Military Technical College
Kobry El-Kobbah,
Cairo, Egypt

14th International Conference on
Applied Mechanics and
Mechanical Engineering.

Numerical Prediction of the Critical Velocity for Forced Ventilation Road Tunnel

By

N. S. Khadour* A. A. ElMaihy* A. G. M. Ibrahim* S. E. Elshamarka*

Abstract:

Reliable design of tunnel ventilation system requires knowledge of fire-induced smoke behaviour. The critical velocity to prevent upstream smoke propagation in the event of a tunnel fire is an important parameter in the design process of a ventilation system. The current techniques for prediction of the values of the critical velocity are mainly based on semi-empirical equations. These equations are obtained from the Froude number preservation combined with available experimental data. Full-scale experimental test programs are expensive which stimulates the application of numerical simulation to these design problems. Simulated full-scale point supply ventilation system of Memorial Tunnel in USA is investigated. Three-dimensional simulation of smoke flow in the tunnel is carried out using FLUENT 6.3.The model includes component models for turbulence, fire, radiation heat transfer, and smoke production. The impact of fire size on critical velocity is investigated. The findings from the numerical simulations supported the published results of experimental test program and also the three semi-empirical correlations found in literature. This suggests that the simulation techniques can be used with confidence to predict the critical ventilation velocity for large-scale tunnels. Moreover, it recommends the point supply ventilation system as a competitive alternative for conventional longitudinal ventilation system.

Keywords:

Road tunnel, Ventilation, Point supply, Critical velocity and Simulation

* Egyptian Armed Forces

1. Introduction:

In the event of a road tunnel fire and due to its confining geometry, smoke generated from the fire could severely impede visibility and evacuation. Ventilation systems are crucial in the design of all road tunnels. They maintain acceptable levels of contaminants produced by vehicles during normal traffic operation (normal ventilation), and control heavy smoke in event of fire (emergency ventilation). Mechanical ventilation systems such as longitudinal, transverse and semi-transverse ventilation systems are commonly employed. Such ventilation systems have to be designed with accuracy to control the longitudinal motion of the fire-induced smoke. Sufficient ventilation capacity is needed to prevent smoke movement upstream of the fire, that is, to prevent backlayering. This ventilation rate (average flow velocity) is often referred to as the critical velocity; V_{cr}. Critical velocity is, therefore, the minimum ventilation velocity that is able to prevent smoke backlayering in a tunnel fire. The backlayering phenomenon is illustrated in figure (1).

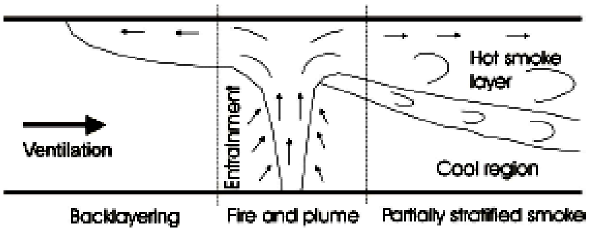


Fig (1): Backlayering of smoke in tunnels, [1]

Ventilation requirements are specified by national codes and international standards. However, performance-based design of tunnel emergency ventilation systems can provide greater benefits than designing a system utilising a code or standard. A code compliant ventilation system design may, in some circumstances, result in the system being over sized or may not perform as expected. Therefore, designing the system on performance basis may achieve cost and design benefits. There are many theoretical and experimental studies of critical ventilation velocity in tunnel fires (refer to the review by Roh et al. [2]).

The most commonly used methods for prediction of the values of the critical ventilation velocity for various fire sizes are mainly based on semi-empirical relationships obtained from Froude number preservation using some experimental data. Three widely used semi-empirical correlations will be presented briefly in the following.

The most widespread method of calculating critical velocity is that described by Kennedy et al. [3]. It is essentially a one-dimensional analysis based on the perfect gas law, and assumes steady state behaviour, negligible smoke mass generation and no heat transfer to the walls. The fire is assumed to be ‘wide’ compared to the tunnel geometry, such that ventilating air cannot pass around the source of the fire and therefore all ventilating air acts to cool the fire. Fully mixed behaviour is assumed. This will not be the case for smaller fires and this method will therefore under-predict

the critical velocity for these smaller fires. The method over-predicts the critical velocity for large fires, as shown by comparison with test measurements from the Memorial Tunnel Fire Ventilation Test Program (MTFVTP), [4].
Bakar et al. [5] used the tunnel mean hydraulic diameter, $H_{D,}$ to replace the tunnel height H as the characteristic length in the dimensionless analysis given by Oka et al. [1]. They concluded that there are two regimes of variation of critical velocity against fire size. At low fire heat release rate (HRR), the critical velocity varies as the one third power of the HRR however, at higher rates the critical velocity become independent of the HRR.

By using a two-dimensional approach, Kunsch [6] developed a model for the critical ventilation velocity in dimensionless groups form. The uncertainties in critical velocity values for the aforementioned semi-empirical correlations grow dramatically in case of peculiar tunnel configurations or in case of high slopes. The uncertainty which arises in such situations is frequently compensated by means of costly safety margins.

It is not practical to investigate every configuration experimentally, hence predictive models are required. The most recent technique to study the behaviour of emergency tunnel ventilation system is by using computational Fluid Dynamics (CFD). This approach is capable of modelling the multi-dimensional fire induced smoke in tunnels of arbitrary geometry. Different CFD software packages were used such as Fire dynamics simulator FDS [7], JASMINE [8], Flow 3D [9], PHONICS [10], SOLVENT [11] and FLUENT [12, 13]. However, ssuccessful numerical simulations of fires in large complicated structures, such as tunnels, are largely dependent on the accuracy of the CFD model and input data such as; initial conditions of the tunnel environment, ventilation flows, and most importantly, the HRR from the burning objects. Wide spread comprehensive research studies to effectively measure these input parameters; especially HRR of different burning materials were conducted. The tunnel fire test series EUREKA “FIRETUN”, carried out between 1990 and 1992 by teams of fire researchers representing Austria, Finland, France, Germany, Italy, Norway, Sweden, Switzerland and UK [14]. The MTFVTP [4] was the largest tunnel fire test series to date in terms of actual scale where 98 pool fire tests ranging in size from 10 MW to 100 MW were carried out combined with extensive investigations on the critical velocity.

The longitudinal ventilation systems are commonly installed to prevent upstream smoke flow and offer a safe evacuation path. Point supply ventilation system is a competitive alternative especially where natural ventilation is not recommended for long tunnels in event of fire. Point supply ventilation introduces unidirectional airflow through a single opening located in the ceiling of the tunnel. There are uncertainties in the current correlations for prediction of the critical ventilation velocity for point supply ventilation. Therefore, the impact of the fire size on critical ventilation velocity of such systems warrants further investigation. Thus, point supply system has been selected here for analysis where CFD is used to assess its performance under fire conditions.

This paper concerns the specification of critical velocity of point supply ventilation system. Detailed three-dimensional simulation of smoke distributions in the tunnel and the impact of changes in fire size on the critical velocity are investigated. The numerical results are validated against the data from the MTFVTP and also three semi-empirical correlations algorithms found in literature namely; Kennedy et al. [3], Bakar et al.[5] and Kunsch [6].

2. CFD Simulation Technique

CFD simulation was used to predict the critical velocity for point supply ventilation system of the Memorial Tunnel, test case 321A of the MTFVTP, [4]. Simulations were made using a commercial CFD package FLUENT ver. 6.3 with its pre-processor GAMBIT [15].

2.1 The physical Domain

The Memorial Tunnel has a two-lane, 853 meter long road tunnel, figure (2). It is a part of the West Virginia Turnpike near Charlestown, West Virginia, USA. The tunnel is 3.2% upgrade from south to north portal. The cross-sectional area of the tunnel is 60.4m^2, as shown in figure (3), without ceiling and 36.2m^2 when the ceiling is in place. Test 321A consisted of a single point supply of air through a 28m^2 opening on the ceiling located 135m north of the fire centre line. All the standard ports and supply flues in the ceiling were closed.

Four steel pans were installed at the approximate quarter point of the tunnel (238m) from the south portal in which fuel (low sulfur No. 2 fuel oil) were burned to generate nominal HRR running from 10 MW to 100 MW.

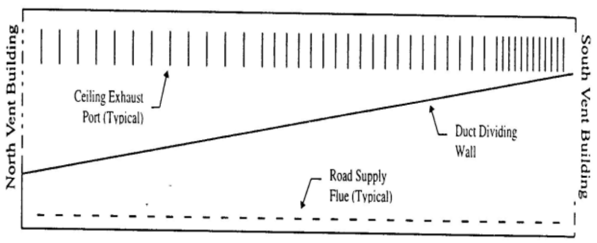


Fig (2): Plane view of the original Memorial Tunnel duct and ceiling, [4].

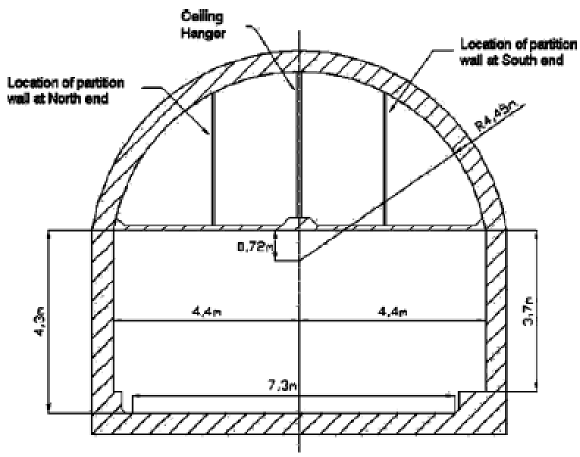
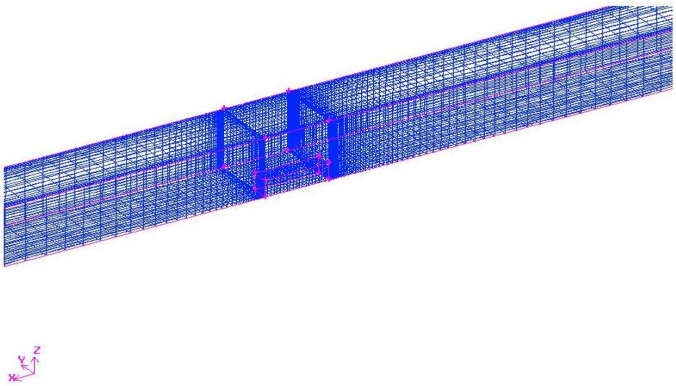


Fig (3): Memorial tunnel cross-section looking north, [4]

2.2 The Computational Domain

The length of the calculation domain was taken shorter than the length of the tunnel. In previous work [8], it has been shown that modeling of 100m north of the point supply is sufficient to acquire all the necessary information. So, modeling more than this length isnot necessary and only increases the calculation time. In order to halve the calculation time and the memory needed, the tunnel was considered to be symmetric about its vertical longitudinal centre-line. The north portal of the tunnel was completely closed to force all the air supplied by the fan past the fire site.

The simulated tunnel was represented by a duct of length 476m long (in X direction) with rectangular cross-section of dimensions 4.2m (in Y direction) and 8.8m (in Z direction). The longitudinal grid distribution near fire source and cross-sectional grids are shown in figure (4). The inlet port (point supply opening) is 28m^2 area (4 m in X direction and 7m in Z direction centred about the plane of symmetry at 135m north the centre-line of the fire source). The exit port is the south portal of the tunnel. The fire is represented by a rectangular parallelepiped of height 1m above tunnel floor. The mid-section of the parallelepiped is located at 238m from the south portal.



Fig(4): Grid - (a) near the fire location,

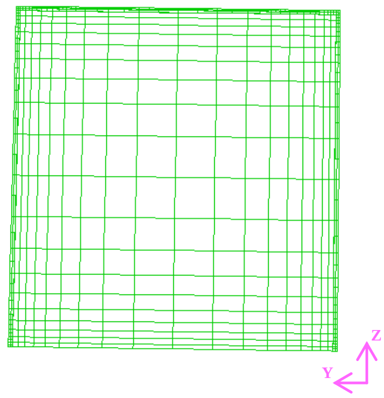


Fig (4): Grid - (b) cross-sectional area

Submap cells were employed in the building of the mesh. Different zones with refined meshes were also introduced where required. Different grids A, D and E were used to test the dependence of the solution on the grid size. The total number of cells for grids A, D and E were 170548, 310582 and 379312, respectively. It was found that grid D using (20×20) cells in transversal section with 800 cells along the longitudinal direction of the tunnel satisfied the grid sensitivity test, figure (5).

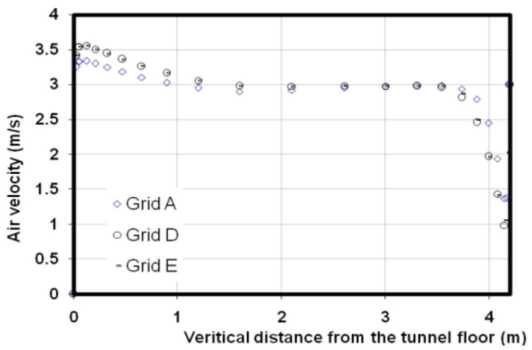


Fig (5): Velocity profile in transversal section at the center of supply opening (x =372m)

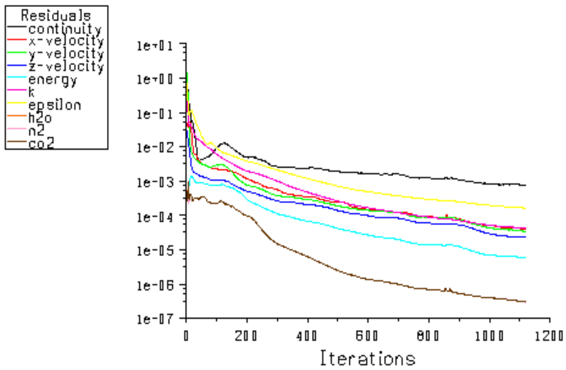


Fig (6): A sample of residuals history versus number of iterations

2.3 Governing equations

Based on finite volume approach the flow field is solved using Reynolds-averaged Navier-Stokes (RANS) equations. The energy equation is used to predict temperature field. The standard k-ε model was used to model turbulent flow. The governing equations and default values of model constants are listed in [15].

The standard wall functions in FLUENT based on the proposal of Launder and Spalding [16] is used. The value of y^+ was checked for the tunnel walls namely, tunnel floor, tunnel ceiling and the side wall. For most of the tunnel length the value of y^+ lays between 30 and 100.

The species equation was solved for N-1 species where N is the total number of fluid phase chemical species presented in the system. In this work, the mixture has 4 species namely, air, CO_2, H_2O and N_2.

2.4 Representation of Fire

Most of the fire simulations [7, 11, 13, 17] were made without combustion models. Therefore, fire was modelled using a volumetric heat source and a source of mass and species flow corresponding to the combustion products. The fire conditions were simulated using a rectangular parallelepiped whose volume varied with the fire size. The choice of fire source volume and corresponding HRR was based on recommendations of NFPA 502, [18] and PIARC, [19]. The mass flow corresponding to the combustion products was introduced through the surfaces of the parallelepiped. The heat release rate $\dot{Q}$ was computed from the rate of fuel consumption $\dot{m}_f$, the heating value of the fuel H_l and combustion efficiency η, as:

$$\dot{Q} = \dot{m}_f H_l \eta \qquad (1)$$

In the fire region, additional source terms were included in the mass continuity, species equations and energy equations. For the continuity equation, the source term is calculated from the specified rate of fuel consumption and the stoichiometric ratio for the fuel assuming complete combustion. Thus,

$$\dot{m}_{Comb.\,\mathrm{Pr}od.} = \dot{m}_f (1+S) \qquad (2)$$

where S is the stoichiometric ratio (kg of air/kg of fuel) for the fuel.

2.5 Representation of Radiation Heat Transfer

The radiative fraction approach was used. Thermal radiation in the participating medium was ignored and a fixed fraction of the total heat released in the fire was assumed to be lost to the surroundings without affecting the temperature distribution within the tunnel. The remaining energy was transported away by the fluid. Experiments on diffusion flames indicate that the radiation fraction typically lies in the range 0.2 to 0.4 [17]. Thus, the convective energy Q_c was estimated by

$$\dot{Q}_c = \dot{m}_f H_l \eta (1-R_x) \qquad (3)$$

where R_x is the radiative fraction and taken as 0.3.

2.6 Boundary Conditions

Boundary conditions specify the flow and thermal variables on the boundaries of the physical model. Therefore, they are a critical component of CFD simulations and it is important to be specified appropriately.

There are two portals in the computational domain, the inlet flow port (point supply opening) and the tunnel exit port. Velocity inlet boundary conditions were used to define the velocity and scalar properties of the flow at inlet port. The inlet velocity ranged from 1.5 to 4 m/s. The turbulence was specified at the inlet port by turbulent intensity and hydraulic diameter. Outflow boundary condition was used to model flow exits where the details of the flow velocity and pressure were not known prior to solution of the flow problem.

Tunnel walls were set as stationary walls. The no-slip condition was imposed on the solid surfaces. Heat flux was specified on the walls to be zero. The tunnel walls material was dolomite.

The fire was set as a fluid zone from which mass and energy were released to the tunnel. The lower surface was set as wall while the other surfaces were set as interior boundary conditions to permit energy and mass sources to cross to the tunnel. The values of the source terms used to define the fire were calculated based on the HRR and the combustion equation assuming stoichometric air to fuel ratio. The mid longitudinal section of the tunnel and the rectangular parallelepiped representing the fire were set as symmetry planes.

2.7 Solution Procedures

A three dimensional, steady, implicit pressure based solver was used. The pressure-velocity coupling was calculated through the Simple scheme. Second order upwind discretization was used for momentum, kinetic energy k, energy dissipation rate ε and energy equations while first order upwind was used for species equation. Gravitational body force was included in the momentum equation, defining the buoyancy terms function of the temperature variation according to Boussinseq approach. The convergence was monitored dynamically by checking residuals. The residuals decrease to 10^{-3} for all equations except the energy equation, for which the criterion is 10^{-6}. Generally, 800-1200 initiations were required to obtain a suitable level of solution convergence. Figure (6) shows a sample of the residuals history with the number of iterations. The residuals history was obtained for all runs and they behaved similarly. The numerical results were obtained using PC (3.0 GHz Pentium 4), 2 GB RAM, and 120 GB hard disk. Each calculation required about 8 hours of CPU time and 40-50MB of hard disk memory.

3. Results

Steady-state simulations were carried out for the previously mentioned computational domain and component models with different values of longitudinal velocity in the range from 1.5 to 3 m/s and fire HRR namely 5, 10, 20 and 40 MW.
The simulation was run with a selected ventilation velocity (volumetric flow rate) and the formation of the back-flow was checked. Runs were carried out until a ventilation velocity encompassed the conditions exhibiting back-flow and no back-flow along the tunnel ceiling relative to the upstream edge of the fire zone.

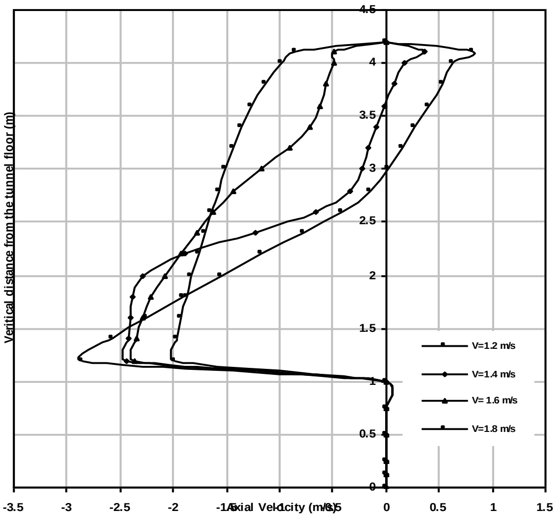


Fig (7): Axial velocity at x=240m and 5MW HRR

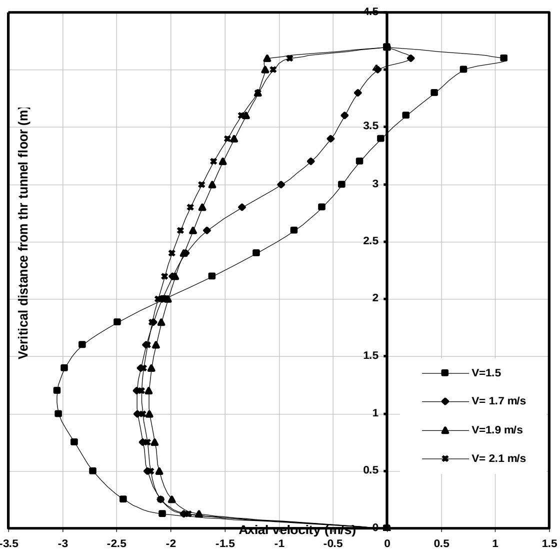


Fig (8): Axial velocity at x=240m and 10MW HRR

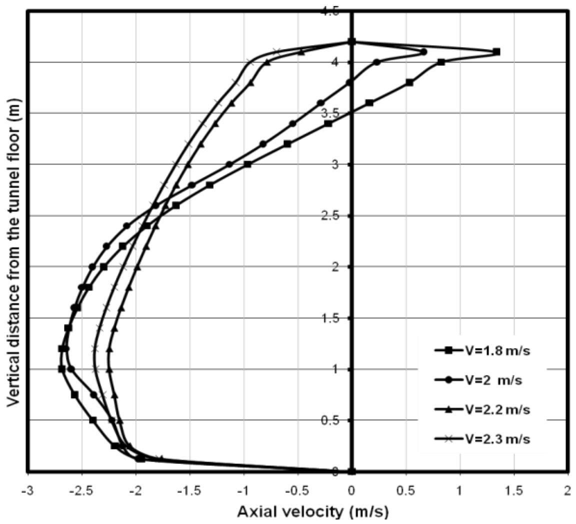


Fig (9): Axial velocity at x=240m and 20MW HRR

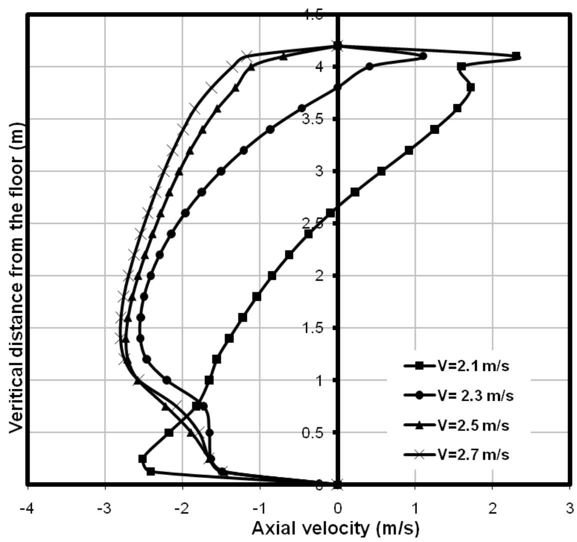


Fig (10): Axial velocity at x=240m and 40MW HRR

The critical velocity was determined by plotting the axial velocity against a vertical line located in the symmetry plane at the upstream edge of the rectangular parallelepiped representing the fire (x=240 m). Figures (7) to (10) show the axial velocity against tunnel height for 5, 10, 20 and 40 MW fire size, respectively. Backlayering occurs when the axial velocity are in the positive X-direction. Critical velocities were found to be 1.6, 1.9, 2.2 and 2.5 for fire sizes of 5, 10, 20 and 40 MW, respectively.
Table (1) is a comparison of the current CFD results, experimental results of the Memorial Tunnel and three semi-empirical correlations. The reported critical velocities in MTFVTP are little higher than the current simulation results, and this deviation is smaller than those of semi-empirical correlations. The deviation of CFD results could be accepted as it is not significant. The reason for this deviation appears to arise from the approximations made in the numerical model by ignoring the combustion and radiation models. Also, the measurement uncertainty was not given in the MTFVTP

report but cannot be neglected.

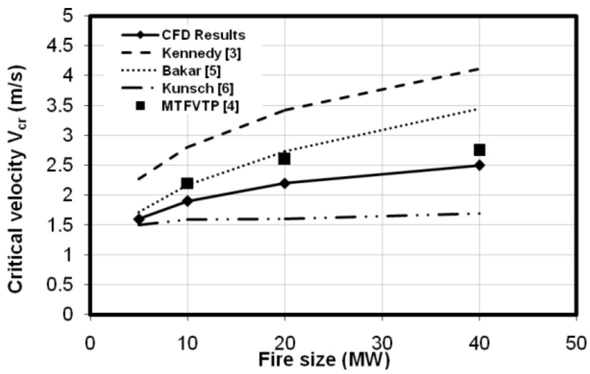


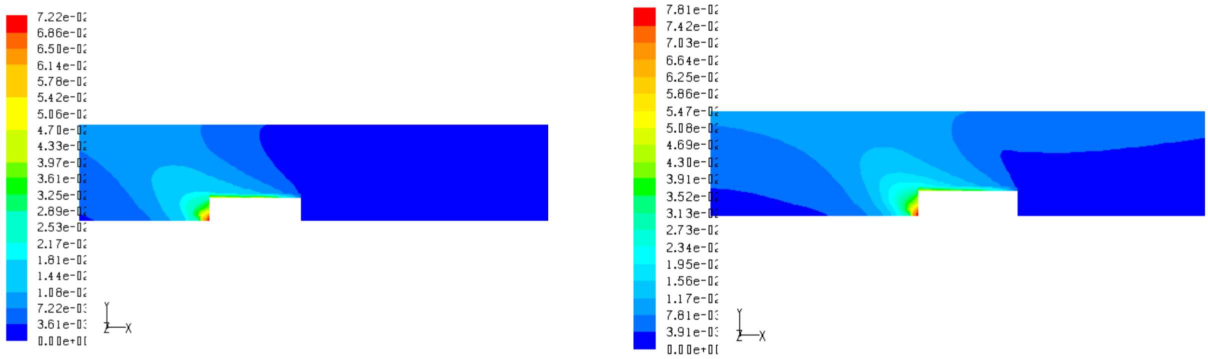
Fig (11): Critical velocity comparison

Fire Size MW	Critical Velocity (m/s)				
	CFD	MTFVTP [4]	Kenndy et al. [3]	Baker et al. [5]	Kunsch [6]
5	1.6	----	2.27	1.72	2.4
10	1.9	2.2	2.8	2.17	2.54
20	2.2	2.6	3.42	2.74	2.63
40	2.5	2.75	4.11	3.45	2.67

Table (1): Comparison between current CFD results and published results

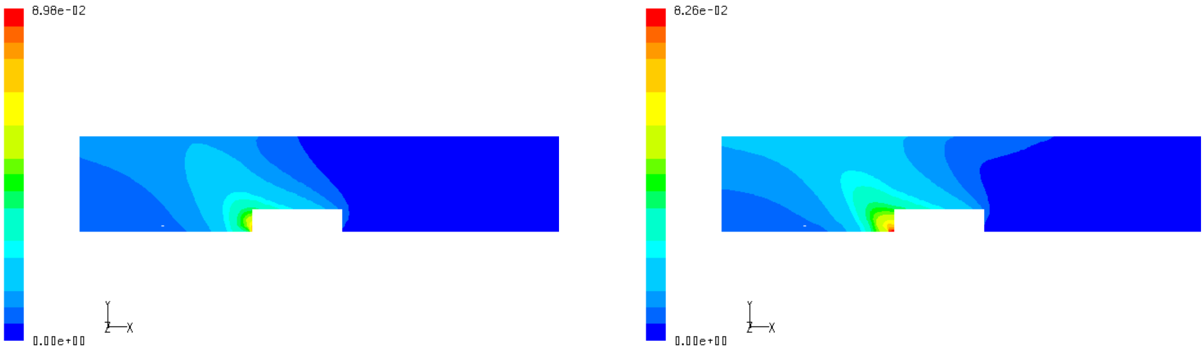
The plot of critical velocity based on semi-empirical correlations and those predicted as a function of HRR is shown in figure (11). The values representing MTFVTP measurements are also shown for comparison. CFD results give better agreement with the experimental trend compared with semi-empirical correlations especially at high HRR. It is noted that these correlations are based on analyses of one-dimensional or two-dimensional flows while the CFD results are based on three-dimensional flow in tunnel. Moreover, these correlations do not include the fire-source geometry.

The simulation results captured well the backlayering phenomenon as shown in figures (12) to (15) where CO_2 concentration contours in computational domain were plotted at different ventilation velocities. Examining these figures shows the minimum velocity at which no backlayering takes place. The demarcation line of the two regions separating back-flow and no back-flow defines the critical ventilation values. Figure (12) shows the predicted CO_2 distribution at the symmetry plane for 5 MW fire size. The numerical simulation captures the features of the backlayering, and downstream smoke flow. The predicted critical velocity is about 1.6 m/s. The CO_2 contours at the symmetry plane for 10 MW HRR is shown in figure (13). As indicated, the increase in HRR is coupled with increase in the critical velocity. The predicted critical velocity for 10 MW fire size is about 1.9m/s. Figure (14) shows the CO_2 contours at the symmetry plane for 20 MW. The predicted critical velocity is estimated as 2.2 m/s. Finally, the critical ventilation velocity of 2.5 m/s can be shown in figure (15) for HRR of 40 MW. The values determined from CO_2 contours are the same as determined from figures (6) to (9).



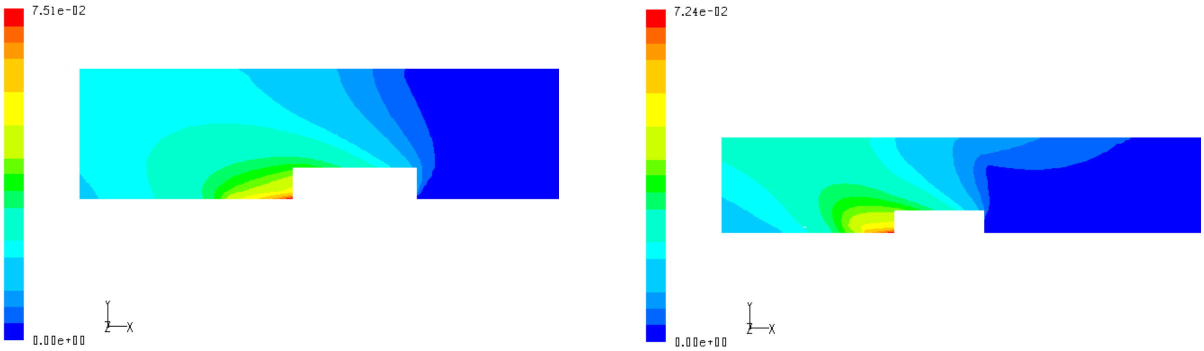
V= 1.6 m/s V= 1.4m/s

Fig (12): CO_2 concentration for fire size of 5 MW and different ventilation velocity



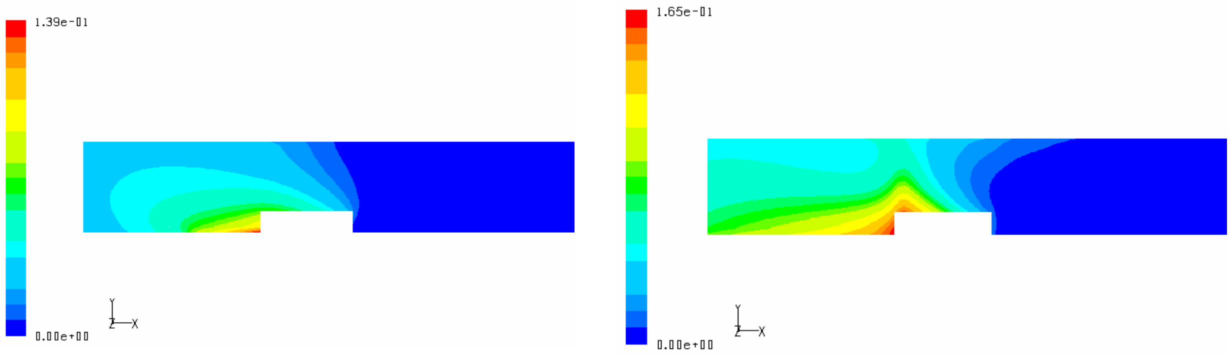
V= 1.9 m/s V= 1.7 m/s

Fig (13): CO_2 concentration for fire size of 10 MW and different ventilation velocity



V= 2.2 m/s V= 2 m/s

Fig (14): CO_2 concentration for fire size of 20 MW and different ventilation velocity



V=2.5 m/s V=2.3 m/s

Fig (15): CO_2 concentration for fire size of 40 MW and different ventilation velocity

4. Conclusions:

The control of smoke flow during a tunnel fire is crucial in the design of tunnel ventilation systems. Evaluation of emergency ventilation system of a tunnel can be done either experimentally or using full scale 3-D numerical simulation, where the critical velocity is taken as a sole parameter to assess the performance of ventilation system. In literature there are many correlations for estimation of longitudinal ventilation critical velocity based on Froude number preservation combining with available experimental data. The main purpose of this paper was to use CFD technique to simulate the smoke flow and predict the critical velocity of point supply ventilation system for full-scale road tunnel. CFD code, FLUENT 6.3, was employed to predict the smoke flow under different sizes of tunnel fires. The results of critical velocity were compared with existing point supply experimental results and semi-empirical correlations of longitudinal ventilation critical velocity. The agreement with these results and the CFD results was fair. The following observations were made:

- The present correlations of critical velocity for longitudinal ventilation are valid for point supply ventilation system due to similar developed flow field.
- The point supply ventilation system, even it is of simpler configuration, attained a similar critical velocity as longitudinal ventilation system.
- CFD predicts well the backlayering phenomenon for different fire sizes of road tunnel.
- CFD is very useful tool in both design-phase and design validation phase. It provides cost benefits by ignoring the costly safety margins needed to compensate the uncertainty in critical velocity values predicted by current empirical correlations.

References:

[1] Oka Y., and Atkinson G.T., “Control of smoke flow in tunnel fires”, Fire Safety J; 25:305–22, 1996.

[2] J. S. Roh, H. S. Ryou, D. H. Kim, W. S. Jung, and Y. J. Jang; “ Critical velocity and burning rate in pool fire during longitudinal ventilation”, Tunnelling and Underground Space Technology 22 : 262–271, 2007.

[3] Kennedy W.D, Gonzales J.A, and Sanchez J.G., “Derivation and application of the SES critical velocity equations”, ASHRAE Trans: Research; 102(2):40–4, 1996.

[4] MHD/FHWA, “Memorial Tunnel Fire Ventilation Test Program”, Comprehensive Test Report, Massachusetts Highway Dept., Federal Highway Administration, Washington, DC, 1995.

[5] Wu Y. and Bakar M.Z.A., "Control of Smoke in Tunnel Fires Using Longitudinal Ventilation Systems- a Study of the Critical Velocity", Fire Safety J; 35: 363-390, 2000.

[6] Kunsch JP., “Simple model for control of fire gases in a ventilated tunnel”, Fire Safety J; 37:67–81, 2002.

[7] Cochard S., "Fire Dynamics Simulator Version 2.0 for simulating tunnel fires", Tunnel Management International, Vol. 6, Issue 4, 2003.

[8] Kumar, S. and Cox, G., "Mathematical Modeling of Fire in Road Tunnels -Effects of Radiant Heat and Surface Roughness”, Transport and Road Research Laboratory Contractor Report No.101, Crowthom, 1988.

[9] Armstrong, J., Bennett, E.C., Matthews, R.D., Smith, T.W. and Tabarra, M., "The Ventilation of Vehicle Tunnels by Jet Fans - The Axi-symmetric Case," Proceedings of the Seminar on Installation Effects in Fan Systems, pp. 59-64, Mechanical

Engineering Press, London, 1993.
[10] Rhodes, N., "Smoke Modeling & Critical Velocity in Tunnels", ITC Conference, London, 1996.
[11] Ballestros R., Santolaria C. and Blanco E., "Influence of the slope in the ventilation Semi-transversal System of an Urban Tunnel", Tunneling and Underground Space technology 31 ; 21-28, 2006.
[12] Kang K., and MacDonald H. M., "Computational study of longitudinal ventilation control during an enclosure fire within a tunnel", Journal of fire protection Engineering, vol. 16-August 2006.
[13] Vega M. G., Diaz K. M. A., OroJ. M. A., Ballesteros R., and Sanrolaria C., "Numerical 3D simulation of a longitudinal Ventilation systems: Memorial Tunnel Case", Tunneling and underground space technology 2007.
[14] EUREKA-Project EU499: FIRETUN, "Fires in Transport Tunnels: Report on full-scale tests", D-40213 Dusseldorf ,1995.
[15] FLUENT 6.2 Documentation, © Fluent Inc. 2005-01-04.
[16] Launder, B. E., and Spalding, D. B., "The Numerical Computation of Turbulent Flows", Comp. Meth. App. Mech. Eng.; 3: page 269, 1974.
[17] Kailash C. Karki and Suhas V. Patankar, "CFD Model for jet fan ventilation systems", Published in proceeding of the 10th international symposium on Aerodynamics and ventilation of vehicle tunnels principles and design 1-3 November 2000. Boston USA.
[18] National Fire Protection Association, "NFPA 502 Standards for Road Tunnels, Bridges, and Other Limited Access Highways", NFPA, Quincy, MA, USA, 2001.
[19] Fire and Smoke Control in Road Tunnels, PIARC Committee on Road Tunnels (C5), 1999.

Nomenclatures

H_l	Lower heating value of fuel (kJ/kg)
k	Kinetic energy of turbulence (m^2/s^2)
$\dot{m}_f$	Mass consumption rate of the fuel (kg/s)
$\dot{m}_{Comb.\,Prod.}$	Mass flow rate of the combustion products (kg/s)
$\dot{Q}$	Heat release rate from the fire (MW)
Q_c	Convected energy by the fluid (MW)
R_x	Radiative fraction
S	Stoichiometric air to fuel ratio
V_{cr}	Critical velocity (m/s)
y+	Dimensionless no (Local Reynolds based on mean velocity in viscous sub-layer)
ε	Turbulence dissipation rate (m^2/s^3)
η	Combustion efficiency

Abbreviation

CFD	Computational Fluid Dynamics
HRR	Heat Release Rate
MTFVTP	Memorial Tunnel Fire Ventilation Test Program
NFPA	Standard National Fire Protection Association
PIARC	Permanent International Association of Road Congresses