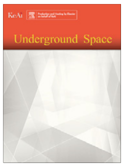
Available online at www.sciencedirect.com

ScienceDirect

Underground Space 11 (2023) 153–170

www.keaipublishing.com/undsp



CFD simulations of a semi-transverse ventilation system in a long tunnel

Alireza Dastan [a], Mohammad Rahiminejad [b], Mozhgan Sabz [b,c], Meisam Abbasi [b], Ali Mokhtari [b], Sasan Sadrizadeh [d,*], Omid Abouali [b,d,*]

[a] *Department of Mechanical Engineering, Faculty of Engineering, University of Isfahan, Isfahan 81746-73441, Iran*
[b] *School of Mechanical Engineering, Shiraz University, Shiraz 71936-16548, Iran*
[c] *Department of Mechanical Engineering, University of Alberta, Edmonton T6G 2R3, Canada*
[d] *Department of Civil and Architectural Engineering, KTH University, Stockholm 10044, Sweden*

Received 1 March 2022; received in revised form 22 October 2022; accepted 16 December 2022
Available online 23 April 2023

Abstract

In the present work, a semi-transverse ventilation system in a long tunnel with a length of 4.9 km, as a complex case study, is numerically studied by performing a set of three-dimensional steady incompressible computational fluid dynamics (CFD) simulations. The ventilation system consisted of a ceiling duct connected to two axial fans at the ending portals, and a series of jet fans in the main tunnel for supporting airflow in the desired direction. To focus on what can and cannot be achieved in commissioning tests, the ventilation system's performance in various scenarios is numerically evaluated with two different tunnel states; empty tunnel and complete traffic congestion with 1176 stationary vehicles – which is almost impossible to evaluate during a commissioning test. By considering two hypothetical locations for the extraction zone from the main tunnel (in a distance of 450 and 1000 m from one portal), it is shown that the required number of jet fans in a traffic condition drops from 57 for the first extraction location to 43 (25% decrease) when the ventilation system extracts from the second zone. We show that if only the close axial fan to the extraction zone is activated, the required number of jet fans reduces by 56% and 72% for the first and second extraction locations, respectively. This finding can provide a cheaper and easier controlling scenario for emergency ventilation.

Keywords: CFD; Semi-transverse ventilation; Tunnel; Ceiling duct; Traffic

1 Introduction

Nowadays, tunnels play a vital role in reducing traffic and facilitating transportation. There are several long tunnels in many countries that connect cities or different areas. As human safety is a considerably important issue in tunnels, and numerous tunnel fire incidents have occurred, the importance of an appropriate and acceptable ventilation system is undeniable. Many studies have been done in this regard, and it has been proved that computational simulation is a powerful and significantly cost-effective method for evaluating ventilation systems in tunnels. Some numerical studies investigated natural ventilation in tunnels (Fan et al., 2018; Rafiei, 2015; Wang et al., 2019; Zhong et al., 2016; Zhou et al., 2019), but this type of ventilation is not convenient in long tunnels. On the other hand, mechanical ventilation systems including longitudinal ones which are appropriate for unidirectional, short, or medium-length tunnels (Betta et al., 2009; Cascetta et al., 2016; Chow et al., 2010; Du et al., 2015; Fang et al., 2019; Lee & Ryou, 2006; López González et al., 2014; Vega et al., 2008),

---

* Corresponding authors at: Department of Civil and Architectural Engineering, KTH University, Stockholm 10044, Sweden (O. Abouali); School of Mechanical Engineering, Shiraz University, Shiraz 71936-16548, Iran (S. Sadrizadeh).
E-mail addresses: ssad@kth.se (S. Sadrizadeh), abouali@shirazu.ac.ir (O. Abouali).

https://doi.org/10.1016/j.undsp.2022.12.004

semi-transversal in which either supply or exhaust duct should be installed along the tunnel (Ballesteros-Tajadura et al., 2006; Lin et al., 2014, 2016; Zhang et al., 2014), and full-transversal having both supply and exhaust ducts along the tunnel (Du et al., 2016; Levy et al., 1999; Sturm et al., 2012) can be efficient for smoke exhaust in a case of a fire emergency (National Fire Protection Association [NFPA], USA, 2020a).

According to the literature, changing the number of active fans during the ventilation process profoundly affects ventilation quality. For example, Myrvang and Khawaja (2018) have performed a special investigation to find a more accurate correlation between air velocity and the number of stages of ventilation system in tunnel ventilation by numerical and experimental models. They activated a different number of fans and showed that a linear increase of the air velocity would not occur by enhancing the number of ventilation stages because of an exponential trend of pressure losses. In another study, Stroppa (2004) simulated tunnel ventilation with a one-dimensional time-dependent numerical model to reduce computational costs. They studied different fan stages to achieve the CO and contamination concentration under normal conditions, and reduced smoke spreading was evaluated under fire conditions. Stroppa (2004) compared their simulation results with experimental site measurements and confirmed that simulation is an appropriate and cost-effective method for understanding the main effects. Colella et al. (2010) performed a multi-scale simulation on the ventilation system in two unidirectional, two-lane, and partly underwater tunnels, having a complex semi-transverse ventilation system supported by some jet fans at Dartford, UK. They evaluated various combinations of jet fans and found that an acceptable condition can be generated in one of the tunnels while some of the jet fans are turned off. However, in the other tunnel, more jet fans were necessary for appropriate ventilation. The effect of changing the operation mode of jet fans on the waiting time for a steady state of airflow has been investigated by Król and Król (2018). They used ANSYS Fluent for modelling and considered the effect of height on dropping static pressure, the ambient weather condition, and the wind influence on the airflow relaxation time.

The challenges in the ventilation of long tunnels can be worsened when heavy traffic occurs in the tunnel. The occurrence of the traffic can be due to an incident in the tunnel, or a situation that happened downstream outside of the tunnel. Not only in a fire emergency case but also in traffic without a fire incident, the ventilation system should be able to maintain safety in the tunnel. For instance, vehicular emissions should be removed by ventilation facilities to preserve acceptable air quality in the tunnel. During bumper-to-bumper traffic, the tunnel temperature should be controlled to prevent high temperatures downstream. Moreover, when a fire occurs, the ventilation system must secure a safe zone for passenger evacuation, regardless of traffic conditions.

Undeniably, studying the function of ventilation systems under different traffic conditions is an essential issue on which several studies have been carried out (Cascetta et al., 2019; Eftekharian et al., 2015; Musto et al., 2020). For example, Tao et al. (2016) investigated CO_2 dispersion in a short tunnel under traffic congestion numerically. They studied three different traffic conditions under different car speeds of 0, 10, and 40 km/h with tunnel fan-off/on conditions in a three-dimensional model. The results showed that in stationary traffic and without mechanical ventilation, CO_2 level is unsafe in the downstream region. In contrast, at a traffic speed of 40 km/h, the elimination of contamination is acceptable. However, at the speed of 10 km/h, CO_2 concentration was high in the downstream and tunnel exit region. In another work, a correlation between different traffic jam conditions and pressure loss in the tunnel has been established by Musto et al. (2020) to estimate the required thrust for jet fans for tunnel ventilation. They modelled different traffic jam conditions and air velocities and derived an equation to calculate pressure loss as a function of the traffic and average air velocity.

There are two different approaches regarding emergency ventilation in tunnels. In one, back-layering (movement of smoke upstream of the fire against the normal direction of ventilation airflow) is tried to be prevented by applying a critical air velocity in relatively short or mid-range tunnels. This approach provides a safe region upstream of the fire for passenger evacuation and rescue team access. However, in longer tunnels in which pushing the smoke downward may cause the regions far from the incident point to become dangerous due to the accumulation of smoke, the fire products should be extracted as close as possible to the fire – this approach is called single point (zone) extraction by NFPA, USA (2020a). In transverse or semi-transverse ventilation systems, confinement of the smoke between open dampers (extraction zone) is necessary to reduce the number of tunnel users exposed to fire products. This is achieved by blowing low-velocity air from both sides of the fire through the function of additional jet fans or supply air fans. In this case, it is recommended that similar airflow velocities from both sides of the fire are maintained in the tunnel to prevent smoke propagation while maintaining smoke stratification even in the relatively short length of the tunnel (nominally, the length between the dampers being opened for smoke extraction) (Almbauer et al., 2004; World Road Association [PIARC], 2011). This might impose operational and control challenges on the ventilation system. Although the requirements for temperature sensors, closed-circuit TV, and human operators for fire detection, damper actuators (remotely controlled dampers), jet fans which can be operated in both directions, and flow meters cause the ventilation system and its controlling scenarios to be very complicated, this ventilation method was proven to be applicable in different tunnels (Yuan et al., 2016).

Sturm et al. (2017) studied different methods of fire ventilation and requirements for sensors and technologies, where they found that in many tunnel fires 'low velocity' philosophy (in contrast with the critical velocity approach) is an appropriate method for preventing propagation of smoke and self-saving. In a mixed computational and experimental study, Zhong et al. (2017) observed that small longitudinal velocity could not prevent back-layering during tunnel fires, and large longitudinal velocity (about 1.5 times back-layering critical velocity) causes bifurcation flow, in which the ventilation air divides into two streams and flows alongside the walls. Thus, finding an appropriate velocity to prevent smoke propagation is necessary. Liu et al. (2019) evaluated the elimination of smoke during a fire occurrence in a tunnel. They used tunnel ventilation fans (TVFs), jet fans, and ceiling ducts to exhaust produced smoke as soon as possible and reduce smoke propagation. They studied various cases with different locations for fire. Their results showed that semi-transverse ventilation comprising jet fans, TVFs, and downstream ceiling extraction is the best ventilation during tunnel fire.

In addition to the mentioned function of ventilation systems in the extraction of smoke and other hazardous materials from the tunnel, their role in controlling air temperature is also critical. High-temperature air may lead to malfunctioning of the instruments including sensors, actuators, and cabling system installed in the tunnel. This is particularly important for the future design of tunnels where the presence of, for example, vehicles with hydrogen fuel cells may increase in the fleet. The characteristics of a fire incident of these types of vehicles might be different from those of fossil fuel-burning vehicles in terms of heat release rate, temperature rise, and smoke production (the smoke from other materials than fuel including tyres, seats, etc. is still a common concern) (Gu et al., 2020; Shibani et al., 2022a, 2022b).

Internationally recognized codes and standards related to the function of tunnels, such as those issued by NFPA, USA (2020a, 2020b) and PIARC (2011), generally recommend performing a commissioning test after when the design of the emergency ventilation system is approved by one-dimensional (1D) engineering analyses and three-dimensional (3D) computational fluid dynamics (CFD) evaluations. These commissioning tests are usually carried out in a no-fire situation which is called a cold-test (or ultimately cold air with smoke, cold-smoke). The academic literature suffers from the lack of a thorough study on different aspects of a commissioning test which might not or cannot be similar to a real emergency and even the simulations performed in the previous stages of the design. Here, we show the limitations that the outcomes of commissioning tests of a tunnel can have by comparing the results of simulations, some of which (such as traffic jam conditions) are not achievable even in a real-world cold-test. Through these, some important issues related to the function of a tunnel ventilation system are also revealed. It should be noted that the results presented here might be different from those that can be achieved in an incident with fire, as the related effects including temperature rise due to the release of heat and therefore air density changes, chimney effect, smoke propagation, etc. are not considered. Dimensional analysis can be a good tool for extending the results of a particular case to other similar ones. As most of these non-dimensional parameters (apart from general ones such as Reynolds number and friction factor (Eftekharian et al., 2015)) are provided for the case of fire which is not considered in the cold-test simulations of the current work, the readers are encouraged to see the related references (Hu et al., 2013; Savalanpour et al., 2021).

In the present work, a series of cold-tests are numerically investigated in a 4900 m tunnel equipped with a semi-transverse ventilation system. In these tests, the performance of the ventilation system in an emergency scenario happening at a close distance to the inlet portal of the tunnel is studied. The ventilation system is to suck out the cold air from the extraction zone around a hypothetical incident by exploiting the portal exhaust fans and a number of jet fans in the main tunnel for balancing the air velocity reaching the zone from both sides. Two situations including the empty tunnel and full traffic congestion (which is almost unachievable in a real cold-test) are simulated to emphasize the aspects which might be ignored in a commissioning test. Furthermore, air leakage through some parts of the ventilation system (drop ceiling and closed dampers) are considered in the simulations. To the best of our knowledge, a numerical study with these details, investigating the effects of traffic on the ventilation system, has not been reported in the literature. The results will hopefully lead to a better decision-making process for controlling scenarios of an emergency situation.

2 Numerical methods

2.1 Model description

Taloon tunnels are a triplet parallel tunnel system (2 one-way main tunnels and one service tunnel in between) with a length of 4900 m located in Tehran-North freeway, Iran. A three-dimensional computational model for one of the main tunnels (Tehran to North) was considered for this study. The cross-section of the tunnel is divided by a drop ceiling structure to provide a ceiling duct for the semi-transverse ventilation system (see Fig. 1). The effective cross-section (below the ceiling duct) is 62.1 m^2 with a maximum height of 5.7 m and maximum width of 11.9 m which provides a two-lane pass-way (3.65 m each) for the vehicles. Tehran portal elevation from the sea level is roughly 2244 m with a 2% negative slope in Tehran to the North direction. By ignoring the negligible curvatures in the horizontal plane of the tunnel, it is modelled as a straight tunnel without considering the longitudinal slope (we are performing cold air tests). In addition, for simplification, the lateral 1.5% slope of the road was also ignored. Two computational boxes at two ending portals with dimensions of approximately 50 m × 50 m × 30 m were added

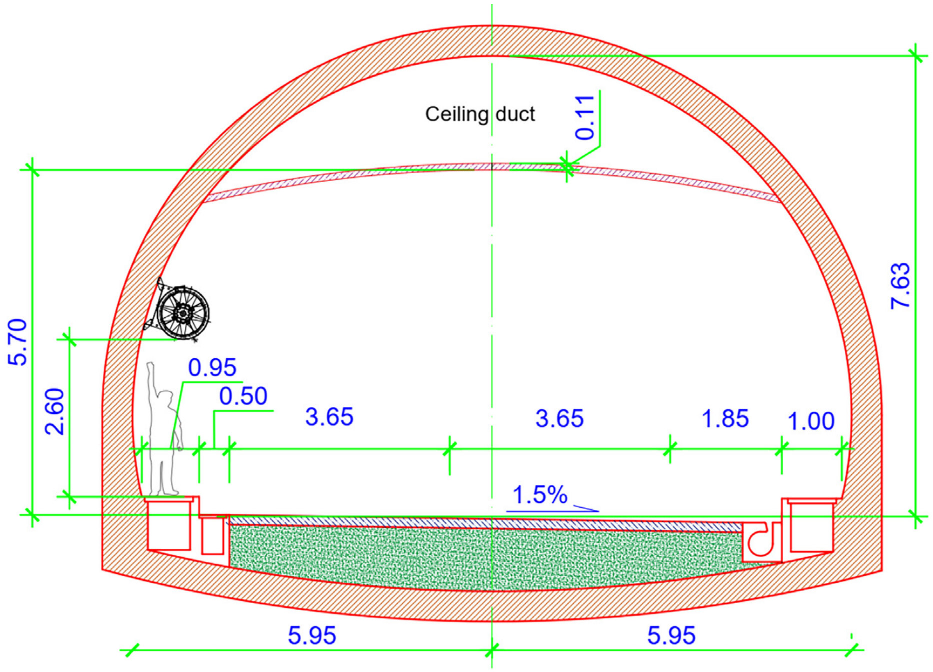


Fig. 1. Tunnel cross section (Unit: m).

to the computational domain (Fig. 2(a)) to accurately impose the boundary conditions at the tunnel portals.

2.1.1 Ceiling duct

A drop ceiling with a thickness of 11 cm was modelled throughout the tunnel (Fig. 2(b)). This structure causes the ceiling duct cross-section above the dropped ceiling to be 11.9 m^2. The leakage through this ceiling was modelled using a porous medium approach that allows airflow leakage based on the pressure difference on both sides. The volume flow rate passed through per unit length of the drop ceiling ($\frac{\mathrm{m}^3/\mathrm{s}}{\mathrm{m}}$) is approximated by the following empirical equation (according to the data provided by the designer):

$$q' = 6 \times 10^{-4}\sqrt{\Delta p}, \tag{1}$$

where Δp is the pressure difference between the tunnel and the ceiling duct. This effect is added to the simulations by considering the 11 cm zone of the drop ceiling as porous media. By considering a homogenous porous medium, a source term given by the following equation is added to the momentum equation (Ansys Fluent, 2015):

$$S_i = -\left(\frac{\mu}{\alpha}u_i + \frac{1}{2}C\rho|u|u_i\right), \tag{2}$$

where μ, ρ, and u_i are the dynamic viscosity, density, and velocity components, respectively. For the porous medium, α is the permeability, and C is the inertial resistance factor. In a simple form of momentum equation, the source term and pressure drop can be related as

$$\Delta p = -S_i\Delta n, \tag{3}$$

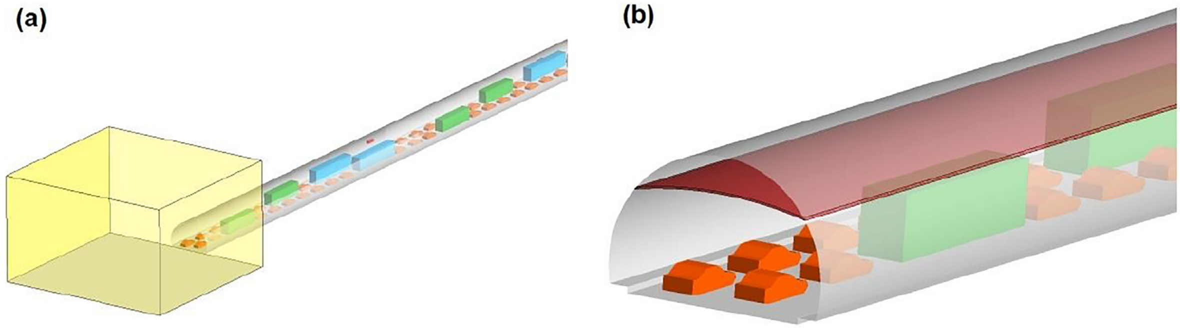


Fig. 2. (a) Computational box (colored in yellow) attached to the ending portals of the tunnel for applying boundary conditions, and (b) a close view of the tunnel and the drop ceiling, colored in dark red.(For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

where Δn is the thickness of the region. Combining Eqs. (1)–(3) together leads to the values of $\frac{1}{\alpha} = 0$ m^{-2} and $C = 5.2 \times 10^9$ m^{-1} for the drop ceiling.

2.1.2 Ceiling duct dampers

Ceiling dampers are the sites for exchanging air between the main tunnel and the ceiling duct. In the studied tunnel of this work, 45 dampers (numbered from Tehran toward the North portal) with a 6.2 m^2 (3.1 m × 2.0 m) cross-section were installed. While the first one is 275 m away from the inlet portal there is a 100 m distance between two adjacent dampers (consistent with PIARC recommendations (PIARC, 2011)). Moreover, like the dropped ceiling, the possibility of leakage from dampers should be considered when they are in closed mode. The leakage of a closed damper per unit area ($\frac{\text{m}^3/\text{s}}{\text{m}^2}$) is given by (provided by the tunnel designer)

$$q'' = 3 \times 10^{-3} \sqrt{\Delta p}. \tag{4}$$

The same porous media approach as that of ceiling duct is employed for the closed dampers by applying of $\frac{1}{\alpha} = 0$ m^{-2} and $C = 2.2 \times 10^6$ m^{-1}.

In semi-transverse ventilation systems, it is recommended that 1–3 damper(s) in the vicinity of the incident should be opened to extract contaminated air from the region. Here, three consecutive dampers in the region are opened, to reduce the chance of plug-holding in a real emergency, while the rest of the dampers in the tunnel are closed. To incorporate the head loss due to the interaction of airflow with the structure of open dampers, a constant minor loss coefficient of $k = 2$ is applied for the open dampers.

2.1.3 Jet fans

Generally, standard jet fans for tunnel ventilation have a simple cylindrical structure with three main parts: motor and fan blades in the middle part and silencers on both sides. The jet fans used in the Taloon tunnel have a 71 cm inner diameter, producing 700 N thrust in standard condition. The airflow with an average speed of 38 m/s exits jet fans. The jet fans were installed 2.6 m above the left walkway in consecutive order by the respective distance of 80 m, as depicted in Fig. 1. The first jet fan was installed at a distance of 90 m from the inlet; consequently, 60 jet fans are distributed along the tunnel to enable us to simulate various scenarios.

In the simulations, jet fans were considered as a cylinder of a length of 2.41 m (two side parts of 78.5 cm and the middle section of 84 cm) with an inner diameter of 71 cm and thickness of 7.5 cm. The motor and propeller were not included in the modelling, but instead, a constant velocity zone of 38 m/s was applied in the middle cylinder of the jet fan in the desired direction to model the induced air velocity by the function of the jet fan.

2.1.4 Axial fans

In the preliminary design stage, two large axial fans with a nominal flow rate of 137.7 m^3/s at 2500 Pa static pressure were suggested for the semi-transverse ventilation scenario to be installed at each tunnel portal connected to the ceiling duct. The characteristic curve shown in Fig. 3 was used in the simulations by applying appropriate fan boundary conditions to find the operational points of the fan in different scenarios based on the pressure change across them.

2.1.5 Vehicles in the traffic

As the present work aims to shed light on the effects of complete traffic congestion on the semi-transverse ventilation system during a cold-test, stationary vehicles are modelled in the tunnel. In severe traffic conditions, vehicles' presence profoundly affects the ventilation system due to airflow interaction with these obstacles. Here, it is supposed that in the complete traffic congestion, there are 20 stationary passenger cars, two buses, and two trucks in every 100 m of the two-lane tunnel. Consequently, there are 1176 vehicles in the tunnel in congested traffic. In the present work, the passenger cars are assumed to be the car shown in Fig. 4. A simplified model of this car was used in a previous work (Eftekharian et al., 2014), while here, the model is modified to be more similar to real geometry, as studied in the other works (Dastan et al., 2019; Eftekharian et al., 2015). A constant width of 170 cm was considered for the passenger cars, while the maximum height is 141 cm, and the bottom of the car is 19 cm above the road surface.

Additionally, for simulating buses (Volvo B12) and trucks (18-wheeler truck), single boxes were modelled with dimensions of 2.5 m × 12.5 m × 3.4 m (the bottom plane is 40 cm above the road surface) and 2.5 m × 16.3 m × 3.7 m (the bottom plane is 60 cm above the road), respectively. To arrange the vehicles in the tunnel, first, they are randomly distributed in about 100 m of the tunnel length (with a longitudinal distance of about 1.65 m). Then, the achieved pattern is repeated to occupy the entire tunnel.

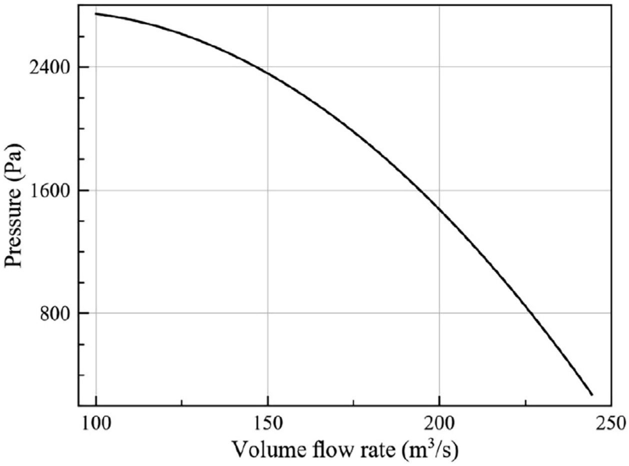


Fig. 3. Characteristic curve of the axial fans.

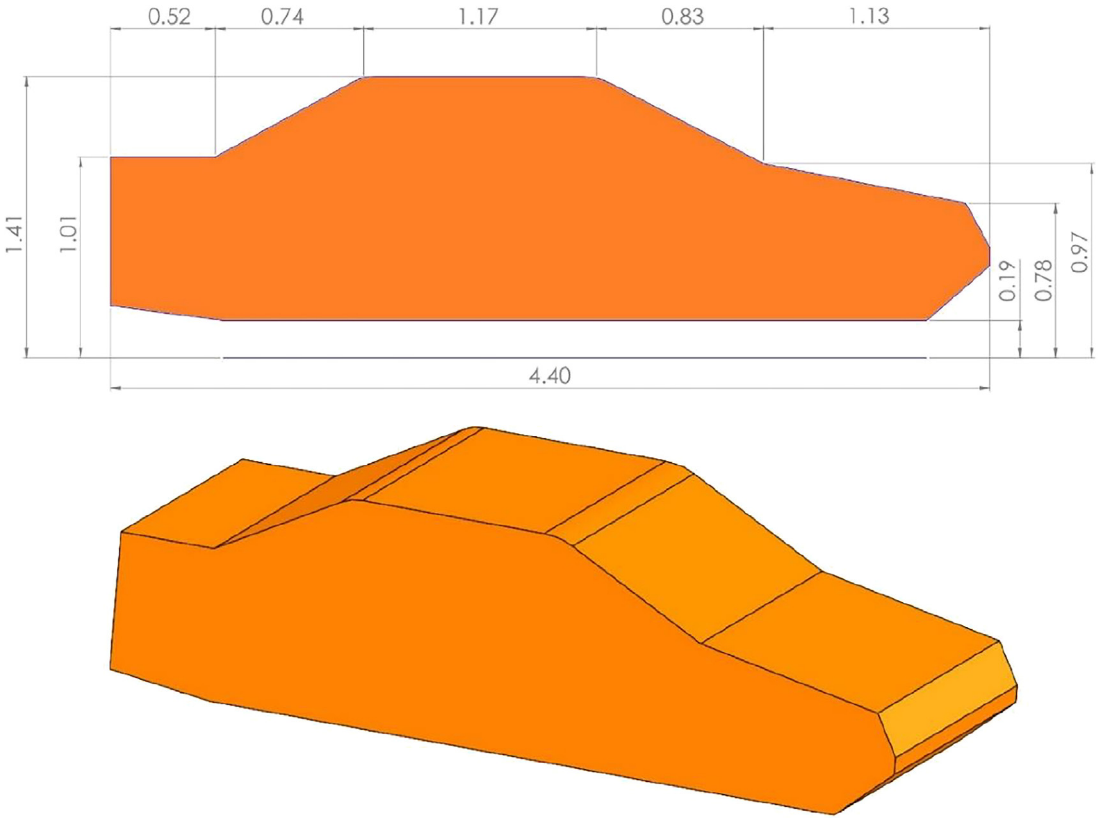


Fig. 4. Two views of the personal car used in the simulations (Unit: m).

Figure 5 illustrates the arrangement of the vehicles in heavy traffic.

It should be mentioned that the arrangement of vehicles in the traffic might influence the function of the ventilation system through the changes which can occur in the interaction of airflow and the traffic. It was shown that the average drag coefficient of personal cars in a platoon can be smaller than that of an isolated vehicle by roughly between 5% and 24% for a range of vehicle distance between 10 and 1 m, respectively (Kaluva et al., 2020). This reduction in the average drag coefficient reaches an asymptotic value in the mentioned range when the number of vehicles in the platoon becomes larger than 6–7 cars. Therefore, here, we expect that most of the cars in large-scale traffic have almost the same effect on the pressure drop of the tunnel. On the other hand, larger vehicle arrangements (buses and trucks) can have much more dominant effects on the function of the ventilation system. In another work on the longitudinal ventilation of the same tunnel (Dastan et al., 2019), it was shown that only by changing the location of larger vehicles to the right lane (for example, by forcing these vehicles to use only the right lane inside the tunnel), while keeping the number of the vehicles in the tunnel unchanged, the airflow rate imposed by 44 jet fans increases by 20%. That is, in the second arrangement, the momentum injected into the domain by the function of jet fans located on the left side of the tunnel is less likely to be dissipated by the interaction with the large obstacles.

2.2 Grid generation

A multi-zone technique was employed to generate a high-quality mesh. For the grid sensitivity study, a shorter tunnel, having stationary traffic described in Section 2.1.5, with 150 m length, was considered. Three different meshes with 2100, 4200, and 6300 3D cells per unit length of the tunnel were produced. The same airflow rate was applied for this tunnel, and the velocity distribution on a predefined line within the tunnel was compared for the generated meshes. Based on the results shown in Fig. 6, the velocity magnitudes on the chosen line do not show a significant difference between Grids 2 and 3. Therefore, the mesh with about 4200 cells per unit length of the tunnel was chosen.

Consequently, a mesh with about 21 million hexagonal cells was generated for the tunnel having stationary traffic. In some simulations presented in the following, an empty tunnel was also studied. A grid consisting of about 12 million cells was generated for this case. Figure 7 illustrates the generated mesh for different parts of the domain.

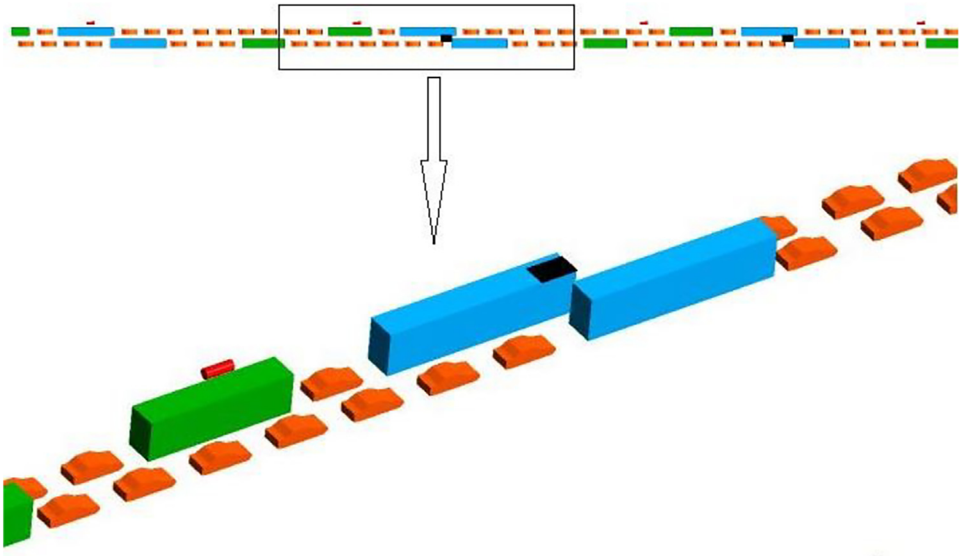


Fig. 5. Internal constellation of the vehicles in the traffic. (Here, the personal cars, buses, and trucks are orange, green, and blue, respectively. The jet fans are shown with small red cylinders on one side of the tunnel, and the dampers are in black.)(For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.).

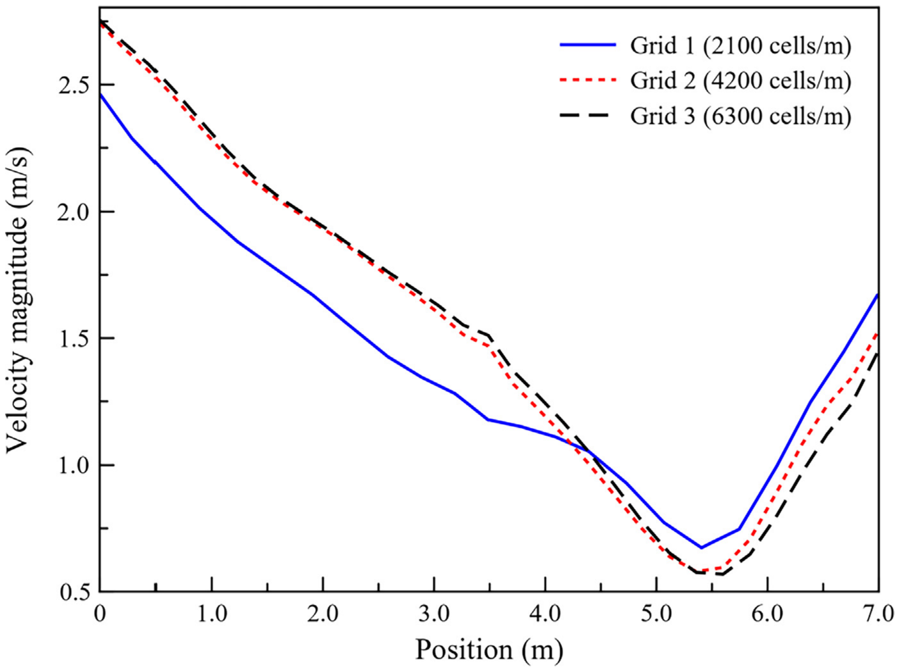


Fig. 6. Velocity distribution on a pre-defined line within the tunnel for three different grids.

2.3 Governing equations and boundary conditions

In the simulations, continuity, and momentum equations are solved numerically for incompressible steady airflow field in the domain:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{5}$$

$$\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + S_{i,} \tag{6}$$

where ρ, u_i, x_i, P, μ, and μ_t are the density, components of the velocity vector, components of location vector, pressure, dynamic viscosity, and turbulence viscosity, respectively. Regarding the choice of turbulence model, three different models from the k-ε family (standard, RNG, and realizable) were tested for a short tunnel with only one stationary car, and the data of the vehicle drag coefficients and tunnel pressure drop was compared. As no significant difference (less than 8%) was observed in the results, the less costly standard k-ε turbulence model was

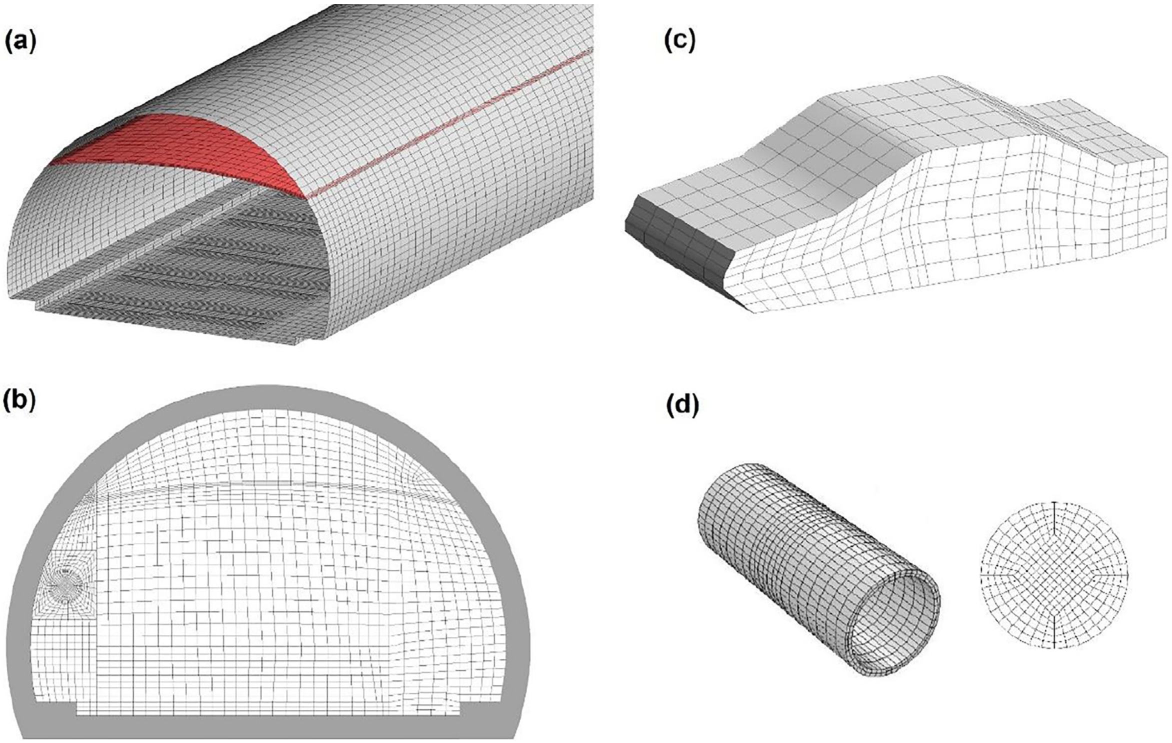


Fig. 7. Grid generated on different parts of the domain. (a) tunnel walls and ceiling, (b) cross-section of the tunnel, where there are no vehicles (the tunnel wall is colored differently for clarity), (c) the surface of the personal car, and (d) surface and cross-section of a jet fan.

chosen for the simulations. As the aim is to simulate the cold-tests, the effect of fire and the released heat was not considered in the simulations presented in this paper and the temperature and density of the fluid are assumed to be constant. Based on the tunnel elevation from sea level, the atmospheric pressure in this area was approximately 77.5 kPa, which leads to the air density of 0.9 kg/m^3 at 300 K. Dynamic viscosity of the air was supposed to be 1.7894×10^{-5} Pa·s. Roughness heights of 3 mm and 1 cm were applied for the tunnel's wall and the road of the tunnel, respectively. Although it seems that these numbers are larger than typical values, they were used to approximate the effects of pressure losses due to small obstacles such as firefighting boxes and lighting systems in the tunnel. No-slip boundary condition was applied on all walls, and zero-gauge pressure is considered on the surfaces of the portal boxes. Fluent software (Ansys Fluent, 2015) is used to solve the equations mentioned above.

2.4 Validation

In order to validate the numerical methods utilized in this study, the combined experimental and numerical work of Cascetta et al. (2016) is used for comparison. In their work, a ventilation system equipped with jet fans in a tunnel with a rectangular cross-section was studied. In the experimental investigation, a setup with a scale of 1/25 was tested. The test tunnel had a cross-section of 0.25 m × 0.50 m and a length of 4 m. One jet fan with a diameter of 0.028 m and length of 0.156 m was installed in the beginning part of the tunnel in a way that its outlet section was 0.8 m away from the tunnel inlet. The jet fan outlet velocity was 8.8 m/s. Roughness heights of 0.0004 and 0.0012 m were considered for the tunnel floor and roof walls, respectively. As the experimental tunnel was part of a longer one, and to incorporate the effects of other jet fans on the flow field of this part, in the simulations, the inlet velocity of 0.76 m/s and outlet gauge pressure of 0.172 Pa were applied on the portals of the modelled tunnel. By comparing their numerical data and achieved experimental results, Cascetta et al. (2016) showed that the reduced scale model can interpret the phenomena that happen in the real scale geometry, and therefore, can be a good technique for the investigation of tunnel ventilation. Accordingly, a comparison between the air velocities achieved by the simulations of this work and the experimental measurements of Cascetta et al. (2016) for the reduced model is illustrated in Fig. 8. Our numerical approach slightly under-estimate the peak velocity value close to the jet fan outlet ($d = 0.3$ m), while over-estimates at a further distance ($d = 2.0$ m), both by about 8%. However, the comparison shows an acceptable agreement between the numerical data and that achieved by the experiments, which can verify the validity of the numerical methods used in the current study.

3 Results and discussions

In this study, the performance of a semi-transverse ventilation system in the extraction of air from an imaginary

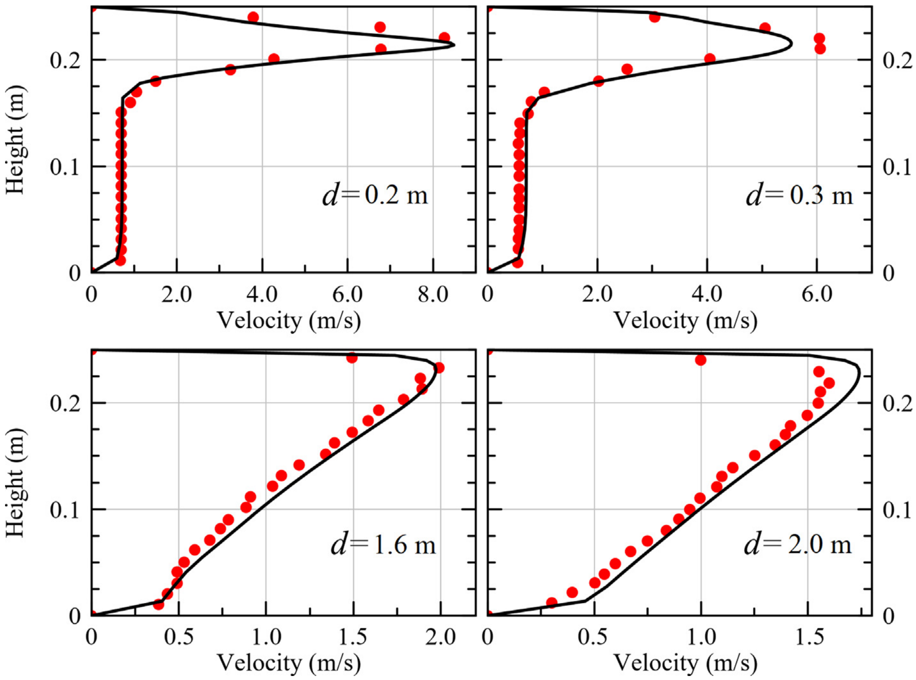


Fig. 8. Comparison between the simulated air velocities of this work (solid lines) and experimental measurements of Cascetta et al. (2016) (circles) on vertical lines at the center of the tunnel. The vertical axis is the distance of the measurement point from the floor, and the axial distance (*d*) from the jet fan outlet is shown for each figure.

incident zone in a 3D long tunnel was investigated. A 1D simulation is significantly time- and cost-effective. However, in a 1D study (or even in the analytical engineering design), several effects should be ignored, or some simplifying assumptions should be made. For example, the arrangements of the traffic, the complicated interaction of high-velocity flow with the large obstacles in the traffic, and the non-uniform flow in the tunnel cross-section or the fan inlet/outlet cannot be observed in a 1D design, which all may have effects on the ventilation system performance. Particularly, the details of the sucked air through the dampers being perpendicular to the stream in the main tunnel is a 3D flow by its nature. In a 1D assumption, the pressure drop due to the traffic is simply achieved by multiplying the number of vehicles by the drag coefficient of a single vehicle, although the shadow effect might be significant in changing the effective drag coefficient of each vehicle. Several other simplifications or ignorance should also be considered in a 1D design regarding the real-world smoke spreading from fire, smoke and air temperature in the tunnel, flow bifurcation reaching a fire, radiation effects, etc., which are out of the scope of the current study (cold-tests). Therefore, we expect to have a better picture of what occurs in a tunnel by performing a 3D simulation.

As mentioned in the introduction, it is recommended that an equal amount of air is sucked from both sides of the extraction zone. That is, the tunnel air velocity on both sides of the zone should be balanced to provide a confined smoke zone, and consequently, a safe region for evacuation and rescue team access. This is performed by employing jet fans in the main tunnel. To simulate this situation in our numerical investigations, being a representative of a real-world cold-test, it is assumed that the contaminated air should be confined within a 300 m region and three closest dampers to the region are opened for the extraction from the main tunnel. Two vertical planes with a distance of 150 m from the centre of the extraction region were used for evaluating the airflow rates reaching the region from both sides. Apart from the simulation of a commissioning test, the preliminary design concept of using exhaust fans at both ends of the tunnel for extracting the contaminated air has been challenged by evaluating the scenarios in which the close exhaust fan to the extraction zone was only employed. The effects of traffic congestion on the performance of the ventilation system were studied in each scenario.

Furthermore, two different extraction zones (relatively close to one portal) were evaluated to determine the system's capability in removing the contaminated air produced in the region. In Extraction zone 1, it was supposed that the centre of the confined region was 450 m from the Tehran portal, while in Extraction zone 2, the centre of the region was 1000 m from the inlet portal. In all simulations, cold-tests are performed.

3.1 Extraction zone 1 (450 m from inlet)

When the air is to be extracted from a region whose centre is at 450 m from the inlet portal, dampers $2^{\#}$, $3^{\#}$, and $4^{\#}$, respectively located at 375, 475, and 575 m from the inlet, are opened, while the rest are closed. Two different conditions inside the tunnel (empty and stationary full traffic) are investigated for this extraction zone. At the same time, two different ventilation scenarios were also employed by turning the farther exhaust portal fan on or off.

3.1.1 Empty tunnel

In this section, the simulations performed in an empty tunnel where the extraction zone centre was at 450 m distance are presented. The airflow rates at different tunnel locations are shown on the schematic flow diagram of the tunnel in Fig. 9(a).

According to the results, the exhaust fans at Tehran and North portals extract 210 and 147 m^3/s, respectively, consistent with the physical configuration of the domain. The pressure loss from the open dampers to the North portal in the ceiling duct is significantly larger than that from the extraction region to the Tehran portal (corresponding to about 4.4 km compared to 0.5 km). This is also the same for the air sucked through the tunnel portals toward the extraction region. Only 36% of the air entering the tunnel is from the North portal (130 m^3/s), and surprisingly, only 32 m^3/s of that reaches the extraction zone from that side. This is because the leakage occurs through the ceiling (85 m^3/s) and closed dampers (22 m^3/s) along the tunnel and drives the air from the high-pressure region (main tunnel) to the low-pressure zone (ceiling duct). Most of the air (41%) is passed through the closest damper to the inlet fan (damper $2^{\#}$), while the airflows from dampers $3^{\#}$ and $4^{\#}$ are 76 and 71 m^3/s, respectively. More importantly, without employing any jet fans in the tunnel, the airflow rates from two sides of the tunnel are significantly unbalanced, and the flow rate at a plane which is 150 m to the left of the zone core is about 6.9 times that at the right.

On the other hand, by turning on eight jet fans on the right side of the zone in the reverse direction (blowing toward the Tehran portal), the airflows reaching the extraction region from both sides become balanced (only a 5% difference), as shown in Fig. 9(b). The number of required jet fans for balancing is found through a trial-and-error procedure. By comparing the two parts of Fig. 9, it can be observed that, due to the thrust applied by jet fans, the flow rate from the North portal significantly increases

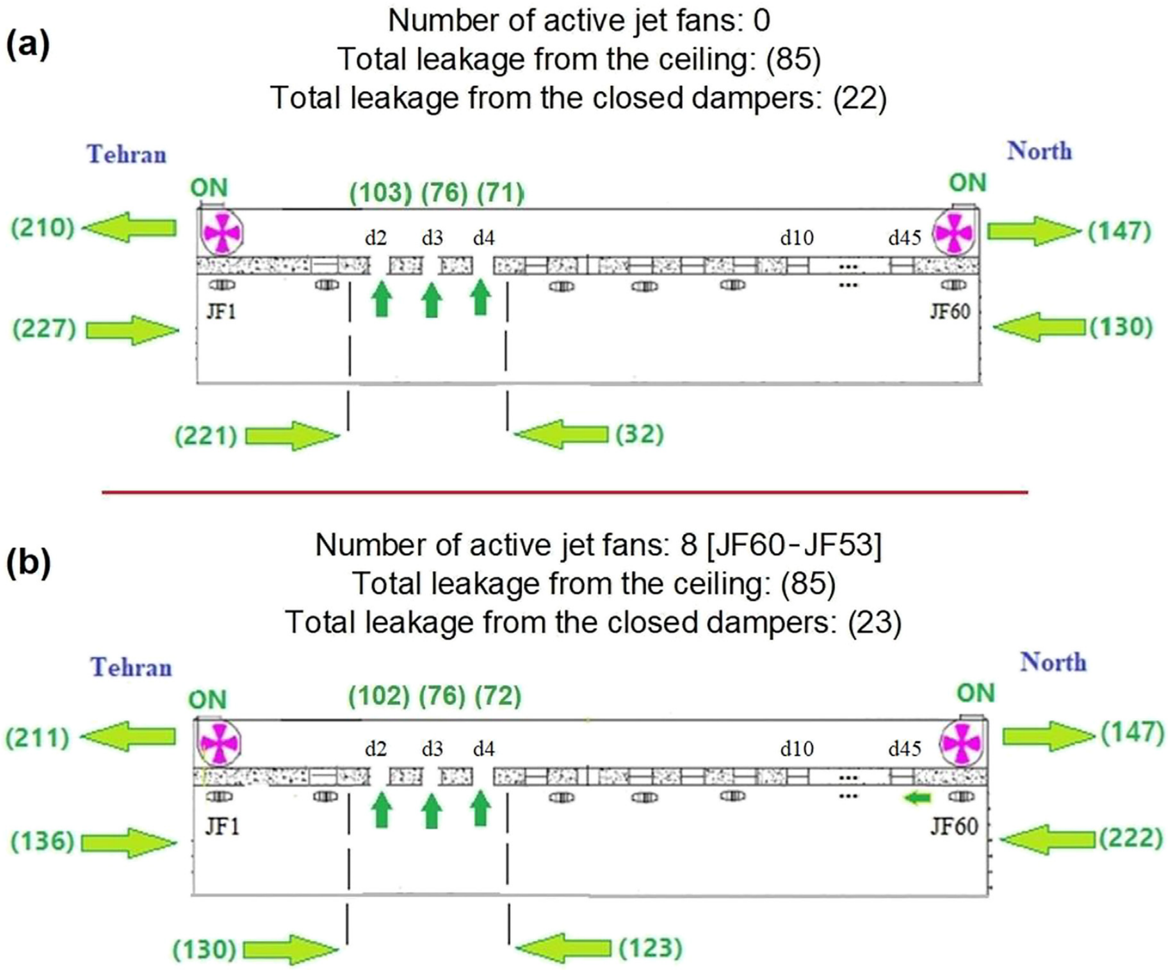


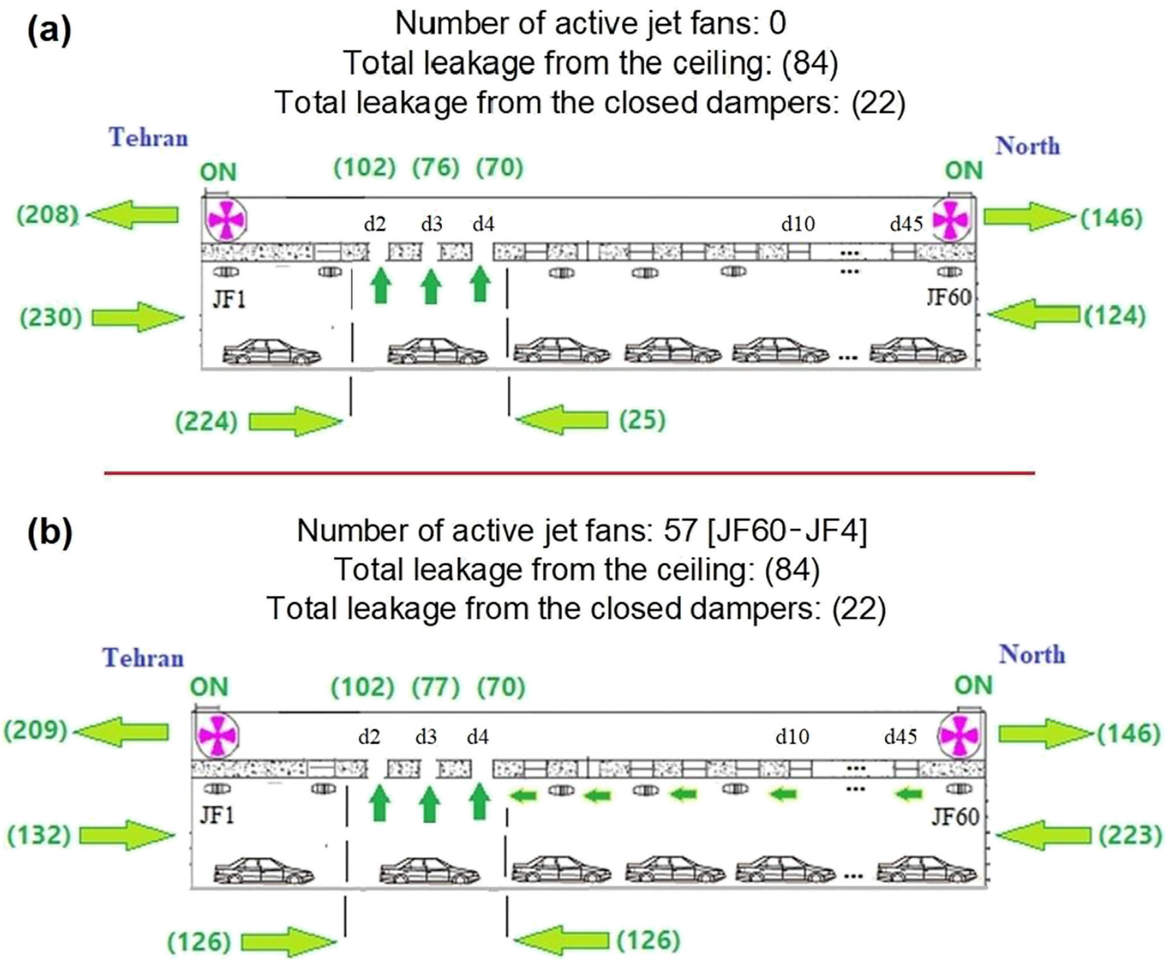
Fig. 9. Airflow rates (numbers in parentheses) in different locations of the empty tunnel in the case of Extraction zone 1 (Unit: m^3/s). Dampers $2^{\#}$, $3^{\#}$, and $4^{\#}$ are open and two exhaust portal fans are employed: (a) in unbalanced condition, and (b) in balanced condition with 8 jet fans. The airflow rates at both sides of the zone are measured on the planes being 150 m apart from the zone centre. Here, "JF" and "d" stand for jet fan and damper, respectively.

and causes the flow rates from each side to be nearly the same. In addition, it can be concluded that the jet fans do not have a significant effect on the airflow behavior in the ceiling duct, including exhaust fan flow rates, air through the dampers, and leakage.

3.1.2 Tunnel with stationary traffic

In this section, as presented in Section 3.1.1, both portal fans are employed as the recommendation of the preliminary 1D design. Dampers $2^{\#}$, $3^{\#}$, and $4^{\#}$ are opened to suck out the air from the extraction zone located at 450 m from the inlet portal. However, here, full stationary traffic exists in the tunnel. A situation which might lead to this unexpected but possible scenario is due to either a collision upstream of a traffic jam or two separate incidents at the beginning and the end of the tunnel. The simulations for two cases without employing jet fans and employing 57 jet fans to balance the flow rates at both sides of the zone are illustrated in Fig. 10. When no jet fan is employed, and the flow is unbalanced, the effect of the traffic on the airflow rates at different parts of the domain is not significant (compare Fig. 9(a) and Fig. 10(a)). This can be because the airflow rates of the portal fans are mainly dependent on the pressure loss in the ceiling duct. In addition, the distance between the extraction zone and the Tehran portal in the main tunnel is relatively short, for which the effect of traffic is not significantly dominant. Therefore, it was found that 57 jet fans should be turned on to balance the flow rates reaching the extraction region (Fig. 10(b)). Surprisingly, this means that some jet fans (JF5 and JF4) at the left side of the region should blow in the opposite direction towards the air stream being sucked from the Tehran portal. Another shortcoming of this scenario is that the jet fans employed in the extraction region may cause the stratification of the smoke, which can be beneficial for providing a safe zone in a real-world situation, to be destroyed (Guo et al., 2021).

The significant number of required jet fans for balancing the airflow shows the effects of heavy traffic on the function of the ventilation system by applying extra pressure loss in the main tunnel due to the interaction of airflow and the obstacles on the road. This effect cannot be captured in a commissioning test and might be incorrectly addressed in the controlling scenarios of emergencies.


(a) Number of active jet fans: 0
Total leakage from the ceiling: (84)
Total leakage from the closed dampers: (22)

(b) Number of active jet fans: 57 [JF60-JF4]
Total leakage from the ceiling: (84)
Total leakage from the closed dampers: (22)

Fig. 10. Airflow rates (numbers in parentheses) in different locations of the tunnel with stationary traffic in the case of Extraction zone 1 (Unit: m^3/s). Dampers $2^{\#}$, $3^{\#}$, and $4^{\#}$ are open and two exhaust portal fans are employed: (a) in unbalanced condition, and (b) in balanced condition with 57 jet fans.

The presented results show that a high fraction of the entered flow from the North portal leaks to the ceiling, due to induced low pressure in the ceiling duct by the axial fan employed at the exit portal. Hence, there is an essential doubt regarding the suggested scenario in the preliminary design phase for turning the farther axial fan on when the extraction zone is near the entrance. Therefore, in the following, the effects of turning the farther exhaust fan off on the ventilation system's performance and the required number of jet fans for flow balancing are investigated.

3.1.3 Ventilation system with only one portal fan

Based on the results achieved in the previous scenarios, a change is applied to the ventilation system, where only the axial fan located near the extraction zone is employed. Figure 11 shows the results of these simulations with and without using jet fans for balancing the airflow when stationary traffic happens in the tunnel. Turning the farther exhaust fan off, and therefore, reducing the negative pressure in a significant part of the ceiling duct causes a reduced amount of air leakage through the ceiling and closed dampers. The total leakage (107 m^3/s) in the scenarios with two exhaust fans (Fig. 10) is more than 3 times that of similar scenarios with only one fan (34 m^3/s). However, the airflow rate extracted through the open dampers from the extraction zone when only the Tehran portal fan is used is about 75% of the scenario in which both portal fans are employed because the extraction capacity is reduced by turning one axial fan off.

As shown in Fig. 11(a), the airflow is not balanced without employing jet fans, as expected. However, only 25 jet fans in the reverse direction (compared to 57 jet fans in a similar situation with two portal fans) can impose a balanced airflow on both sides of the zone (Fig. 11(b)). Two main explanations can justify the lower number of jet fans for balancing. First, by comparing the unbalanced condition of using two portal fans and only one (Fig. 10(a) and Fig. 11(a)), due to the reduced amount of leakage, more considerable amount of air reaches the extraction zone from the right side (43 m^3/s compared to 25 m^3/s). Second, because of the lower amount of air extracted from

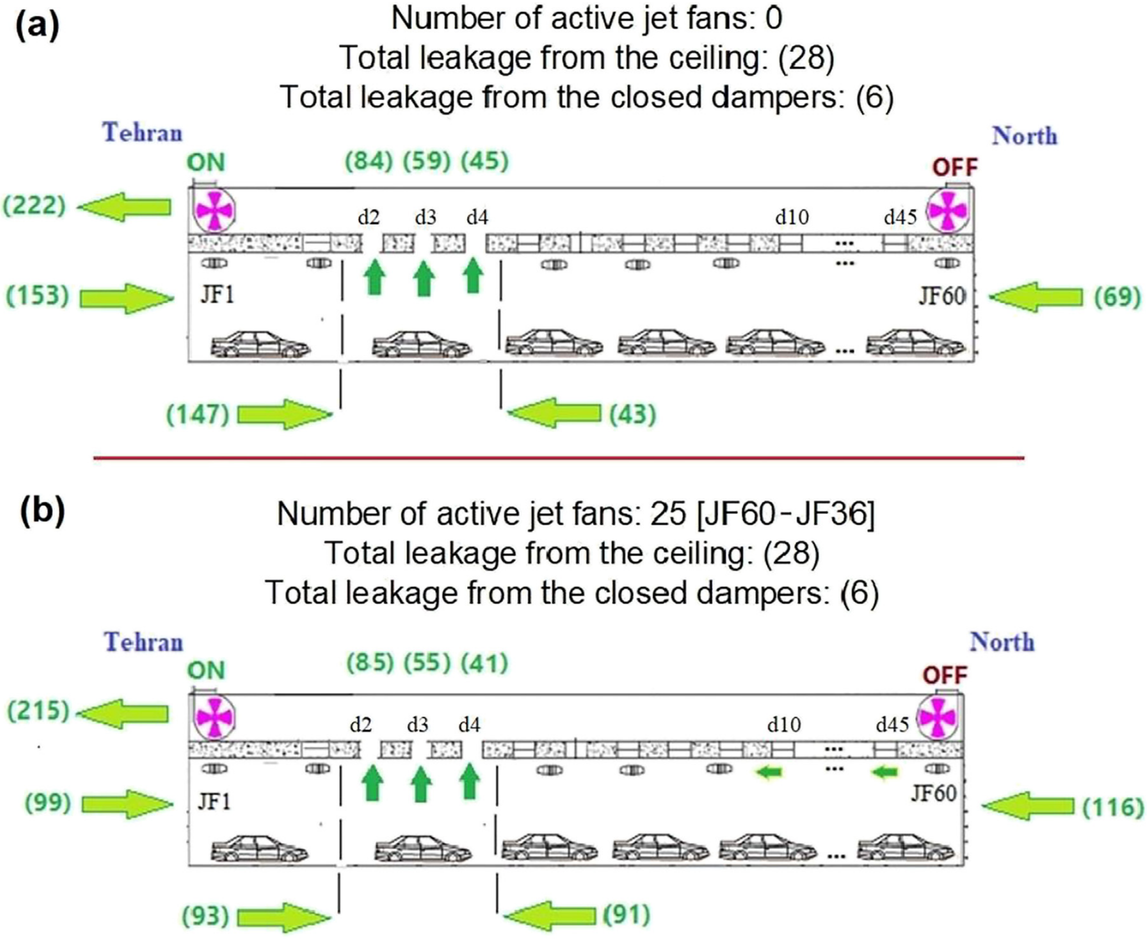


Fig. 11. Airflow rates (numbers in parentheses) in different locations of the tunnel with stationary traffic in the case of Extraction zone 1 (Unit: m^3/s). Dampers 2#, 3#, and 4# are open and only the Tehran portal exhaust fan is employed: (a) in unbalanced condition, and (b) in balanced condition with 25 jet fans.

the zone, making the balance can be done in generally lower flow rates (approximately 91 m^3/s compared to 126 m^3/s shown in Fig. 11(b) and Fig. 10(b), respectively).

By simulation of a short tunnel or even a cut tunnel representing the concerned part of a long one (for example, the distraction zone only), several effects might have been ignored. As seen in the presented results, the leakage from the ceiling has a significant effect on the airflow rate reaching the extraction zone. In addition, as the farther fan sees a significantly larger pressure difference due to the loss in the long duct, it has to provide less airflow in a different working point of its characteristic curve illustrated in Fig. 3. All of these would not be captured if we had simulated a cut short tunnel.

The results of this section suggest that when the extraction zone is close to either side of the tunnel portals, having the closer fan in action might be beneficial in terms of energy saving due to the reduced number of jet fans for airflow balancing and employing only one portal fan instead of two. However, the question of “how close the extraction zone to the portal should be?” remains. In the following section, this issue is investigated by performing the simulations for Extraction zone 2.

3.2 Extraction zone 2 (1000 m from inlet)

This section assumes that the extraction zone is located at a distance of 1000 m from the tunnel entrance with full stationary traffic. Hence, dampers $7^{\#}$, $8^{\#}$, and $9^{\#}$ of the tunnels (the distances to the entrance are 875, 975, and 1075 m, respectively) are opened, and other dampers are closed. The balancing criterion is evaluated on two vertical planes at the distance of 850 and 1150 m from the inlet (150 m away from the zone centre). To investigate the semi-ventilation system with only one portal fan in an extraction zone closer to the centre of the tunnel than in the case of Extraction zone 1, two different scenarios with and without employing the farther exhaust fan are simulated.

3.2.1 Ventilation system with two portal fans

Figure 12 shows the simulation results of the ventilation system in the tunnel with full stationary traffic with Extraction zone 2. By moving the extraction zone toward the centre of the tunnel, the values of the airflow rates of portal fans approach each other, and their differences reduce from about 65 m^3/s for Extraction zone 1 to about 40 m^3/s for the second zone. However, the amount of total leakage

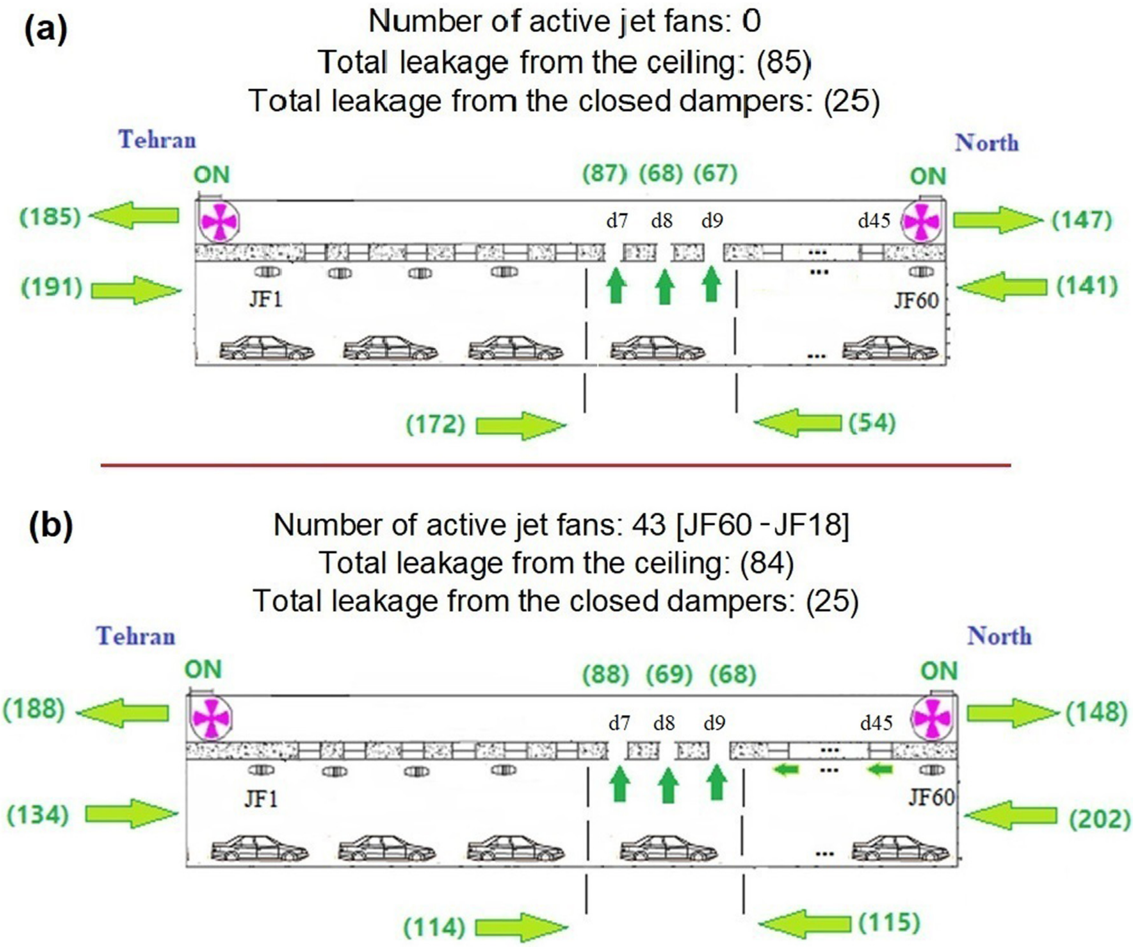


Fig. 12. Airflow rates (numbers in parentheses) in different locations of the tunnel with stationary traffic in the case of Extraction zone 2 (Unit: m^3/s). Dampers $7^{\#}$, $8^{\#,}$ and $9^{\#}$ are open, and two exhaust portal fans are employed: (a) in unbalanced condition, and (b) in balanced condition with 43 jet fans.

seems to be independent of the extraction location and remains at about 110 m^3/s when both portal fans are utilized. For balancing the airflow at the planes adjacent to the zone (Fig. 12(b)), 43 jet fans should be turned on to blow air from North to Tehran. This number is smaller than that required for balancing the similar scenario for Extraction zone 1 (57 jet fans). However, it still seems that this is not an efficient scenario for tunnel ventilation when the extraction zone is about 1000 m from the inlet of the tunnel. Therefore, in the following, the semi-transverse ventilation system with only the close portal fan to the zone is investigated.

3.2.2 Ventilation system without the farther portal fan

Here, the idea of turning the farther portal fan off for ventilation of the tunnel with an extraction zone at a distance of 1000 m from the inlet portal is evaluated. However, the practicality of this idea fades away for the extraction zones very close to the centre of the long tunnel, for which both portal fans should be employed, and the flow balance can be maintained with either a few or no jet fans.

Figure 13 shows the results of simulations for the case in which the extraction zone is 1000 m apart from the inlet portal. By turning off the farther portal fan, the total leakage through the ceiling and dampers significantly reduced (from 110 m^3/s to about 41 m^3/s) as in the case of Extraction zone 1 discussed in Section 3.1. However, the leakage in this extraction location when only one portal fan is used is slightly higher than that of Extraction zone 1 (41 m^3/s compared to 34 m^3/s), because the longer length of the tunnel is exposed to the low pressure applied by the Tehran portal fan in the ceiling duct, which, in turn, induces the leakage. As expected, a significantly smaller number of jet fans (12 jet fans compared to 43 jet fans) is required for balancing the airflow at two sides of the zone. This is because, as said, the leakage is lower, and also the airflow rate at which the balancing occurs (about 75 m^3/s) is lower than that in the case of having two portal fans for the same extraction zone (about 115 m^3/s). It should be noted that even this low flow rate in the current scenario with only one portal fan is adequate for removing fire products from the region in a semi-transverse ventilation system based on the recommendation of PIARC (2011).

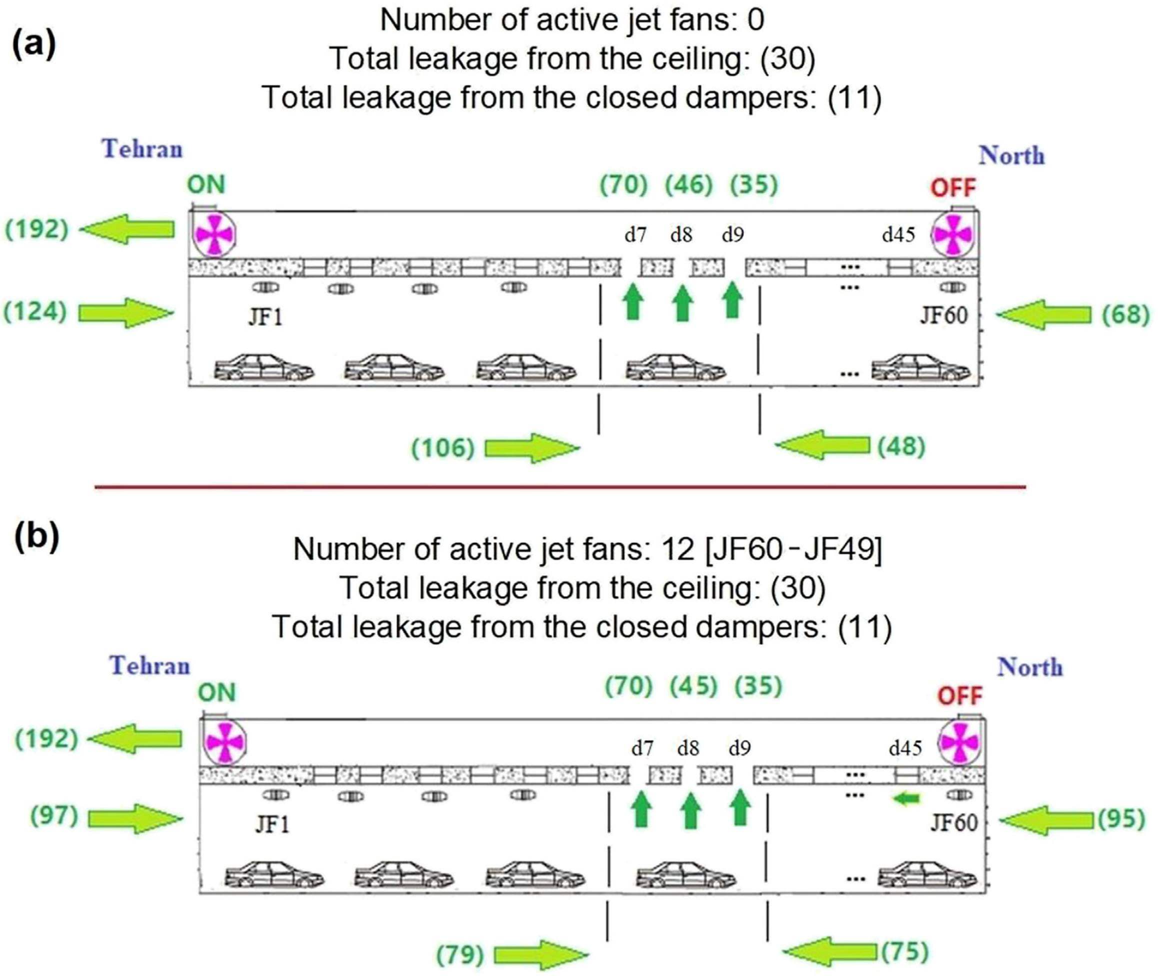


Fig. 13. Airflow rates (numbers in parentheses) in different locations of the tunnel with stationary traffic in case of extraction zone 2 (Unit: m^3/s). Dampers 7#, 8#, and 9# are open. Only the Tehran portal exhaust fan is employed: (a) in unbalanced condition, and (b) in balanced condition with 12 jet fans.

Figures 14 and 15 show the velocity vectors on a vertical plane between the two lines of the vehicles in the vicinity of each open damper for the balanced case of Extraction zone 2, when both portal fans and only the Tehran portal fan are employed, respectively. Comparing these figures with their corresponding flow diagrams shown in Fig. 12(b) and Fig. 13(b), it is observed that the airflow reaching the zone in the main tunnel from the North side is not completely extracted through damper $9^{\#}$ and therefore, at the left side of this damper, the velocity vectors are still toward the Tehran portal. On the other hand, the airflow, coming from the Tehran portal to the extraction region, in the scenario in which only the Tehran portal fan is on, is almost sucked through damper $7^{\#}$ (see Fig. 13(b) and Fig. 15). Therefore, the air velocity between dampers $7^{\#}$ and $8^{\#}$ is negligible. Another interesting finding of Fig. 15 is that in this scenario, due to the air leakage to the ceiling duct on the distance between the extraction zone and the North portal, there exists airflow in the duct from the North side toward damper $9^{\#}$. This can guarantee that the smoke extracted through open dampers is not distributed on the unfavorable side of the ceiling duct even through a diffusion mechanism. However, when both portal fans are employed (Fig. 14), the extracted smoke should travel in both directions all along the tunnel in the ceiling duct, which might impose additional risk to the generally safe zones of the tunnel in the case of any failure in the ventilation system.

The results achieved in this work suggest that, in a semi-transverse ventilation system of long tunnels, a critical distance, L_{cr}, from the portals should be defined to be used in the control philosophy of tunnel ventilation. This means that if the extraction zone is at a shorter distance than L_{cr}, then, only the exhaust fan of the near portal should be activated. Conversely, employing the axial fans in both

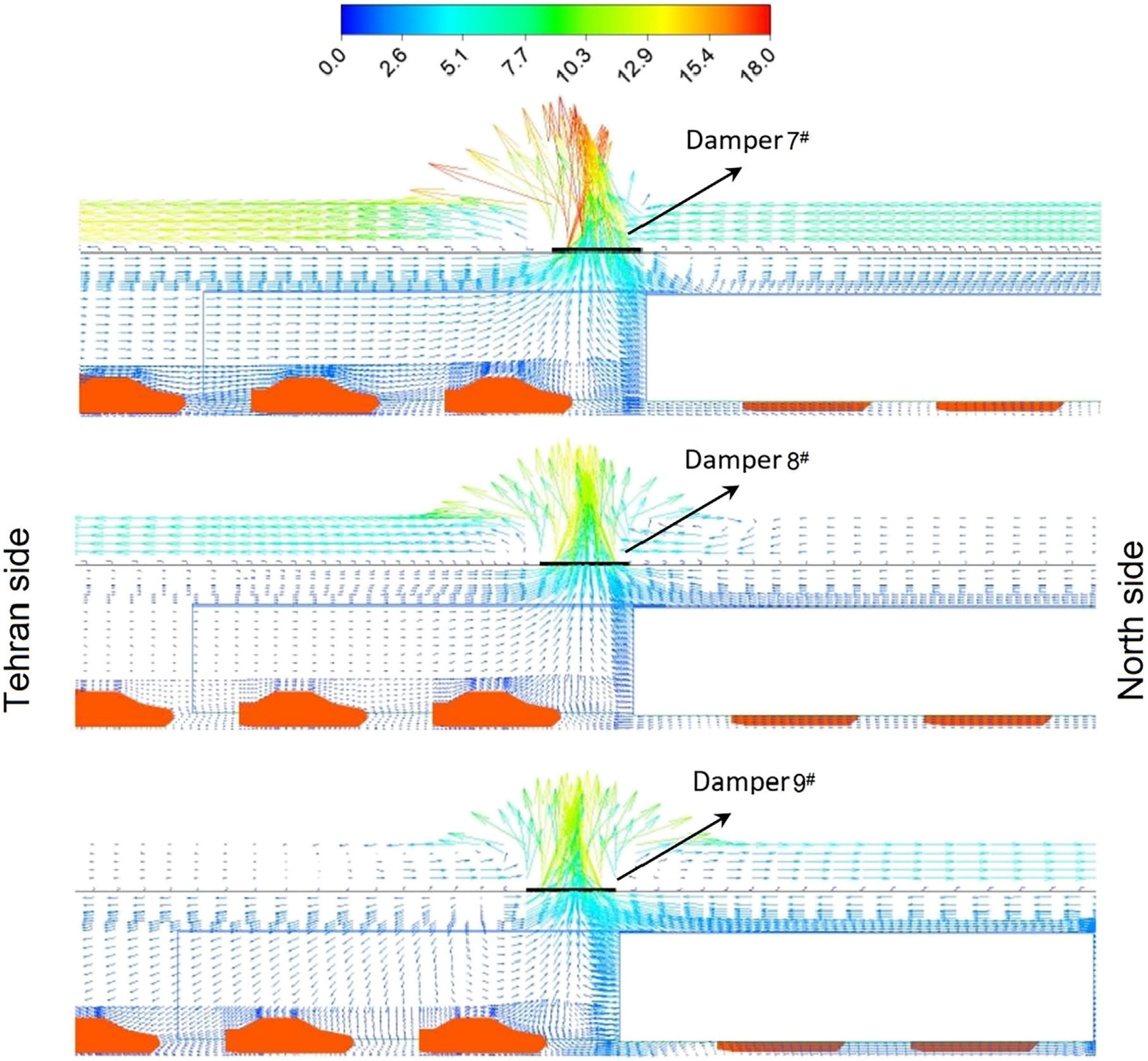


Fig. 14. Velocity vectors on a vertical plane between two lines of vehicles around the opened dampers, in the tunnel with stationary traffic for Extraction zone 2, while two exhaust portal fans are employed (Unit: m/s). Large vehicles are illustrated through their borderlines for clarification. The bottom surface of the drop ceiling is shown through a horizontal line.

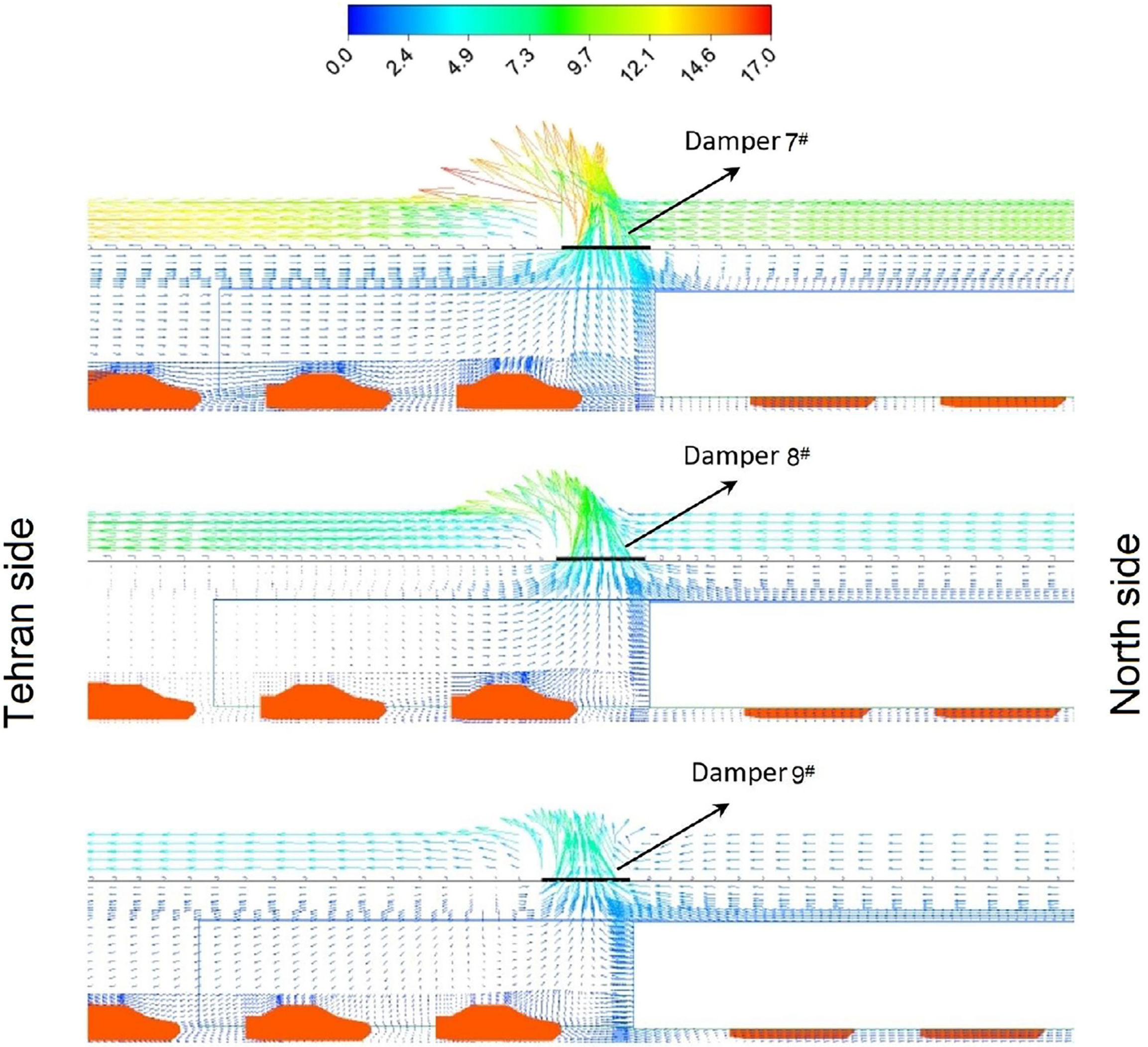


Fig. 15. Velocity vectors on a vertical plane between two lines of vehicles around the opened dampers, in the tunnel with stationary traffic for Extraction zone 2, while only the Tehran portal exhaust fan is employed (Unit: m/s).

portals connected to the ceiling duct will be permitted if the distance of the zone to the portals is higher than L_{cr}. The reason for the existence of such a critical length is that negative pressure in a longer length of the ceiling duct from both sides of the extraction zone results in a significant leakage through the ceiling, and therefore, less amount of air can naturally reach the extraction zone due to the portal fan suction. Therefore, employing only one portal fan leads to less leakage, and consequently fewer jet fans are required for the compensation. This critical length depends most probably on the operational curve of the exhaust fans, the cross-section areas of the tunnel and extraction duct, the heat release rate of the fire, and traffic congestion conditions. The simulations of the semi-transverse ventilation system of the investigated tunnel with the specified operational curve illustrated in Fig. 3 for the portal fans and stationary complete traffic congestion in cold-test conditions show that L_{cr} is at least 1 km. The extraction zones located at a distance smaller than this critical length can be balanced with a reasonable number of jet fans if only the exhaust fan of the close portal is utilized.

4 Conclusion

In this study, the semi-transverse ventilation system consisting of 45 dampers, 60 jet fans, and two portal exhaust fans connected to a ceiling duct in a 4900 m tunnel was numerically investigated. The goal was to simulate some commissioning tests and address the effects that might be either achievable or unachievable in such real-world tests. Considering two hypothetical locations for the extraction zone (450 and 1000 m apart from the tunnel inlet), three dampers near the zone are opened, and the axial fans connected to the ceiling duct are activated to extract air from the main tunnel. It is essential to have a balanced airflow at both sides of the extraction zone to provide a confined region and facilitate evacuation. Various scenarios with different traffic conditions (empty tunnel and tunnel with complete stationary traffic) were evaluated. In the commissioning tests of the ventilation system, an empty tunnel is evaluated. However, the control scenario should be also functional in complete traffic congestion, and the simulations presented here address the differences between possible real-world emergencies and the commissioning tests. Furthermore, some general ideas regarding the controlling scenario, including the effect of getting one or two portal fans activated were investigated.

For balancing the airflow at both sides of an extraction zone close to one of the tunnel portals, several jet fans should be activated in the reverse direction to blow more air from the longer side of the tunnel to the zone. The results showed that the presence of stationary traffic in the tunnel, and therefore their interaction with the airflow, has a significant effect on the number of required jet fans for airflow balancing (about 3–7 times depending on the farther fan working condition). The results depicted that the ventilation control scenarios in emergencies should be carefully designed, in terms of employing the main portal exhaust fans connected to the ceiling duct. Here, it was shown that the distance between the extraction zone and the closest exhaust fan is crucial. If the distance is less than a critical length (which was verified to be at least 1 km for this work), only the closest exhaust fan should be activated, whereas, where the distance is more than this critical length, activation of both exhaust fans would be necessary. In the former case, turning the farther exhaust fan leads to two major problems. First, the required number of jet fans for making a balanced airflow increases dramatically (for example from 25 to 57 jet fans for the Extraction zone 1). Second, a higher fraction (about 3 times) of the air moving in the main tunnel is leaked to the ceiling duct without reaching the extraction zone.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgement

Tehran-North Freeway Company financially supported this research except for S. S., and this support is acknowledged.

References

Almbauer, R. A., Sturm, P. J., Bacher, M., & Pretterhofer, G. (2004). Simulation of ventilation and smoke movement. In *Proceedings of the 2nd International Conference on Tunnel Safety and Ventilation* (pp. 32–38). 19–21 April, 2004, Graz, Austria.

Ansys® Fluent (2015)*, Release 16.1, ANSYS Fluent User's Guide,* Ansys, Inc., Canonsburg, PA, USA.

Ballesteros-Tajadura, R., Santolaria-Morros, C., & Blanco-Marigorta, E. (2006). Influence of the slope in the ventilation semi-transversal system of an urban tunnel. *Tunnelling and Underground Space Technology, 21*(1), 21–28.

Betta, V., Cascetta, F., Musto, M., & Rotondo, G. (2009). Numerical study of the optimization of the pitch angle of an alternative jet fan in a longitudinal tunnel ventilation system. *Tunnelling and Underground Space Technology, 24*(2), 164–172.

Cascetta, F., Musto, M., & Rotondo, G. (2016). Innovative experimental reduced scale model of road tunnel equipped with realistic longitudinal ventilation system. *Tunnelling and Underground Space Technology, 52*, 85–98.

Cascetta, F., Musto, M., Rotondo, G., D'Alessandro, C., & De Maio, D. (2019). Novel correlation to evaluate the pressure losses for different traffic jam conditions in road tunnel. *Tunnelling and Underground Space Technology, 86*, 165–173.

Chow, W. K., Wong, K. Y., & Chung, W. Y. (2010). Longitudinal ventilation for smoke control in a tilted tunnel by scale modeling. *Tunnelling and Underground Space Technology, 25*(2), 122–128.

Colella, F., Rein, G., Carvel, R., Reszka, P., & Torero, J. L. (2010). Analysis of the ventilation systems in the Dartford tunnels using a multi-scale modelling approach. *Tunnelling and Underground Space Technology, 25*(4), 423–432.

Dastan, A., Rahiminejad, M., Abbasi, M., & Abouali, O. (2019). CFD simulations of longitudinal ventilation of a road tunnel in congested traffic condition. In *Tunnels and Underground Cities: Engineering and Innovation meet Archaeology, Architecture and Art* (pp. 4759–4768). CRC Press/Balkema.

Du, T., Yang, D., Peng, S. N., Liu, Y. L., & Xiao, Y. M. (2016). Performance evaluation of longitudinal and transverse ventilation for thermal and smoke control in a looped urban traffic link tunnel. *Applied Thermal Engineering, 96*, 490–500.

Du, T., Yang, D., Peng, S. N., & Xiao, Y. M. (2015). A method for design of smoke control of urban traffic link tunnel (UTLT) using longitudinal ventilation. *Tunnelling and Underground Space Technology, 48*, 35–42.

Eftekharian, E., Abouali, O., & Ahmadi, G. (2015). An improved correlation for pressure drop in a tunnel under traffic jam using CFD. *Journal of Wind Engineering and Industrial Aerodynamics, 143*, 34–41.

Eftekharian, E., Dastan, A., Abouali, O., Meigolinedjad, J., & Ahmadi, G. (2014). A numerical investigation into the performance of two types of jet fans in ventilation of an urban tunnel under traffic jam condition. *Tunnelling and Underground Space Technology, 44*, 56–67.

Fan, C. G., Zhang, L., Jiao, S. C., Yang, Z. W., Li, M. H., & Liu, X. P. (2018). Smoke spread characteristics inside a tunnel with natural ventilation under a strong environmental wind. *Tunnelling and Underground Space Technology, 82*, 99–110.

Fang, Y., Yao, Z. G., & Lei, S. (2019). Air flow and gas dispersion in the forced ventilation of a road tunnel during construction. *Underground Space, 4*(2), 168–179.

Gu, X. C., Zhang, J. D., Pan, Y., Ni, Y. Q., Ma, C. M., Zhou, W., & Wang, Y. Y. (2020). Hazard analysis on tunnel hydrogen jet fire based on CFD simulation of temperature field and concentration field. *Safety Science, 122*, 104532.

Guo, Y. H., Yuan, Z. Y., Yuan, Y. P., Cao, X. L., & Zhao, P. (2021). Numerical simulation of smoke stratification in tunnel fires under longitudinal velocities. *Underground Space, 6*(2), 163–172.

Hu, L. H., Tang, W., Chen, L. F., & Yi, L. (2013). A non-dimensional global correlation of maximum gas temperature beneath ceiling with different blockage–fire distance in a longitudinal ventilated tunnel. *Applied Thermal Engineering, 56*(1–2), 77–82.

Kaluva, S. T., Pathak, A., & Ongel, A. (2020). Aerodynamic Drag Analysis of Autonomous Electric Vehicle Platoons. *Energies, 13*(15), 4028.

Król, A., & Król, M. (2018). Transient analyses and energy balance of air flow in road tunnels. *Energies, 11*(7), 1759.

Lee, S. R., & Ryou, H. S. (2006). A numerical study on smoke movement in longitudinal ventilation tunnel fires for different aspect ratio. *Building and Environment, 41*(6), 719–725.

Levy, S. S., Sandzimier, J. R., Harvey, N. A., & Rosenbluth, E. M. (1999). CFD model for transverse ventilation systems. In *Proceedings of the First International Conference on Tunnel Fires and One Day Seminar on Escape from Tunnels* (pp.223–233). 5–7 May 1999, Lyons, France.

Lin, P., Lo, S. M., & Li, T. (2014). Numerical study on the impact of gradient on semi-transverse smoke control system in tunnel. *Tunnelling and Underground Space Technology, 44*, 68–79.

Lin, P., Zhang, Y. C., Li, T., & Si, Y. L. (2016). A numerical study on the impact of vehicles' blockage on the performance of semi-transversal smoke control system in tunnel fire. *Procedia Engineering, 135*, 248–260.

Liu, C., Zhong, M. H., Tian, X. L., Zhang, P. H., & Li, S. W. (2019). Study on emergency ventilation for train fire environment in metro interchange tunnel. *Building and Environment, 147*, 267–283.

López González, M., Galdo Vega, M., Fernández Oro, J. M., & Blanco Marigorta, E. (2014). Numerical modeling of the piston effect in longitudinal ventilation systems for subway tunnels. *Tunnelling and Underground Space Technology, 40*, 22–37.

Musto, M., Cascetta, F., D'Alessandro, C., De Maio, D., & Rotondo, G. (2020). Novel correlation to evaluate the pressure losses for different traffic jam conditions in road tunnel with alternative jet fan, second part. *Tunnelling and Underground Space Technology, 96,* 103213.

Myrvang, T., & Khawaja, H. (2018). Validation of air ventilation in tunnels, using experiments and computational fluid dynamics. *The International Journal of Multiphysics, 12*(3), 295–312.

National Fire Protection Association (2020a). *NFPA 502: Standard for Road Tunnels, Bridges, and Other Limited Access Highways*. MA, USA: Quincy.

National Fire Protection Association (2020b). *NFPA 130: Standard for Fixed Guideway Transit and Passenger Rail Systems*. MA, USA: Quincy.

World Road Association (2011). *Road tunnels: Operational strategies for emergency ventilation*. Report No. 2011R02. La Defense Cedex, France.

Rafiei, M. (2015). Numerical simulation of a full scaled fire test of the tunnel with natural ventilation. *International Journal of Computer Applications, 115*(1), 9–16.

Savalanpour, H., Farhanieh, B., & Afshin, H. (2021). Proposing a general formula to calculate the critical velocities in tunnels with different cross-sectional shapes. *Tunnelling and Underground Space Technology, 110,* 103798.

Shibani Rajkumar, J., Aryai, V., Salehi, F., Abbassi, R., Ghodrat, M., & Dadashzadeh, M. (2022a). Computational fluid dynamics modelling of a hydrogen fire safety in a scaled tunnel environment. *Safety in Extreme Environments, 2021*, 1–15.

Shibani Salehi, F., Baalisampang, T., & Abbassi, R. (2022b). Numerical modeling towards the safety assessment of multiple hydrogen fires in confined areas. *Process Safety and Environmental Protection, 160*, 594–609.

Stroppa, W. (2004). Practical experience in ventilation design for longitudinal and semi transverse ventilation systems. In *Proceedings of the 2nd International Conference on Tunnel Safety and Ventilation* (pp. 227–234). 19–21 April 2004, Graz, Austria.

Sturm, P. J., Beyer, M., Bacher, M., & Schmölzer, G. (2012). The influence of pressure gradients on ventilation design–special focus on upgrading long tunnels. In *Proceedings of the 6th International Conference on Tunnel Safety and Ventilation* (pp. 90–99). 23–25 April 2012, Graz, Austria.

Sturm, P., Beyer, M., & Rafiei, M. (2017). On the problem of ventilation control in case of a tunnel fire event. *Case Studies in Fire Safety, 7*, 36–43.

Tao, Y., Dong, J. L., Xiao, Y. M., & Tu, J. Y. (2016). Numerical analysis of pollutants dispersion in urban roadway tunnels. In *Proceedings of the 20th Australasian Fluid Mechanics Conference*. 5–8 December 2016, Perth, Australia.

Vega, M. G., Argüelles Díaz, K. M., Fernández Oro, J. M., Tajadura, R. B., & Santolaria Morros, C. (2008). Numerical 3D simulation of a longitudinal ventilation system: Memorial Tunnel case. *Tunnelling and Underground Space Technology, 23*(5), 539–551.

Wang, K. H., Chen, J. M., Wang, Z. K., Gao, D. L., Wang, G. Y., & Lin, P. (2019). An experimental study on self-extinction of tunnel fire under natural ventilation condition. *Tunnelling and Underground Space Technology, 84*, 177–188.

Yuan, J. P., Fang, Z., Tang, Z., Li, Q., & Wang, J. H. (2016). Influence of Smoke Damper Configuration on Point Extraction Ventilation System of an Urban Tunnel. *Procedia Engineering, 135*, 99–106.

Zhang, Q., Guo, X., Paladini, A., & Eandi, S. (2014). A Model of Semi-Transversal Tunnel Fire Ventilation. In *Proceedings of the 6th International Symposium on Tunnel Safety and Security (ISTSS)*. 12–14 March, 2014, Marseille, France.

Zhong, M. H., Shi, C. L., He, L., Shi, J. H., Liu, C., & Tian, X. L. (2016). Smoke development in full-scale sloped long and large curved tunnel fires under natural ventilation. *Applied Thermal Engineering, 108*, 857–865.

Zhong, W., Duanmu, W. K., Wang, T., & Liang, T. S. (2017). A study of the critical velocity of smoke bifurcation flow in tunnel with longitudinal ventilation. *Fire Technology, 53*(2), 873–891.

Zhou, Y., Yang, Y., Mao, Z. L., Bu, R. W., Gong, J. H., Wang, Y. X., & Yi, L. (2019). Analytical and numerical study on natural ventilation performance in single-and gable-slope city tunnels. *Sustainable Cities and Society, 45*, 258–270.