CFD MODELING OF A ROAD TUNNEL WITH MULTIPLE SOURCES OF CO. CASE OF STUDY: BOQUERÓN-I TUNNEL

Rafael J. Urbina, Simón Bolívar University, Venezuela
Luis Rojas-Solórzano, Simón Bolívar University, Venezuela
Armando J. Blanco, Simón Bolívar University, Venezuela

ABSTRACT

This work aims to the modeling of CO transport in one section considered critical of the Boqueron I tunnel, located on the outskirts of Caracas, the capital city of Venezuela, with a scenario where vehicles are stopped by an interruption of traffic. This scenario considers a relationship between the number of large-sized vehicles (buses or trucks) and small-sized vehicles (passenger cars) reported by transit statistics and also, it considers the semi-transverse ventilation system in the tunnel. It is explored the influence of the ventilation on the flow patterns and its relationship to the regions with the highest CO concentration. The finite-volume based finite element method is used for the discretization of the computational domain and the integration of the governing equations. The transient 3D-incompressible Navier-Stokes, energy, mass and species conservation equations, along with the k-e turbulence equations, were discretized, using higher-order numerical schemes in space. The numerical simulation is performed using a fully implicit coupled treatment of the set of resulting discrete transport equations.

1. INTRODUCTION

The problem of traffic on the main roads has been a topic of discussion and concern by authorities and society at large. In recent years, the automotive fleet has been steadily growing due to the development of the country. This situation increases the vehicle transit density in urban and suburban roadways, as well as its related problems. The safety of the people who travel through the road tunnels is a matter of concern, since the renewal of fresh air constitutes a critical aspect because of the emissions produced by engine combustion. These emissions have pollutant agents (Whesterholm and Egeback, 1992) and special attention has been placed on carbon monoxide (CO) since it constitutes a choking agent. In relation to time of exposure to atmospheres polluted with CO, there have been cases reported where exposures during periods of time of up to three hours in concentrations of about 1,500 ppm could cause death (Osawa et al., 2003). Other authors maintain as reference that an exposure of around 30 minutes to an atmosphere of 3,000 ppm could be fatal (Vega et al., 2007).
The Caracas-La Guaira highway is one of the most important and travelled roads in Venezuela. On this highway, there are two tunnels identified as Boquerón-I and Boquerón-II, with lengths of 1,825 m and 490 m respectively. Due to the length of the Boquerón I tunnel and the great amount of heavy traffic that travels on the highway, it is considered a high risk tunnel. The Boquerón-I tunnel consists of two unidirectional circulation galleries joined by four transversal galleries as shown in Figure1.

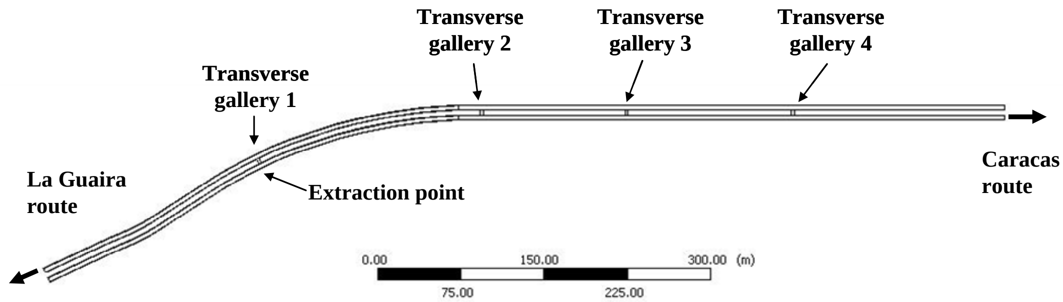
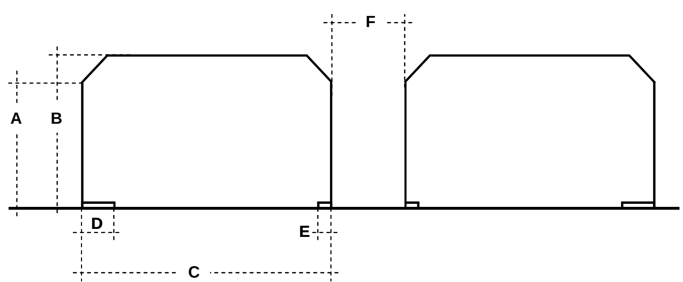


Figure 1. Boquerón-I tunnel.

The ventilation system of the tunnel is semi-transversal and consists of an upper gallery extended along the tunnel. The injection of fresh air to the upper gallery generates a pressurized environment creating the supply of fresh air to the circulation galleries through ventilations openings placed in an equidistant manner. The tunnel has a point of air extraction located in the circulation gallery on the Caracas-La Guaira direction at a distance of 350 m from the entrance on the La Guaira side. Figure 2 shows the transversal layout of the tunnel with the respective measurements.



	A	B	C	D	E	F
m	3	4,33	8,78	7,30	1	7

Figure. 2. Schematic representation of the cross section, Boquerón-I tunnel.
(without the ventilation gallery)

The Boquerón I tunnel was opened in 1953, and the available studies, performed by Rivas and Falcón (1987) and Lezama and Falcón (1987), point out that the area of highest concentration of CO is located between transversal galleries 3 and 4 in the circulation gallery in the La Guaira-Caracas direction. A unidimensional mathematical model was developed in these studies to estimate the distribution of the average transversal concentration of CO along the tunnel, which was adjusted by the minimal squares method to the data reported by the transit authority.
Nowadays, the modeling of the fluid flow has reached an important development, not only in what regards to the development of each time more efficient algorithms but also in the availability of programs capable to model, with accuracy, complex systems being a Computational Fluid Dynamics (CFD), a versatile and effective tool. However, most of the developments and numerical models performed through CFD in highway tunnels have been

concentrated in the modeling of fire scenarios, where the source of heat and pollutants is focused and the presence of vehicles inside the tunnel is not considered (Yuan and You, 2007).
This work focuses on the modelling of CO transportation using CFD as a tool, in a section of the Boquerón-I tunnel considered critical due to the levels of concentration of CO, as aforementioned, between transversal galleries 3 and 4. In the modeled scenario, the presence of vehicles inside the tunnel, which are stopped by a traffic interruption and with the engines running is considered. In this case, CFD is shown as an alternative to tests at real scale due to the high cost of these and to the possibility of considering different scenarios. In this work, it is established as premises the use of commonly accessible and moderate computational resources, as well as performing the modelling with the little available information, in such a way that its application can easily be extended to the highest amount of possible practical cases.

2. METHODOLOGY

The work strategy consisted of building a base numerical model, and on this model, an analysis of mesh sensitivity, and impact evaluation of the geometrical representation of the small-sized vehicles was performed. Afterwards, the modelling of the section of the tunnel was performed.
Calculations were performed with a commercial software package CFX. This code uses the finite-volume based finite element method for the discretization of the computational domain and the integration of the governing equations. Turbulence was simulated with the standard k-e model. The 3D-Navier-Stokes, mass and species conservation equations, along with the k-e turbulence equations, were discretized, using higher-order numerical schemes both in time and space. The buoyancy force due to temperature variations was taken into account through the Boussinesq approximation. The numerical simulation is performed using a fully implicit coupled treatment of the set of resulting discrete transport equations. The code was run in a 2.00 GHz processor with 1.99 GB of RAM. Based on the assumption that the vehicles are parked and the engines running long enough to reach stationary state, the model was solved on the basis of stationary state.
To establish the convergence criterion, the rate of change in the average concentration of CO as function of normalised residual was monitored. The calculation of the average concentration of CO was carried out on four horizontal planes and four transverse planes placed randomly. The concentration of CO was selected, because this parameter was considered the most sensitive variable to the solution. In this way, it was obtained that for residual values less than 0.00001 there is not change in the concentration of CO.

For the base model, the procedure started with the definition of the geometrical characteristics of the numerical domain. Two sizes of vehicles were considered: small-sized vehicles (passengers) and large-sized vehicles (mini-buses, buses and trucks). The proportion between both types of vehicles was considered based on transit statistics reported by authorities. The measurements of the small-sized vehicles were of: 1.5 m high, 1.8 m wide and 4.5 m long, while to large-sized vehicles the following dimensions were assigned: 3.3 m high, 2.5 m wide and 10.5 m long. The separation between the vehicles was of 1 m and the large-sized vehicles were placed on the right lane of the road. The total longitude of the model was of 48.5 m, enough to achieve the main flows development. Eight (8) pairs of ventilation openings were represented measuring 1 m x 0.5 m each, placed at a longitudinal distance of 5 m and with a separation of 2.2 m between them, following the structure of the tunnel. Figure 3 shows the base model geometry.

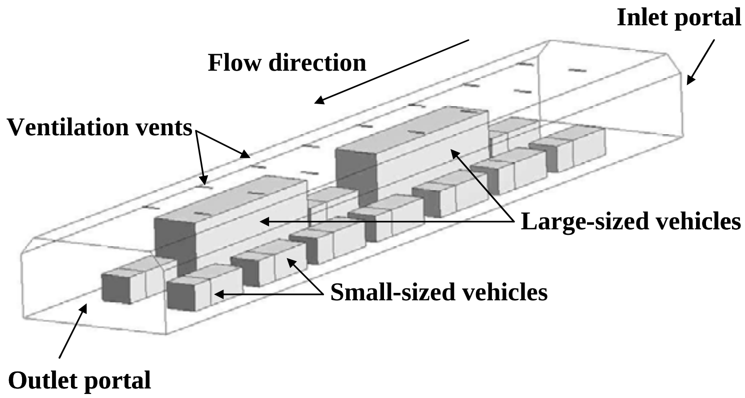


Figure 3. Base model geometry.

The air flow that enters through each one of the ventilation openings was estimated from the work carried out by Campo et al. (1999), who proposed flows for the Boquerón-I tunnel ventilation system in different schedules. The air flow was considered of 0.55 kg/s to 298 K, with an intensity of turbulence of 5% (Casey and Wintergerte, 2000).
The vehicle emissions were considered as a mixture of air and CO with a concentration of CO of 1.5 % molar (Whesterholm and Egeback, 1994), with flows of 0.105 kg/s for small-sized vehicles and 0.315 kg/s for large-sized vehicles. The exhausts were represented as a square –shaped surface (7cm x 7 cm) in order to facilitate the mesh setting. These surfaces were placed at random between two alternatives for each vehicle, either in the bottom left hand, or in the bottom right hand side. The trunks of the vehicles were considered as a flat wall with heat flow. The heat flow for small-sized vehicles was of 0.098 W/m2 and for large-sizedvehicles was of 0.295 W/ m2.
At the inlet portal of the tunnel, the entrance of air at an average speed of 3.1 m/s was considered. This value relates to the speed at the beginning of the section of the tunnel to be modelled, according to measurements taken by the transit authorities and reported by Lezama and Falcón (1987). In relation to the concentration of CO, it was considered to be air free of CO. This does not correspond to the concentration at the entrance of the section. Nevertheless, this base model is performed with comparative purposes, and sensitivity analyses of the studied parameters were carried out upon this basis. Finally, at the outlet portal, a condition of “free passing” with a pressure of 0 Pa was imposed.
The geometry discretization was performed through a hexaedric mesh. In order to improve the transfer of momentum, refinements were placed in the exhaust areas of each one of the vehicles as well as in the ventilation openings. Even though it is a procedure that is not performed in the modelling of road tunnels, an analysis of mesh sensitivity was made. For this analysis, eight mesh were configured with a number of cells that go from 35,506 to 278,132cells. From this analysis, a mesh with 157,053 cells with average size of: 0.30 m in the x direction, 0.30 in the y direction and 0.30 m in the z direction, was selected.
The established monitoring variables were the CO concentration profiles, temperature, and air longitudinal speed along the tunnel and over three imaginary lines located in the areas where it is estimated some users might walk at the moment of leaving their vehicles in the proposed scenario, as it is shown in Figure 4. The selection of these variables, from a practical point of view, is due to that the interaction of these with the people could be a motive of risk. The three

lines were placed at a height of 1.6 m from ground level, considering this value representative of the height of an average user.

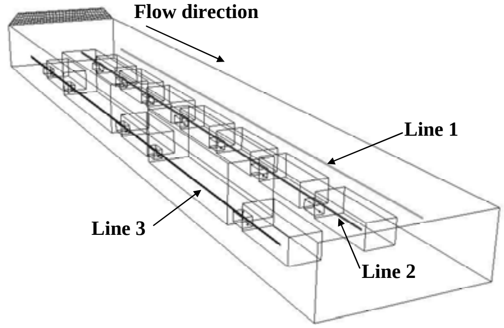


Figure 4. Detail of the scenario and the monitoring lines.

The next step, once established the size of the cells, was to evaluate the impact of the geometrical representation of small-sized vehicles. It was only considered to evaluate the shape of small-sized vehicles since in their representation a bigger simplification was performed. Besides, the shape of large-sized vehicles is closer to the characteristic shape of buses that commonly circulate in the tunnel. For this purpose, a model with a geometrical configuration which was reduced to 50% in the frontal and rear area of the vehicles was built, as shown in Figure 5. When comparing the CO concentration, temperature and speed profiles, differences of up to 34% in the regions of higher values were obtained, which indicates that the geometry of small-sized vehicles impacts the results of the model.

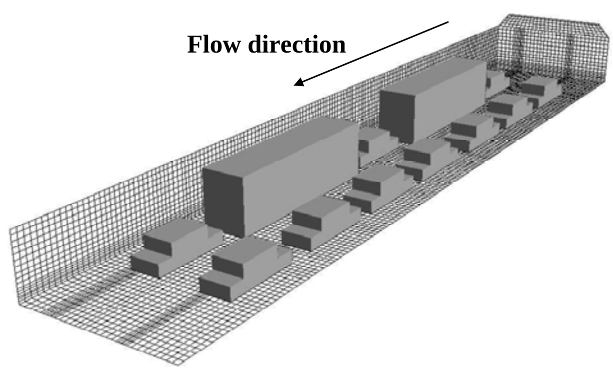


Figure 5. Detail of the geometrical configuration.

Once the impact of these parameters was established, the next step was the modeling of the section of the tunnel. With the purpose of saving efforts, both computational and in the construction of the mesh, the sequential form of modeling was proposed, i.e. a model of the section of the tunnel is built, and the modeling is repeated several times, configuring each run in a way that the exit border condition of the first run (air flow and CO at exit temperature) becomes the border condition at the entrance of the second run, and so on until the desired length which was estimated in 360 m. is covered. This framework assumes an insignificant fall of pressure along the section. On the other hand, it implies some limitations for the global visualization since an integrated mesh along the section is not used. However, these disadvantages are compensated by the fact of being able to simulate sections of the tunnel (or complete tunnels) with relatively economical computational resources, within reasonable time spans from a practical point of view. The small-sized vehicles were represented according to the modification previously studied, considering that this is the most adjustable. The total length of the model was of 61.5 m. This longitude was chosen so that the mesh size allowed a model that could, within the aforementioned economy premises, simulate with a processor of 2.00 GHz and 1.99 GB of RAM. With this longitude, a number of 3 heavy vehicles and 14 light vehicles was configured. It was determined that with this length, the model should be simulated 6 times in the sequence previously described. This way, a total tunnel longitude of 369 m was simulated. Figure 6 shows the configured geometry.

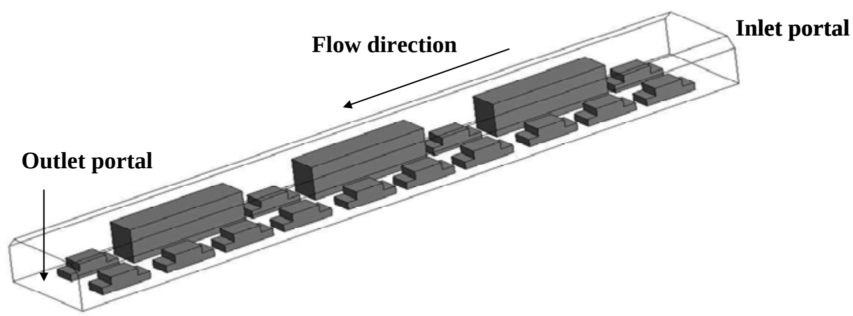


Figure 6. Detail of the geometrical model used in the simulation.

The entrance CO concentration for the first run or simulation was obtained from field measurements reported in the work of Lezama and Falcón (1987) which at the beginning of the section to be modeled, has a value of 365 ppm. It is worth mentioning at this point, that when imposing these values of air entrance speed, CO concentration, as well as temperature in the first run, it is being supposed that these have a uniform distribution. This supposition is valid considering that at the beginning of the tunnel section to model, the effects of mixing must be predominant since they are found in transversal gallery 3.

3. RESULTS

Figure 7 shows the longitudinal speed in the monitoring lines, in relation to the tunnel longitude for the modelled stretch. It can broadly be seen how there is a progressive increase of the longitudinal speed along the tunnel. However, this increase does not show a linear trend. This result is expected considering the effect of the increase of the air mass that circulates in the tunnel, due to the injection of fresh air along it. In the upper part of the Figure 7, the shapes of the vehicles distributed at scale along the tunnel are shown. Correlating the shapes of the vehicles and the profiles of the three lines, it can be observed how in the location that corresponds to the union between the runs of the model there is a sudden decrease of speed that fails to be over 2 m/s in all the cases. This behavior is due to the increase of the transversal area for the flow because of lack of vehicles in these locations. It is also observed how there is similarity in the profiles of each one of the simulations, showing the cyclical nature of the problem. It can be seen how the profiles for line 1 and 3 increase throughout the tunnel, being the increase of largest magnitude the one of line 1, which has a maximum increase rate of 1.6 m/s per each 100 m of tunnel. It can be noticed how from 260 m of longitude, the line 1 profile surpasses the 10 m/s which could generate a risk. This high magnitude is due to that on the side of the vehicles there are less obstacles in the air way, and, as a consequence, a larger transversal area that allows a greater development of the longitudinal flow.

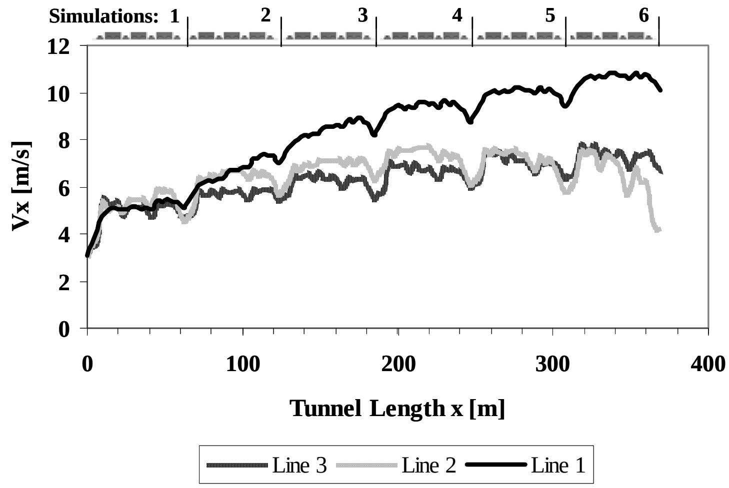


Figure 7. Longitudinal speed in the monitoring lines.

In Figure 8, it is observed how the CO concentration profile that reaches the highest magnitude is that of line 2, which maximum CO concentration is of 2,275 ppm, while the ones of line 1 and 3 reach 1,365 ppm and 926 ppm, respectively (40% and 59% less than the concentration in line 2). This behaviour is because in the line 2 region a contribution of CO is received coming from the vehicles on both lanes, which is transported in transversal direction by turbulent diffusion effect. In the case of lines 3 and 1 profiles, the one of highest magnitude is the one of line 1. This behavior is explained because on that side, CO contributions are smaller, since on the left lane, there are only small-sized vehicles located. From a safety point of view, it can be said that the concentrations are high, since on the central line, at a distance of 275 m, higher values than 1,500 ppm are reached, up to 2,300ppm.

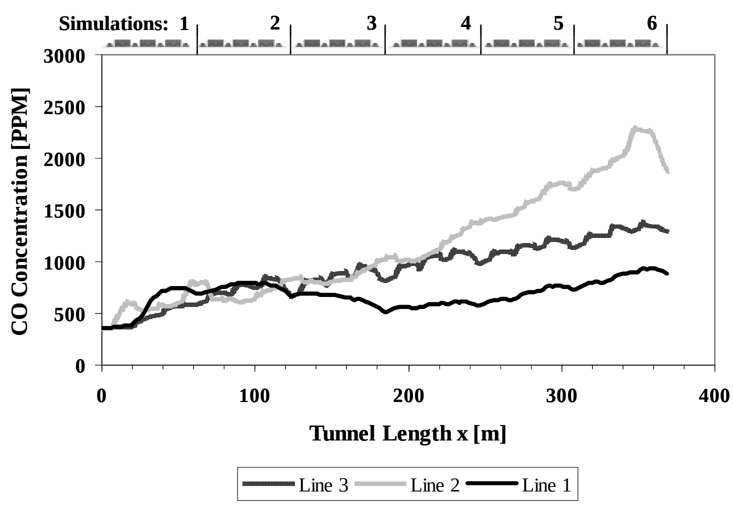


Figure 8. CO concentration in the monitoring lines.

Figure 9 shows the temperature profiles. At first, it is observed how the shape of the profiles of CO concentration and temperature are identical. This is because these two variables are represented in similar transport equations with differences only in the molecular diffusivity and configuration of the border conditions. What is remarkable to observe in Figure 9 is how the increase in temperature in the three profiles is not more than levels that could cause damage to the users, since the highest temperature reached is 311 K in the corresponding profile for line 2.

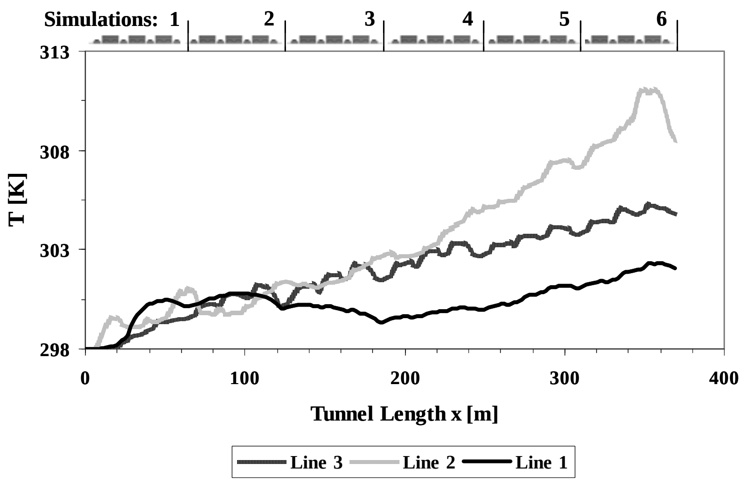


Figure 9. Temperature in the monitoring lines.

To the purpose of evaluating different scenarios that could represent a risk to the users, two additional scenarios are modeled: a) considering a CO concentration at the inlet portal of 1,100 ppm and keeping all other variables identical and b) considering a reduction of 50% of the air flow input.

For scenario (a) the temperature profiles were similar both in shape and magnitude. In the case of the CO concentrations profile, the shape of the profiles was similar but with higher magnitude, reaching a maximum value of 2,700 ppm. Likewise, for scenario (b) the profiles were similar in shape and magnitude, with the exception of the longitudinal speed profiles, which showed a lower magnitude, and reaching a maximum speed of 9 m/s. This fact is due to the reduction of air flow input.

In general, we can say that there is a strong cause- effect relationship between the injected air and the longitudinal air velocity, there being a speed distribution in the area perpendicular to the longitudinal flow, which could create risks in the event people abandon their vehicles and decide to walk out of the tunnel. Care must be taken in the flow of fresh air supplied to the tunnel, since this could originate local flow speeds of over 10 m/s. It could also be seen how the level of concentration of CO is at levels that pose a risk to users.

4. CONCLUSIONS

In this work the CO propagation in the Boquerón I tunnel was performed using Computational Fluid Dynamics (CFD).

a) The modeling strategy used, consisting of dividing the section to model in six smaller portions, sequentially modeled when introducing the obtained values for the different flow variables at the exit of a section as the entrance variables for the next section, proved to be a practical alternative for performing the modeling of large sections of tunnels with limited computational resources and easily accesible. (2.00 GHz and 1.99 GB of RAM). In this particular study, a low-cost satisfactory model of a critical section of the Boquerón-I tunnel was achieved.
b) The performed simulation allowed to determine the region in which the highest longitudinal speeds are achieved. In the study case, this corresponds to the sidewalk beside the line of vehicles, determining that after 260 m of tunnel, the longitudinal speed exceeds the safety values.
c) As opposed to model 1D, the performed simulations allowed to locate the areas of highest CO concentration. In particular, in the case of the considered analysis, it was determined that the region with the highest CO concentrations is the central region (line 2), exceeding 1,500 ppm at 275 m of tunnel, and reaching a value of 2,300 ppm at a longitude of 350 m, being these levels potentially dangerous in the case of moderate stays of time by the users inside the tunnel.
d) For the case considered, as an additional contribution to the possible results to be obtained using a 1D model, the performed modeling allowed to locate the regions in which the highest temperature readings are reached. In particular, for the case studied, it was determined that a rising temperature does not represent a risk to the people inside the tunnel.
e) The modeling strategy followed allowed to analyze other scenarios for the CO concentration as well as for the air flow at the entrance of the section under scrutiny. Results show a proportional increase in the flow variables in light of the changes both in CO concentration and in the local flow speeds. This could allow the easy extrapolation of results obtained in a particular scenario to situations in which variations in the entrance conditions to specific sections of the tunnel were considered.

5. REFERENCES

Campo, R., De Sá, L., y Ucciani, F., “Propuesta para la Recuperación, Operación y Mantenimiento de Sistemas de Ventilación en Túneles de Carretera. Caso específico: TÚNEL BOQUERÓN I”, Tekhne Revista de la Facultad de Ingeniería. Universidad Catolica Andres Bello. Vol. 3. Caracas Venezuela. (1999).

Casey, M., y Wintergeste, T., (2000), “Special Interest Group on Quality and Trust in Industrial CFD. Best Practice Guidelines”, European Research Community on Flow, Turbulence and Combustion. Version 1.0, January, (2000).

Lezama, D., y Falcón, M., “Estudio de la concentración de monóxido de carbono en los túneles “Boquerón” I y II de la autopista Caracas-La Guaira Parte II”, Revista Facultad de Ingeniería UCV., Vol. 2, Nro.1, pp 33-42, (1987).

Osawa, M., Horiuchi, H., Yoshida, K., Tada, T., y Harada, A., “A death in a stationary vehicle whilst idling: inusual carbon monoxide poisioning by exhaust gases”, Legal Medicine, 5, S132-S134, (2003).

Rivas, M., y Falcón, M., “Estudio de la concentración de monóxido de carbono en los túneles “Boquerón” I y II de la autopista Caracas-La Guaira Parte I”, Revista Facultad de Ingeniería UCV., Vol. 2, Nro.1, pp 23-32, (1987).

Vega, M., G., Argüelles, K., Fernández, J.,, Ballesteros, R., and Santolaria, C., “Numerical 3D simulation of a longitudinal ventilation system: Memorial Tunnel case”, Tunnelling and Underground Space Technology, Articulo en imprenta, (2007).

Whesterholm, R., y Egeback, K., “Exhaust Emissions from Light-and Heavy-duty Vehicles: Chemical Composition, Impact of Exhaust after Treatment, and Fuel Parameters Environmental Health Perspectives”, Volume 102, Supplement 4, October (1994).

Yuan, F., y You, S., “CFD simulation and optimization of the ventilation for subway side-platform”, Tunnelling and Underground Space Technology 22 pg 474-482, (2007).