tunnel bore there were simultaneously areas with high air velocity (at the output sections of jet fan) and areas with low air velocity, therefore, it was not possible assure both completely dynamical and kinematical similitude. In order to preserve the kinematic similitude, slightly different pressure drop was imposed in the reduced-scale model from those calculated by Froude method. Preserving the average Froude number in the volume (Fr_{av}) during the scaling process, the dynamic and kinematic similitude was been quite ensured. The simulations were carried out by means a CFD commercial software FLUENT.

The authors have investigated the effect of tunnel ceiling and floor roughness height on the velocity field to identify an appropriate material to make the experimental apparatus of the tunnel equipped with ventilation systems in a reduced-scale model.

Nomenclature

c_p	specific heat, kJ kg^{-1} K^{-1}
D_H	hydraulic diameter, m
Δp	fan pressure drop, Pa
e	Average velocity error, %
Fr	Froude number
g	gravity acceleration vector, m s^{-2}
G_b	Generation of turbulence kinetic energy due to buoyancy, kg m^{-1} s^{-3}
G_k	generation of turbulence kinetic energy due to the mean velocity gradient, kg m^{-1} s^{-3}
HGV	heavy good vehicle
k	kinetic energy, J/kg
L	characteristic length, m
p	pressure, Pa
Pr	Prandtl number
Re	Reynolds number
S_{fan}	momentum source term, N m^{-3}
T	temperature, K
Y_M	contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate
v	velocity vector, m s^{-1}
Greek symbols	
α	pitch angle, °
ε	turbulent dissipation rate, J kg s^{-1}
λ	scale factor
μ	turbulent viscosity, Pa s
ρ	density, kg m^{-3}
σ_k	turbulent Prandtl number for k
σ_ε	turbulent Prandtl number for ε
Subscripts	
av	average
f	full scale tunnel
fl	fluid
s	model scale tunnel
t	turbulent

2. Methodology

2.1. Tunnel and ventilation system detail

The analyzed physical domain is shown in figure 1. It consists in a one-way road tunnel 800 m long, equipped with four jet fans arranged in longitudinal ventilation system; all jet fans were placed at 5.60 m above tunnel floor. The first and the fourth fan (FAN1 and FAN4), were installed at a distance of 100 m from the entry and exit sections of the tunnel, respectively. The horizontal distance between two subsequent fans was equal to 200 m. This spaced distance was chosen in order to the momentum provided from a jet fan to the accelerated air (the primary air flow), was completely transferred to the tunnel air (secondary air flow) before the fluid had reached the subsequent jet fan. Geometrically, the tunnel was composed by four identical modules each 200 m long with a circular cross section of radius equal to 5.05 m, as shown in figure 1. The presence of Heavy Good Vehicle (HGV) obstruction (placed at 450 m from tunnel entrance) was simulated in order to study the kinematic similitude also in presence of fluid dynamic disturbances.

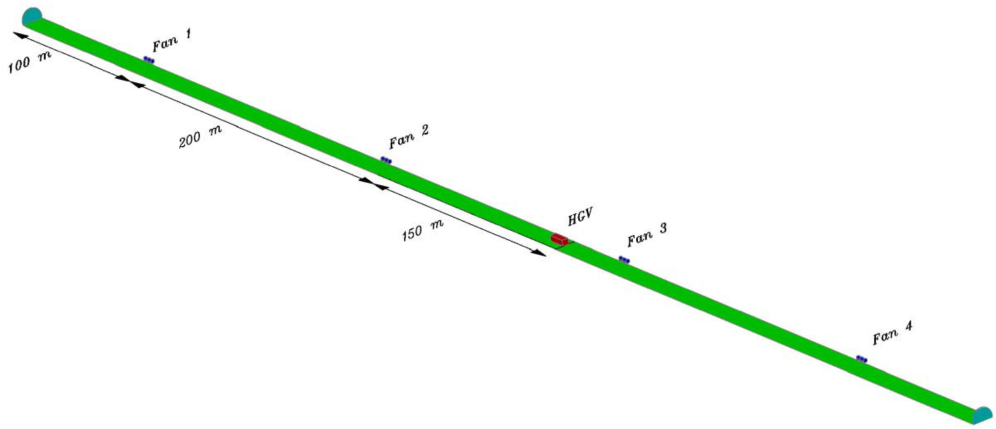

Fig.1. sketch of the road tunnel model analyzed.

2.2. Froude's scaling procedure

In order to create a scaled model structure, three kinds of similitude have to be preserved: geometrical, kinematic and dynamic one. In this study it was been adopted the Froude's scaling procedure and CFD simulations were carried out in order to evaluate this scaling procedure when impulsive fan was considered.

The reduction scale mainly depends on possibility to preserve fluid dynamic behavior (laminar or turbulent), on economic and layout availabilities. For a first analysis, considering the layout availability, it was assumed a geometric scale equal to 50, where the geometric scaling factor, indicated with the symbol λ, is given by the ratio between the characteristic lengths of the two systems, as presented in equation (1):

$$\lambda = \frac{L_f}{L_m} \tag{1}$$

The dynamic similitude was obtained preserving the Froude number defined (equation 2) as the ratio between the gravity forces and the inertia forces due to the ventilation air flow [7].

$$Fr = \frac{v^2}{gL} = \frac{\textit{inertial forces}}{\textit{gravity forces}} \tag{2}$$

Other non-dimensional numbers should be preserved as the Reynolds number, which characterizes the ratio between the inertial forces and the viscosity forces. In practice, it is possible to preserve both characteristic numbers only in few cases. Models, usually, are scaled preserving the Froude number only. As pointed out by Carvel [8] and Karaaslan et al. [9] if the scaling factor is not very high, the equality of Froude number is a reasonable approach to define similarity between the full scale and reduced scale tunnel model.

Since, the Froude number has to be preserved in the scale model tunnel as reported in the relationship (3):

$$Fr = \frac{v_{av,m}^2}{gL_m} = \frac{v_{av,f}^2}{gL_f} \tag{3}$$

from which

$$v_{av,m} = v_{av,f}\sqrt{\frac{L_m}{L_f}} \tag{4}$$

that is necessary to preserve the Froude number [7]. To scale a pressure it is possible to use the relationship (5), for an uncompressible flows:

$$\frac{p_m}{p_f} = \frac{1}{\lambda} \tag{5}$$

3. Mathematical model

The governing equations, for the fluid region in steady state regime, are time-averaged mass, Navier-Stokes and energy equations combined with k-ε realizable turbulence model [10]. For the sake of simplicity, notations used in this study neglect the superscript bar usually employed to denote time-averaged dependent variables. In following equation model with constant value is reported:

$$\nabla \cdot (\rho \vec{v}) = 0 \tag{6}$$

$$\nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot \mu\left[\left(\nabla \vec{v} + \nabla \vec{v}^T\right) - \frac{2}{3}\nabla \cdot \vec{v} I\right] + \rho \vec{g} + \vec{S}_{fan} \tag{7}$$

$$\nabla(\rho \vec{v} T_{fl}) = \nabla\left[\left(\frac{k_t}{c_p} + \frac{\mu_t}{\mathrm{Pr}_t}\right)\nabla T_{fl}\right] \tag{8}$$

$$\nabla \cdot \left(\rho \varepsilon \vec{\mathrm{v}}\right) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b \tag{9}$$

$$\nabla \cdot \left(\rho k \vec{v}\right) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k - G_b - \rho\eta + Y_M \tag{10}$$

The turbulent dynamic viscosity, μ_t, is to be predicted from the knowledge of the turbulent kinetic energy, k, and the turbulent kinetic energy dissipation rate, ε. The transport equations for k and ε were formulated using the Realizable k-ε model. They can be derived from Navier-Stokes equations, but the constants in ε equation are derived using Realizable theory, as suggested in [11]. The constants in the model are $C_{1\varepsilon} = 1.44$; $C_2 = 1.9$; $\sigma_k = 1.0$; $\sigma_\varepsilon = 1.2$, $C_1 = \max\ [0.43,\ \eta/(\eta+5)]$ with $\eta = S(k/\varepsilon)$, $C_{3\varepsilon} = 1$ for buoyant shear layers for which the main flow direction is aligned with the gravity direction. For buoyant shear layers that are perpendicular to the gravitational vector, $C_{3\varepsilon} = 0$.

4. Full and scaled model discretization and boundary conditions

Each module of investigated physical domain was divided in two zones (figures 1 and 2) and different mesh shapes were applied: in the zone that include the jet fan and the Heavy Good Vehicle (zone 1, table 1) was adopted a tetrahedral mesh type, in the rest of domain (zone 2, table 1) was adopted an hexahedral mesh type. To take into account the effect of tunnel ceiling and floor roughness height on the velocity field, for both full and reduced scale models, y^* equal to 150 combined with the wall standard function was adopted to reduce the computational resources [10].

Table 1. Type and mesh size.

Zone	Mesh type	edge / m
1	tetrahedral grid	0.3
2	hexahedral grid	0.4 along x- axis

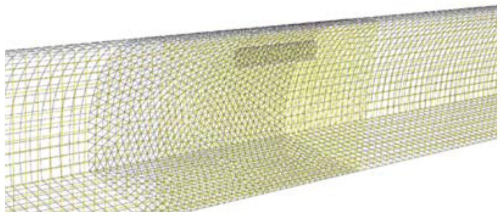

Fig. 2. sketch of the road tunnel model analyzed.

The reduced scale model was obtained from full scale reducing both tunnel geometry and the mesh size of 1/50 factor. In order to find out the boundary conditions for reduced scale model, a preliminary CFD analysis for full scale model was been carried out (see table 2). This analysis provided the average velocity value in the tunnel volume equal to 3.70 and 3.80 ms^{-1} for traditional and alternative ventilation systems, respectively.

In order to preserve the kinematic similitude, average volume velocity values, in reduced-scale model, $v_{av,\,m}$ equal to 0.52 ms^{-1} and 0.54 ms^{-1} for traditional and alternative system, respectively, were expected in according to equation (4). So, different values of pressure drop and wall roughness for both ceiling and floor was considered. In tables 3 and 4 the pressure drop and roughness values, obtained by numerical analysis, were reported for both traditional and alternative jet fans, respectively.

Table 2. Boundary conditions for the preliminary CFD analysis.

	Full scale
Solver	Pressure based
Fluid's material	Air
Pressure drop for traditional fan	2300 Pa
Pressure drop for the alternative fans	1700 Pa
Fluid temperature	300 K
Gauge pressure at Inlet and outlet tunnel section	0 Pa
Floor roughness,	0.01 m
Roof roughness	0.03 m

Table 3. Pressure drop values (Fr_{av}) for traditional jet fan.

	Traditional fans					
	Δp(Pa)	v (ms^{-1})	Roof roughness (m)	Floor roughness (m)	Re	Fr_{av}
Full scale tunnel	2300	3.70	0.03	0.01	$1.7 \cdot 10^{6}$	0.23
Scaled tunnel	50	0.50	$1.0 \cdot 10^{-08}$	$1.0 \cdot 10^{-08}$	4700	0.20
	64	0.51	0.0015	0.0005	4800	0.21

Table 4. Pressure drop values (Fr_{av}) for alternative jet fan.

	Alternative fan					
	Δp(Pa)	v (ms^{-1})	Roof roughness (m)	Floor roughness (m)	Re	Fr_{av}
Full scale tunnel	1700	3.80	0.03	0.01	$1.8 \cdot 10^{6}$	0.24
Scaled tunnel	42	0.49	$1.0 \cdot 10^{-08}$	$1.0 \cdot 10^{-08}$	4600	0.20
	46	0.51	0.0015	0.0005	4800	0.21

In the table 5, the boundary conditions imposed in the CFD analysis, obtained by means the preliminary analysis, were summarized. Also, the following hypotheses were imposed:

- air properties, considered as an ideal gas, are assumed constant with the temperature and valued at the ambient temperature equal to 300 K;
- the swirl component due to the air flux passage through the fan wheel is neglected;
- the pitch angle of the jet fan in the simulation represents the real baffle plates;
- turbulent model: k-ε realizable.

Table 5. Boundary conditions.

	Full scale	Reduced scale
Solver	Pressure based	Pressure based
Fluid's material	Air	Air
Pressure drop for traditional fan	2300 Pa	64 Pa
Pressure drop for the alternative fans	1700 Pa	46 Pa
Fluid temperature	300 K	300 K
Gauge pressure at Inlet and outlet tunnel section	0 Pa	0 Pa
Floor roughness,	0.01 m	0.0015 m
Roof roughness	0.04 m	0.0005 m

5. Results

In the following the distribution of x velocity profiles and turbulent intensity fields, predicted for both the full and reduced scale models for ventilation systems considered, are presented and the full scale velocity profile has been scaled (in according with equation 4) to highlight the differences with reduced scale numerical model.

The figures 3 and 4 show the comparison between the velocity profile of scaled and the full scale tunnel in different sections, where high gradient velocity occurs for both ventilation systems. The figure 3 shows velocity profiles evaluated on symmetry plane (z=0) for different x distances from the HVG, along y axis (from floor y=0 m to ceiling y=6.42 m). In particular, the figure 3a shows that the velocity profile of the full scale and reduced scale are quite similar for both ventilation systems, in spite of the high velocity gradients due to the reduction of the cross section due to the presence of the HVG. Also, the figure 3a shows the quite fully developed x velocity profiles upstream the vehicle; in this section the velocity profiles are quite overlapped. The figures 3b and 3c show that the velocity profile, where high gradient velocity occurs, are slightly different.

In table 6 are summarized the average x-velocity errors, between full and reduced scale, evaluated by equation 11; it is possible read that errors in areas far away from to disturbances are ranged in 1.9% to 5.1% and 3.9% to 5.6% for traditional and alternative systems, respectively.

$$e\% = \frac{1}{N}\sum \frac{v_{x-scaled,N} - v_{x-fully,N}}{v_{x-fully,N}} \cdot 100 \quad (11)$$

where N is the number of cell along y-axix.

Table 6. Velocity average errors between full and scaled model for both ventilation systems.

Axial distance (m)	Average error roughness 0.0015 m (%)	Average error roughness 0.0015 m (%)
	Traditional ventilation system	Alternative ventilation system
420	1.9	3.9
430	2.1	4.1
440	2.8	4.4
450	4.4	4.3
460	2.9	6.6
470	4.4	5.6
480	5.1	5.1
490	5.2	4.8
498	9.6	8.7
502	10.8	8.8
512	10.2	16.4
522	9.8	12.6
552	7.3	7.5
602	0.4	4.8
652	3.0	5.6

Moreover the figure 3 shows that near the vehicle, the reduced scale model equipped with alternative jet fan shows higher errors with respect to the traditional ventilation system. The higher difference for alternative systems in presence of obstacle on the floor is due to the flow air direction inclined toward the floor.

The figure 4 shows velocity profiles evaluated on symmetry plane (z=0) for different x distance downstream an jet fan position along y axis (from floor y=0 m to ceiling y=6.42 m). In particular, the figures 4a and 4d referring to profile on outflow jet fan section show that, for both ventilation systems, the velocity of scaled model is higher than full model one in area's points close to the outflow section; this is due to the greater value of pressure drop chosen with respect to that suggested by Froude scaling model. In this case the average errors are 10.8% and 8.8% for traditional and alternative systems, respectively. Far away from the jet fan section, the velocity error diminishes reaching 0.4% and 4.8% values for traditional and alternative systems, respectively.

The figure 5 shows the turbulent intensity field on symmetry plane (z=0) downstream jet fan, in particular the comparison between full and scaled model for traditional (a, and b) and alternative (c and d) jet fans is presented. As expected, the turbulent intensity values were different for full and reduced scale model because the Reynolds number was not preserved in the reduced scale model for both ventilation systems. The figure 5 shows that the turbulent intensity field was lower in the reduced scale with respect to the full scale model (one order of magnitude), but they presented similar behaviors.

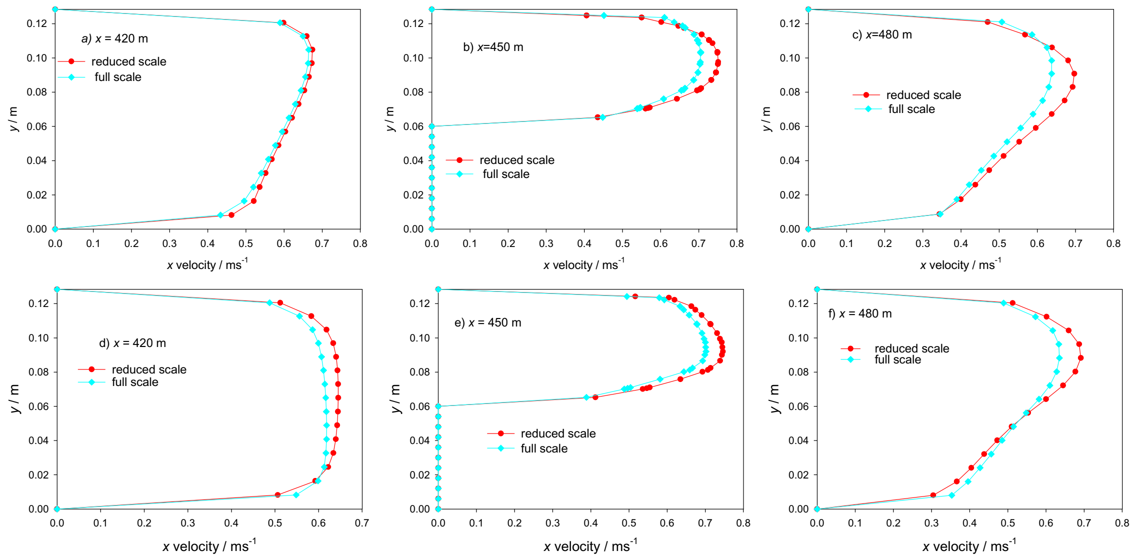

Fig. 3. axial velocity profile on symmetry plane (z=0) across HVG for different x-positions, fixed the roughness value and the pressure drop (see table 5): comparison between full and reduced scale model for traditional (a,b and c) and alternative (d, e and f) jet fan.

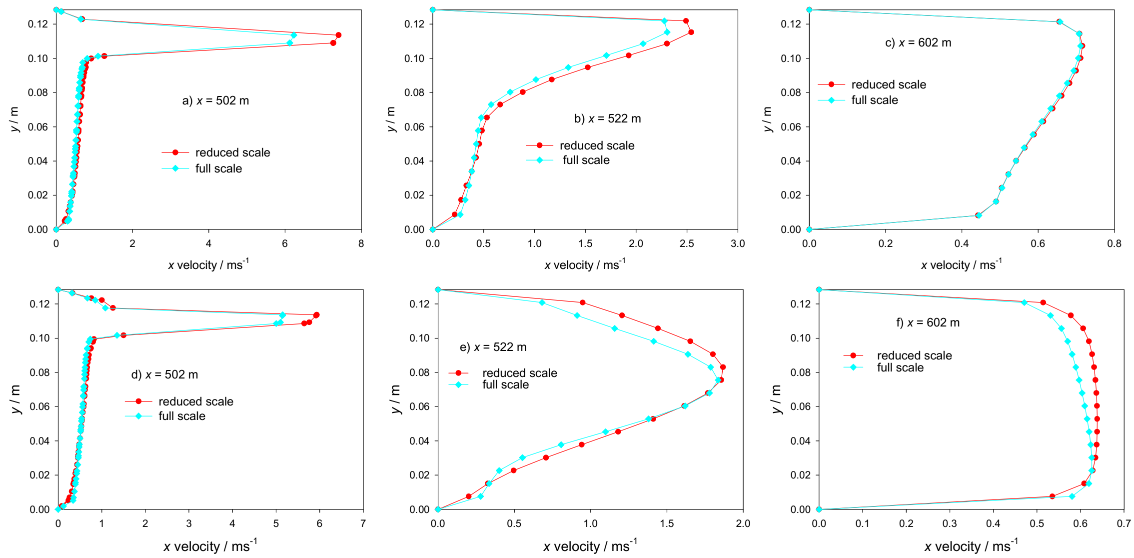

Fig. 4. axial velocity profile on symmetry plane (z=0) for different x-positions downstream jet fan fixed the roughness value and the pressure drop (see table 5): comparison between full and reduced scale model for traditional (a,c and e) and alternative (b,d and f) jet fan.

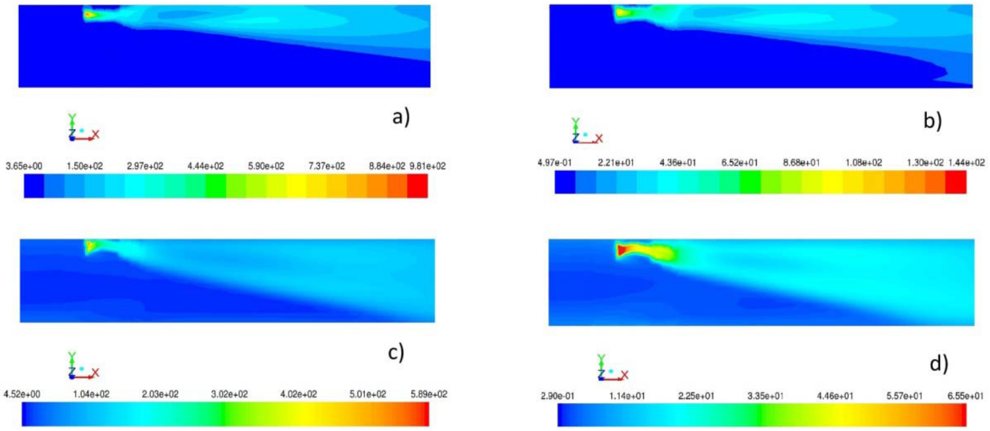

Fig. 5. turbulent intensity field on symmetry plane (z=0) downstream jet fan: comparison between full and scaled model for traditional (a, and b) and alternative (c and d) jet fan.

6. Conclusion

CFD's analysis was carried out in order to evaluate the scaling procedure for road tunnel equipped with realistic impulsive ventilation systems. A comparison between x-velocity values for full and reduced scale models were provided to evaluate the scaling effect on the velocity field.

Results, in terms of x velocity, showed that reduced scale procedure well predicts the flow field when Froude method was applied and properly roughness value were employed. Differences between air velocity values, predicted with the full scale model and reduced scale, were larger in areas with high velocity gradient (presence of obstruction or zone with momentum source). The average x-velocity errors between full and reduced scale models are within 10% for traditional ventilation system and 16% for the alternative, whereas in areas far away from to disturbances are up to 5.6%.

References

[1] Witt KC, Schütze J. Bend it like a Banana. Proceedings BHR Group, 12th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels 2006.

[2] Betta V, Cascetta F, Musto M, Rotondo G. Fluid dynamic performances of traditional and alternative jet fans in tunnel longitudinal ventilation systems. Tunneling and Underground Space Technology 2010; 25: 415-422.

[3] Li YZ, Lei B, Ingason H. Study of critical velocity and backlayering length in longitudinally ventilated tunnel fires. Fire safety journal 2010;45(6): 361-370.

[4] Vauquelin O. Experimental simulations of fire-induced smoke control in tunnels using an "air-helium reduced scale model": Principle, limitations, results and future. Tunnelling and underground space Technology 2008; 23.2: 171-178.

[5] Kurioka H, Oka Y, Satoh H, Sugawa O. Fire properties in near field of square fire source with longitudinal ventilation in tunnels. Fire Safety Journal 2003; 38(4): 319-340.

[6] Wu Y; Bakar MA. Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity. Fire Safety Journal 2000; 35.4: 363-390.

[7] Roh, JS, Ryou HS, Kim DH, Jung WS, Jang, YJ. Critical velocity and burning rate in pool fire during longitudinal ventilation. Tunnelling and underground space technology 2007; 22(3): 262-271.

[8] Carvel, OR. Fire Size in Tunnels. PhD. Thesis,Heriot-Watt University: 2004.

[9] Karaaslan, S, Hepkaya, E, Yucel N. Cfd simulation of longitudinal ventilation systems in a scaled short tunnel. Journal of thermal science and technology 2013; 33.1: 63-77.

[10] Fluent User's Guide, Fluent Inc., 2005.

[11] Shih TH, Liou WW, Shabbir A, Yang Z, Zhu J. A new k-ϵ eddy viscosity model for high Reynolds number turbulent flows. Computers & Fluids 1995; 24(3): 227-238.