NUMERICAL STUDY FOR THE FLOW AROUND MOVING CARS AND ITS EFFECT ON THE DISPERSION OF THE TRAFFIC EMITTED POLLUTION WITHIN A ROAD TUNNEL

Fotios Barmpas, Ioannis Ossanlis and Nicolas Moussiopoulos
Aristotle University of Thessaloniki, Laboratory of Heat Transfer and Environmental Engineering (LHTEE), Thessaloniki, Greece

INTRODUCTION

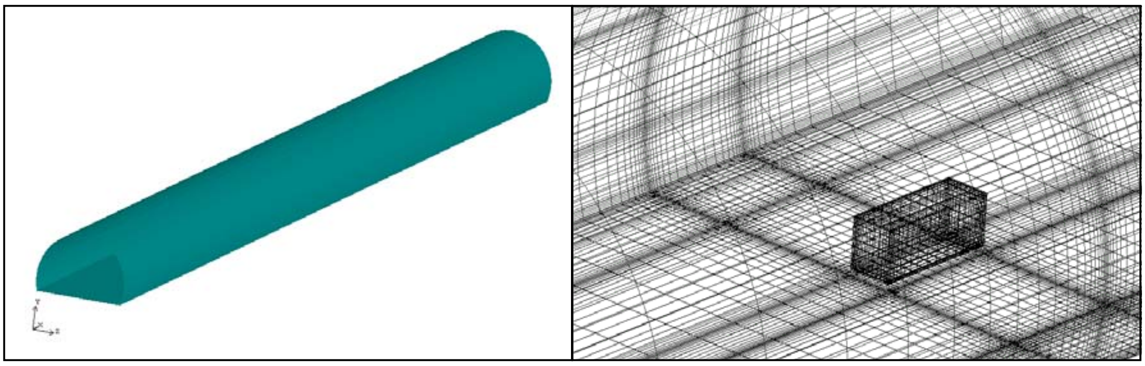
The main aim of the PICADA project was to develop a range of photocatalytic TiO_2 based materials which exhibit both de-pollution and de-soiling properties and to assess their de-pollution performance for building façade covering applications (*www.picada-project.com*). However, in addition to their application on building facades, it was also suggested to test depollution effectiveness of those materials inside road tunnels. More specifically, it was suggested to treat the inner walls of road tunnels, as an alternative means for the reduction of surface concentrations of air pollution associated with road traffic emissions inside the tunnels. Although road tunnels in general are usually well ventilated and the air inside them is renewed relatively frequently, previous research has shown that they suffer from elevated concentrations associated with road traffic emissions, mainly during the rush hours (*Indrehus O. and Vassbotn P.,* 2001).

In view of the aforementioned suggestion, it was decided to conduct a real scale trial in the Traforo Umberto road tunnel in central Rome. However, previous research regarding the flow and dispersion in road tunnels, has shown that the separation of air from a moving vehicle's side and front, gives rise to large transient aerodynamic forces, particularly in the case when the vehicle is moving close to the tunnel's side wall (*Ben Diedrichs et al.,* 2004). In addition, continuous pressure disturbances which propagate alongside the body are generated. In particular, the non-symmetric flow condition inside the bi-directional tunnel triggers substantially high pressure fluctuations (*Suzuki, M.,* 2001). Therefore, in view of these findings, a numerical study was undertaken by the Aristotle University, in order to account for the expected aerodynamic effects of the flow around the moving vehicles on the mixing levels and the dispersion of traffic emitted NO_x inside the tunnel.

METHODOLOGY

The Traforo Umberto road tunnel is a bi-directional tunnel located in central Rome, within a complex urban environment. It is 347 m long and 16 m wide with three traffic lanes, two for passenger cars and one for buses, with the buses and cars moving at opposite directions (Fig. 1a). In all cases considered, due to the complexity of the required mesh, the commercial ANSYS™ ICEM CFD 5.1 (*ANSYS*™, 2005) mesh generating code was employed, to set up a structured hexahedral mesh with sufficient refinement close to the solid walls, in order to resolve the important features of the flow. The minimum cell size was set to 0.05 m with an expansion factor varying between 1.2 and 1.3, resulting to a total mesh size of approximately 1200000 cells (Fig. 1b). In all cases, in order to model the effects of the vehicles motion, all vehicles inside the tunnel are approximated as stationary, simplified blocks without running gears, with the tunnel walls moving with the velocity of the fluid at the inlet, which was set to 50 km/hr.

For the needs of the present study, the commercially available general purpose Computational Fluid Dynamics (CFD) code ANSYS™ CFX 5.7.1 was utilized. This code uses a flexible multi-block grid system and an automatic unstructured hybrid element mesh generator with an



(a) *(b)*

Fig. 1; Tunnel geometry and mesh refinement close to a car in the middle lane approximated as a simplified block

adaptive mesh refinement algorithm. The conservation equations for mass, momentum and scalar quantities like temperature, turbulent kinetic energy and any number of species are solved. The numerical solution is based on first-order in time and second-order in space discretisation (the discretisation method is based on the Finite Volume approach). The basic discretisation technique adopted in CFX-5 is a conventional Upwind Difference Scheme (UDS) with Numerical Advection Correction (NAC) for the advection terms in the momentum and energy equations. The Reynolds stresses and turbulent fluxes of scalar quantities can be calculated by several linear and nonlinear turbulence models. Accurate modelling of the flow around simplified vehicle bodies inside tunnels is challenging due to certain characteristics of the aerodynamics and mainly the decelerations and accelerations of the flow about the nose and the tail of the vehicle, the generation and growth of boundary layers, the separation and formation of free shear layers and the chaotic wake flow. Therefore, for the needs of the study, the κ-ω Shear Stress Transport (SST) two equations turbulence closure model was selected, since it can account for the transport of the turbulent shear stress and can give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. More importantly, it does not involve the complex non-linear damping functions required for the k-ε model and is therefore more accurate and more robust.

During the study, two sets of detailed 3D numerical simulations were performed. Within the first set, the aerodynamic effects of the flow around moving cars in the middle lane on the dispersion in the wake region downstream of the cars were identified, while during the second set, the effect of the complex interaction between the flow around the moving buses close to the walls and the tunnel wall on their side was studied. Moreover, in each set two scenarios were assumed, namely the “Reference” and the “Car” scenarios. During the first scenario, only one moving vehicle was present inside the tunnel, while during the second scenario, a second moving vehicle was assumed in the wake region, downstream of the first vehicle. In all modelled scenarios it was assumed that exhaust gases were emitted only from the leading vehicle. The vehicles exhausts were approximated as area sources, with an emission rate in accordance with the EURO IV specifications for passenger cars and heavy duty vehicles.

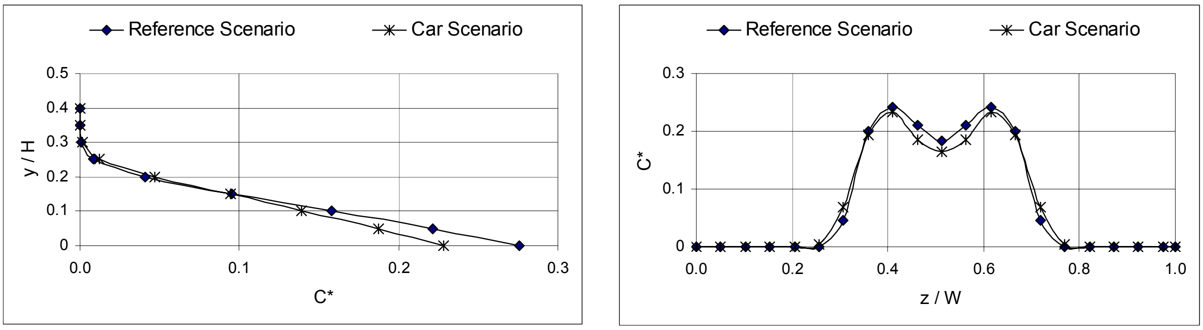
RESULTS AND DISCUSSION

Numerical results regarding the non - dimensional concentration of traffic emitted NO_x from the two set of simulations and for both scenarios, were extracted at specific locations downstream of the moving vehicles. The calculated concentrations are non-dimensionalised via the following formula:

$$C^* = CUH / (Q_s / L) \qquad (1)$$

where C* is the non-dimensional concentration, C is the predicted pollutant concentration, U is vehicle's velocity, H is the height of the tunnel, Qs is the emissions rate and L is the length of the source. In addition, all spatial quantities have been normalized on the basis of the height H and the width W of the tunnel as regards the vertical and the lateral profiles respectively. For both sets of simulations and for both scenarios, lateral and vertical non-dimensional concentration profiles were extracted at 20 m, 40 m, 60 m and 80 m from the tunnel's exit, which are all locations that lie in the wake region downstream of the second following vehicle during the second scenario.

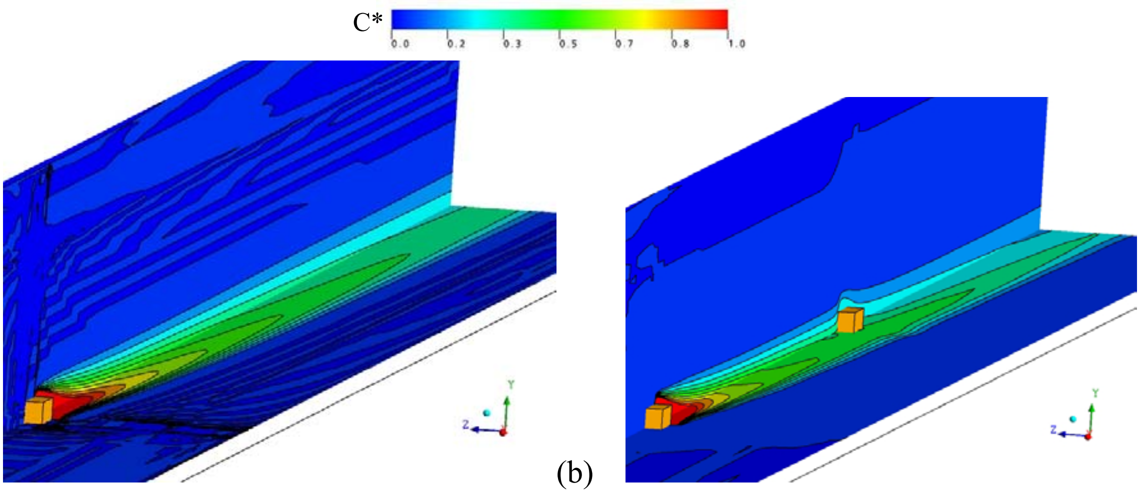
In the case of the moving cars in the middle lane of the tunnel, comparison of non-dimensional concentration profiles at the same locations inside the tunnel between the two scenarios, indicate a strong effect of the aerodynamic phenomena on the dispersion mechanism. More specifically, when a second following car is present inside the tunnel, the predicted concentrations at 80 m from the tunnel exit are considerably lower compared to the predicted ones from the "Reference" scenario (Fig. 2).



(a) *(b)*

Fig. 2; Non-dimensional concentration profiles at 80 m from the tunnel's exit in the case of the moving cars for the two scenarios in the (a) vertical and (b) lateral directions.

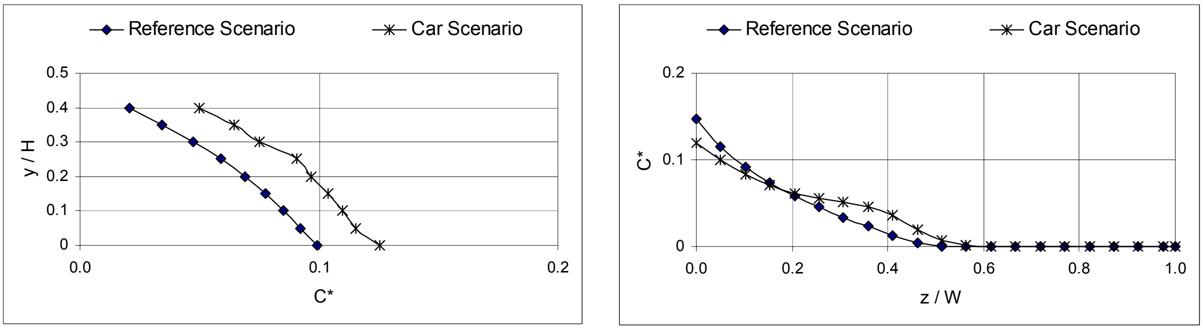
The same effect was observed at all measured locations, both in the lateral and vertical directions. However, this effect seems to be confined only within the aforementioned wake region, where due to the separation of the developing boundary layer, large coherent turbulent structures which dominate the flow such as large eddies and recirculation zones, give rise to the mixing levels and enforce dispersion (Fig. 3).



(a) (b)

Fig. 3; Non-dimensional concentration fields in the case of the moving cars for the two scenarios, in vertical and horizontal planes.

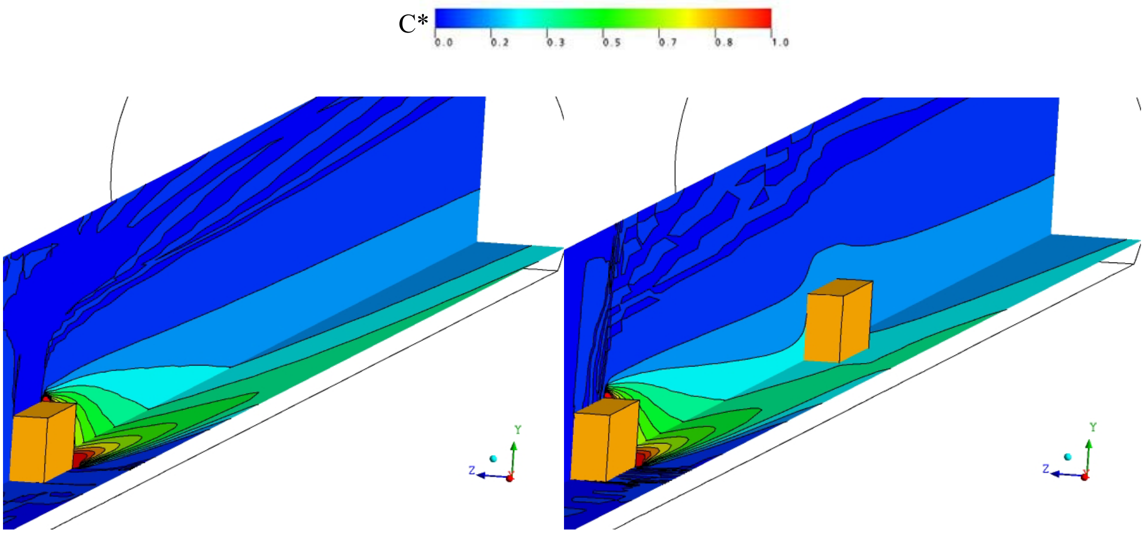
In the case of the moving bus in one of the two side lanes close to the tunnel wall, a strong effect due to the interaction between the flow around the moving buses and the tunnel walls close to the side of the buses is also indicated. The difference in this case however, is that the comparison of non-dimensional concentration profiles between the "Reference" and the "Car" scenarios at 80 m from the tunnel exit, a location which during the second scenario lies within the wake region downstream of the second following bus, shows considerably elevated concentrations for the second scenario (Fig. 4). The same effect was observed at all measured locations, both in the lateral and vertical directions.



(a) *(b)*

Fig. 4; Non-dimensional concentration profiles at 80 m from the tunnel's exit in the case of the moving buses, for the two scenarios in the (a) vertical and (b) lateral directions.

Similar to the case of the moving cars, this effect seems to be confined only within the wake region downstream of the second following bus (Fig. 4). Previous work has shown that separation of air about a moving vehicle's tail wall side inside a road tunnel, gives rise to large transient aerodynamic forces (*Diedrichs B. et al.*, 2004). Therefore, the aforementioned predicted elevated concentrations in the wake region of the second moving bus during the second scenario, could be attributed to the generation of continuous pressure disturbances propagating alongside the wake region downstream of the second moving bus, leading to intense entrapment phenomena inside that region.



(a) *(b)*

Fig. 5; Non-dimensional concentration fields in the case of the moving buses, for the two scenarios in vertical and horizontal planes.

CONCLUSIONS

Overall in conclusion, the numerical results have confirmed a strong dependence of the dispersion mechanism inside a road tunnel on the complex aerodynamic effects due to the motion of vehicles. The steady RANS simulations that were conducted within the frame of the current study using simplified models of moving vehicles, have shown that depending on the position of the moving vehicle inside the tunnel the concentrations of traffic emitted pollution at specific locations, which lie in the wake of these moving vehicles, can be largely affected. In particular, the flow separation at the wall side of the vehicle models and the resulting aerodynamic forcing can lead to a considerable increase in the predicted concentrations due to the formation of intense entrapment phenomena.

However, considerably more research on the field is necessary, since the current issue which is addressed depends on many other parameters which for the needs of the current study have not been taken into account, such as the number of moving vehicles inside the tunnel, the realistic shape of the various vehicles and the running gears. More importantly, due to the unsteady nature of the aerodynamic effects of the flow around moving objects, further research on the basis of advanced LES modeling techniques is necessary.

REFERENCES

ANSYS™ CFX 5.7.1, 2004: User Manual, Ansys Inc.

ANSYS™ ICEM 5.1, 2005: User Manual, Ansys Inc

Diedrichs B., Berg M. and Krajnovi S., 2004: Large Eddy Simulations of the Flow Around High-Speed Trains Cruising Inside Tunnels, *Proceedings of European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS 2004 Jyväskylä, 24—28 July 2004*

European Project A-TEAM (Advanced Training System for Emergency Management), http://www.ess.co.at/A-TEAM/

European Project PICADA (Photocatalytic Innovative Covering Applications for Depollution Assessment), http://www.picada-project.com

Indrehus O., Vassbotn P., 2001: CO and NO_2 pollution in a long two-way traffic road tunnel: investigation of NO_2/NO_x ratio and modelling of NO_2 concentration, *Journal of Environmental Monitoring* **3**, No. 2, 221-225

Karatzas K., Ossanlis I., Kaprara A., Louka P. and Moussiopoulos N., 2003: Tunnel fire smoke modelling for emergency management, *International Journal of Risk, Assessment and Management,* **4**, No. 1, 52-66

Murakami. S.: 1998. Overview of turbulence models applied in CWE-1997. *Journal of Wind Engineering and Industrial Aerodynamics* **74-76**, 1-24.

Ossanlis I., Karatzas K., Kaprara A., Louka P., and Moussiopoulos N. 2002: Wind Field and Fire Simulations in Tunnels, Proceedings of the 6th Conference on Protection and Restoration of the Environment Skiathos, Greece, 1-5 July

Perzon. A. and Davidson. L. 2000: On CFD and transient flow in vehicle aerodynamics, *SAE 2000-01-0873*.

Suzuki M. 2001. Unsteady Aerodynamic Force Acting on High Speed Trains in Tunnel. QR of RTRI, **42**, No. 2.