International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:2, No:4, 2008

Experimental and Numerical Simulation of Fire in a Scaled Underground Station

Nuri Yucel, Muhammed Ilter Berberoglu, Salih Karaaslan, and Nureddin Dinler

Abstract**—**The objective of this study is to investigate fire behaviors, experimentally and numerically, in a scaled version of an underground station. The effect of ventilation velocity on the fire is examined. Fire experiments are simulated by burning 10 ml isopropyl alcohol fuel in a fire pool with dimensions 5cm x 10cm x 4 mm at the center of 1/100 scaled underground station model. A commercial CFD program FLUENT was used in numerical simulations. For air flow simulations, k-ω SST turbulence model and for combustion simulation, non-premixed combustion model are used. This study showed that, the ventilation velocity is increased from 1 m/s to 3 m/s the maximum temperature in the station is found to be less for ventilation velocity of 1 m/s. The reason for these experimental result lies on the relative dominance of oxygen supply effect on cooling effect. Without piston effect, maximum temperature occurs above the fuel pool. However, when the ventilation velocity increased the flame was tilted in the direction of ventilation and the location of maximum temperature moves along the flow direction. The velocities measured experimentally in the station at different locations are well matched by the CFD simulation results. The prediction of general flow pattern is satisfactory with the smoke visualization tests. The backlayering in velocity is well predicted by CFD simulation. However, all over the station, the CFD simulations predicted higher temperatures compared to experimental measurements.

Keywords**—**Fire**,** underground station, flame propagation, CFD simulation, k-ω SST turbulence model, non-premixed combustion model.

I. INTRODUCTION

ACCURATE prediction of fire and smoke propagation in an underground transport station becomes more and more important for designing efficient fire protection systems. Due to the large amounts of people travel through underground railways daily, the fire safety inside tunnels and stations have gained importance after recent catastrophic fires caused heavy casualties and tremendous loss of property. It is necessary to design an effective fire protection system. In general, fires are very complex in nature, such as turbulence, combustion radiation, combustible materials, fire location, space geometry, etc., which affect the fire and smoke propagation. The experiments in a scaled underground station provide useful information. However, they are not sufficient to provide full-scale features.

Park et al. [1] conducted a numerical study to evaluate of fire outbreak in an underground station. They took measurements from an actual underground station platform for numerical analysis to investigate the ventilation of the station and smoke in case of fire. The velocity measured at various points at the platform was compared with the results obtained by numerical analysis. It was shown that the capacity of exhausts installed exerts a significant influence on the movement of heat and smoke in case of fire.

The critical velocity is defined as the minimum air velocity required suppressing the smoke spreading against the longitudinal ventilation flow during tunnel fire situations. Wu and Baker [2] investigated the relationship between the critical velocity and heat release rate of the fire and the effect of the tunnel cross-sectional geometry to the critical ventilation velocity both experimentally and numerically. Experimental tests have been carried out using a series of small scale tunnels having same height but different cross-sectional geometries. Dimensionless velocity and heat release rate with tunnel hydraulic height (tunnel mean hydraulic diameter) as the characteristic length was used in the experimental data analysis. It was shown that the experimental data can be correlated in a simple formula which can be used for scaling. In addition, the CFD predictions have been validated against the experimental measurements.

Three different combustion models, the volumetric heat source (VHS) model, the eddy-break up model and the presumed probability model (prePDF) in enclosure fire simulation are examined numerically by Xue et al. [3]. The results of different combustion models are compared and evaluated with available experimental data. They found that the VHS model performs well at the measuring station close to the fire for shopping mall fire, while the prePDF model appears to perform best at the upstream and relatively far away from the fire source. For the tunnel fire, the eddy breakup model and prePDF model perform equally well.

Li and Chow [4] studied different tunnel fire scenarios, numerically. The tunnel concerned was simplified as a very long rectangular tube. Fire size, ventilation system and heat capacity were considered as parameters. Based on the results,

Manuscript received March 28, 2008. This work was supported by the Scientific Technological Research Council of Turkey under Grant of Support Programme for Scientific and Technological Research Project (Project Code: 106M370).

N. Yucel is with Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering, 06570, Maltepe, Ankara, Turkey (corresponding author to provide phone: +90.312. 231 74 00 ext. 2422; fax: +90.312.231 98 10; e-mail: nuyucel@gazi.edu.tr).

M. I. Berberoglu, S. Karaaslan and N. Dinler are with Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering, 06570, Maltepe, Ankara, Turkey (e-mail: milterx@yahoo.com, karaaslan@gazi.edu.tr, ndinler@gazi.edu.tr).

performance of different safety systems are evaluated and compared.

The smoke movement in a ventilated tunnel fire was simulated through large eddy simulation (LES) by Gao et al [5]. The fire was considered as a volumetric heat source and different scenarios was investigated for different ventilation rates. The results show that the model predicted the flame shape and the smoke backflow accurately.

To analyze the effect of the aspect ratio on smoke movement in tunnel fires, a numerical study by using FDS (Fire Dynamics Simulation) code and experimental study on the scaled version of tunnel were carried out by Lee and Ryou [6]. They found that, the temperatures in the tunnel calculated numerically are close enough to experimental data within 10^{o}C. Aspect ratio of the tunnel cross-section affects the growth and development of smoke in tunnel fires.

The performance of the smoke management system in a typical subway station was studied using FDS version 3.10 by Lin and Chuah [7]. It has been noticed that visibility of smoke is the key factor for escape of the person in an underground station. In addition, proper smoke exhaust system design can maintain a tenable environment for safe evacuation.

A numerical simulation of the flame propagation over horizontal surface of a liquid fuel (heptane) was performed in a model tunnel with a wind velocity by Wang and Joluain [8]. Combustion, soot and radiation models coupled with a LES were tested by win-aided fire propagation behind pyrolysis zone. It was found that for the wind velocity lower than 1.5 m/s, the visible flame can be significantly tilted from horizontal and was elongated over the ceiling surface, the flame length was found to be 8-9 times the pyrolysis length due to lack of oxygen. As the wind velocity increases to 2 m/s, the flame length is only 4 times the pyrolysis length, because of the sufficient oxygen supply in the reacting zone and development of counter rotating vortex structure on the cross-section of the tunnel.

Roh et al. [9] carried out an experimental and numerical study to investigate the effect of ventilation velocity on the burning rate. The experiments are performed on a scaled model with n-heptane. They observed that the burning rate of n-heptane fuel increases as the ventilation speed increases because the supply oxygen effect is larger than the cooling effect. They obtained that the non-dimensional critical velocity is proportional to the one third power of the non-dimensional heat release rate.

A study on the critical ventilation velocity in longitudinally ventilated tunnels was conducted by Hwang and Edwards [10]. The simulation results were compared with available experimental data and simple theories. The CFD results show that the fuel type and ambient temperature have negligible effects on the critical ventilation velocity. The fire burning rate increases by leveling of the critical ventilation velocity.

The objective of this study is to investigate fire behaviors, experimentally and numerically, in a scaled version of underground station. Fire experiments are simulated by burning 10 ml isopropyl alcohol fuel in a fire pool with dimensions 5cm x 10cm, at the center of 1/100 scaled underground station. Experiments are done for zero piston effect (natural ventilation) and different uniform inlet air velocities in order to represent train operations called *‘piston effects’* (forced ventilation). During the experiments, temperature and velocities at different locations of the scaled version of underground station are measured and recorded. Finally, flow field and combustion models are constructed with suitable boundary conditions for numerical simulations. The numerical simulation results are compared with experimental data obtained in this study.

II. PROBLEM DEFINITON AND EXPERIMENTAL SETUP

The photograph and schematic half-model of underground station is shown in Fig. 1 and Fig. 2, respectively. The model is scaled 1/100 and constructed with heat resistant materials. The symmetry plane is closed by a heat resistant glass which permits visual observation and recording the experiments. In order to simulate fire under piston effects a variable velocity blow off fan is placed at the inlet of station. To satisfy uniform inlet velocity a honeycomb mesh is mounted after the fan.

During the fire simulation experiments, the temperatures inside the model station are measured at three different vertical planes. In each plane, temperatures are measured and recorded from 9 locations and in the exit tunnel continuously. In addition to that, flow velocities are measured and recorded at the inlet, center and outlet part of the model station. The locations of the thermocouples and pitot tubes are shown in Fig. 3. For each fire tests, 10 ml isopropyl alcohol is burned in a 5 cm x 10 cm x 4mm pool.

III. CRITICAL VELOCITY

The critical velocity is defined as the minimum longitudinal ventilation velocity needed to avoid the upstream smoke flow (backlayering i.e. the smoke moving in the opposite direction of the ventilation system). The relationship of the critical velocity is a function of fire heat release rate. The magnitude of critical velocity varies with the tunnel cross-sectional geometry. Wu and Bakar [2] suggested that the mean hydraulic tunnel height should be used as the characteristic length in the buoyancy force expression, instead of the tunnel height. The Froude number is defined as the ratio between the buoyancy forces generated by the fire and the inertia forces due to ventilation air flow,

$$Fr = \frac{V^2}{gD_h} = \frac{inertia\, forces}{gravity\, forces}$$

The Froude number preservation in fire situation was proposed by Thomas [11]. He suggested that Richardson number is close to unity at critical condition. Richardson number is defined as

$$Ri = \frac{gD_h}{V^2}\frac{\Delta\rho}{\rho} = \frac{1}{Fr}\frac{\Delta\rho}{\rho}$$

The critical ventilation speed is $V_c = \sqrt{\frac{\Delta\rho}{\rho} gD_h}$.

Fig. 1 Photograph of model underground station

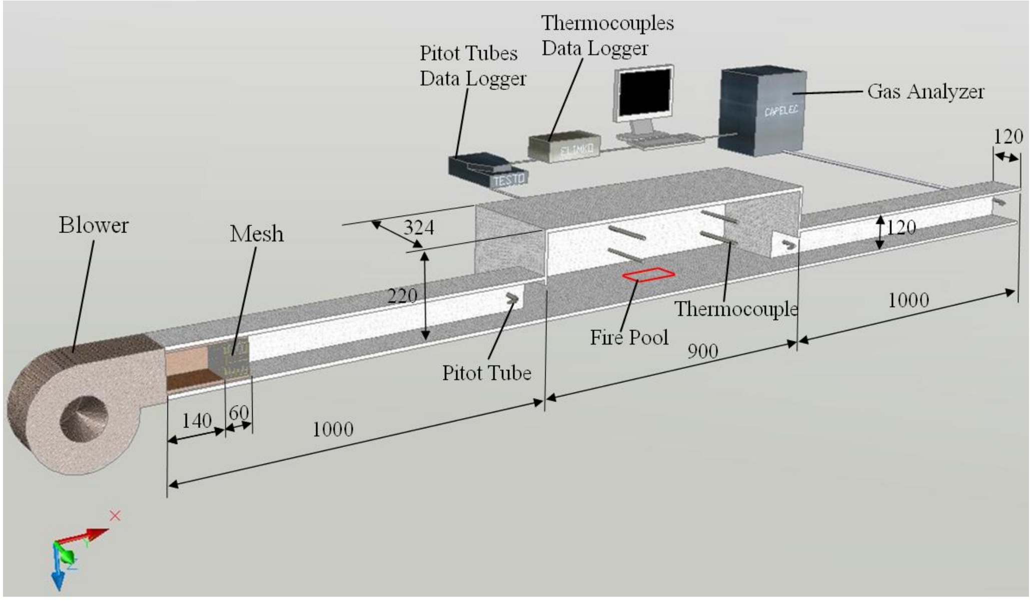


Fig. 2 Schematic diagram of model underground station

Thomas [11] expressed that the critical ventilation as a function of the Heat Release Rate (HRR) by:

$$V_c \approx \left(\frac{g D_h Q}{\rho_o T_o C_p A} \right)^{1/3}$$

Where D_h (m) mean hydraulic diameter of tunnel, Q (W) is heat release rate, A (m^2) is the station cross section area ρ_o (kg/m^3) and T_o (^{o}C) are ambient air density and temperature, respectively. C_p (J/kgK) is the specific heat capacity of air.

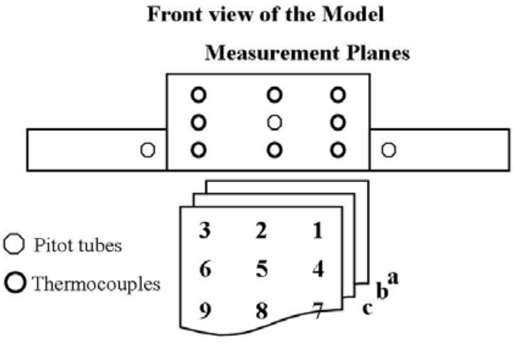


Fig. 3 Locations of thermocouples and pitot tubes

In this study, D_h=0.262m, ρ_0=1.225 kg/m^3, T_0= 21 ^{o}C, C_p=1006 J/kgK, Q=2000 W. The critical velocity in the station is calculated V_c=1.41 m/s for Q=2000 W and V_c=1.47 m/s for Q=2300 W. The ratio of cross sectional areas of station and tunnel is 4.95. The tunnel inlet velocities are chosen as 1 m/s and 3 m/s for this study. Therefore, the corresponding average air velocities in the station are 0.20 m/s and 0.60 m/s, respectively. In the station, the chosen mean air velocities are less than the critical velocities.

IV. EXPERIMENTAL RESULTS AND DISCUSSIONS

In order to examine piston effect to the in-station flow characteristics and fire behavior, the different uniform velocities (natural ventilation, 1 m/s and 3 m/s) were applied to the inlet of the station entrance tunnel. 10 cm x 5 cm and 4 mm high pool with 10ml isopropyl alcohol is placed at the center of station. The experimental measurements of time history of temperature distribution with different location of the model station are shown for 0, 1 and 3 m/s ventilation velocity, in Fig. 4, 5 and 6, respectively. It is seen that overall temperature is highest with natural ventilation situation. However, when the ventilation velocity is increased from 1 m/s to 3 m/s the maximum temperature in the station is found to be less for ventilation velocity of 1 m/s. The burning rate of isopropyl alcohol increases as the ventilation velocity is increased. The reason for these experimental result lies on the relative dominance of oxygen supply effect on cooling effect. Without piston effect, maximum temperature occurs above the fuel pool, however, when the ventilation velocity increased the flame was tilted in the direction of ventilation and the location of maximum temperature moves along the flow direction. The maximum temperature in the station is around 500 ^{o}C for natural ventilation case, however it decreases to 300 ^{o}C for 1m/s ventilation velocity case and 350 ^{o}C for 3 m/s ventilation velocity case. In Fig. 4, three peak temperatures are observed and those temperatures are measured by thermocouples (at locations 8b, 5b and 2b) above the isopropyl alcohol pool. In forced ventilation cases, the peak temperature is measured at location 8b. Along the floor of the station from the fire source, temperatures are higher than those of natural ventilation case.

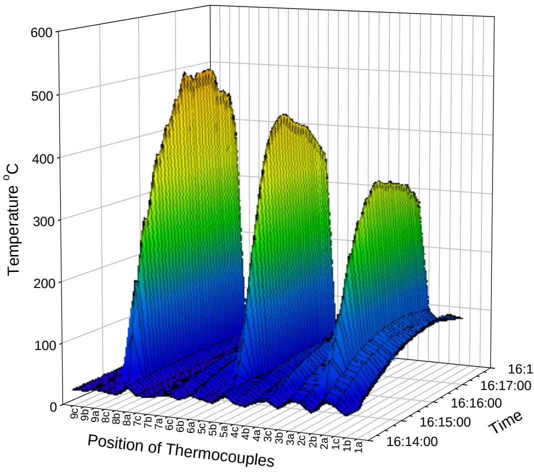


Fig. 4 Time history of temperature distribution on the model station in case of natural ventilation

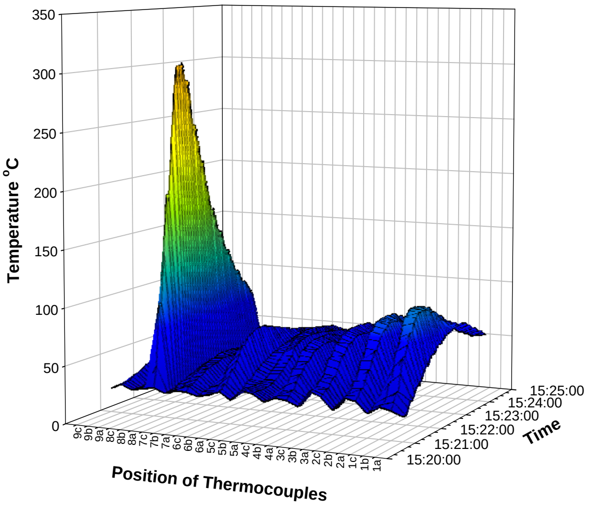


Fig. 5 Time history of temperature distribution on the model station in case of 1 m/s inlet ventilation velocity

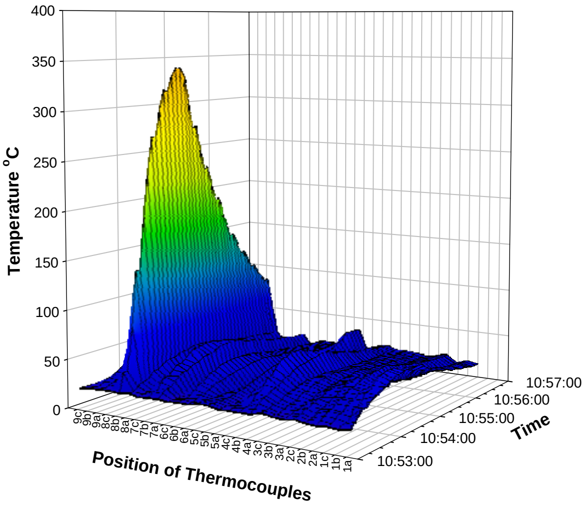


Fig. 6 Time history of temperature distribution on the model station in case of 3 m/s inlet ventilation velocity

V. NUMERICAL SIMULATION RESULTS AND DISCUSSIONS

Commercial FLUENT CFD program was used in numerical simulations. The solid model of the small scaled underground station is generated. In order to get proper numerical grid spacing, grid independency tests were carried out for uniform 1 m/s and 3 m/s inlet velocities and without fire case. Based on the results of grid sensitive test, the optimal mesh of 230.000 tetrahedral cells was used in numerical simulations. At least 1000 iterations were used to obtain converged solutions. Different turbulence models were tested and it was found that k-ω SST turbulence model result matches the experimental results quite well. Then different combustion models which are available in FLUENT program were tested and non-premixed combustion was chosen for fire simulations. It was assumed air uniformly enters the station with 21^{o}C. Atmospheric pressure boundary condition was applied to the outlet of the station. Thermally isolated no slip boundary condition was applied to the solid surfaces.

The CFD-predicted temperature distribution at the station center plane is plotted for inlet air velocity 1 m/s and 3 m/s for the heat release rates of 2 kW and 2.3 kW in Fig. 7 and 8, respectively. In these cases the average velocity in the station are 0.20 m/s and 0.6 m/s. The calculated critical ventilation velocities are 1.41 m/s and 1.47 m/s for considered station model. The predicted temperature profiles are reasonable in most of the upstream and downstream of the flow zone. However, all over the station, the CFD simulations predicted higher temperatures. In experimental study, the maximum temperature measured near the flame zone is about 300^{o}C for 1 m/s and 350^{o}C for 3 m/s. However, the CFD predicted maximum temperature is about 700^{o}C for both cases which was too high for isopropyl alcohol combustion. One of the reasons is that, the turbulent combustion models are based on fast chemistry concepts which tend to overestimate the reaction rates. The other reason, the walls of the station behaves like heat sink and some of the heat is transferred from the heat resistant observation glass. In order to get better estimation for the time history of temperature distribution, it is required to make a good estimation on the heat release rate. In addition to that, heat transferred for the observation glass and heat absorption by the walls of channel must be taken into account.

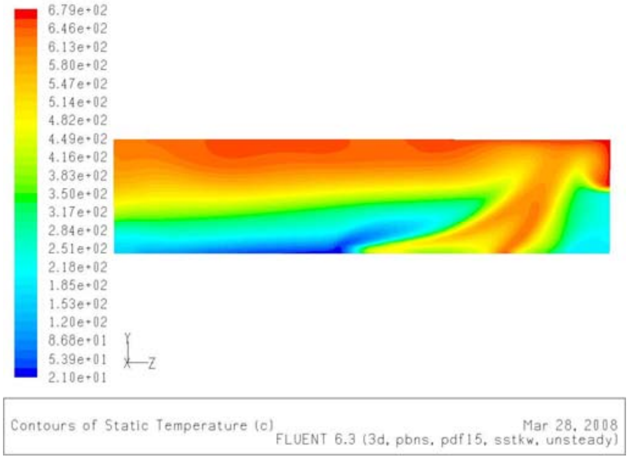


Fig. 7 CFD predicted temperature distribution at time t=13 s of combustion on the centerline plane of fuel source (Q=2 kW, V_c=1.41 m/s, V_{inlet}=1 m/s)

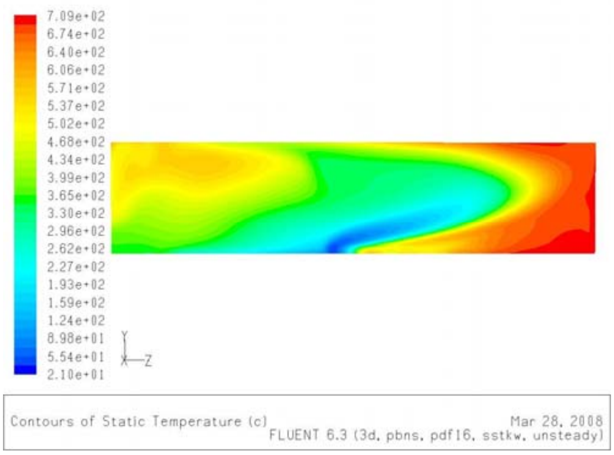
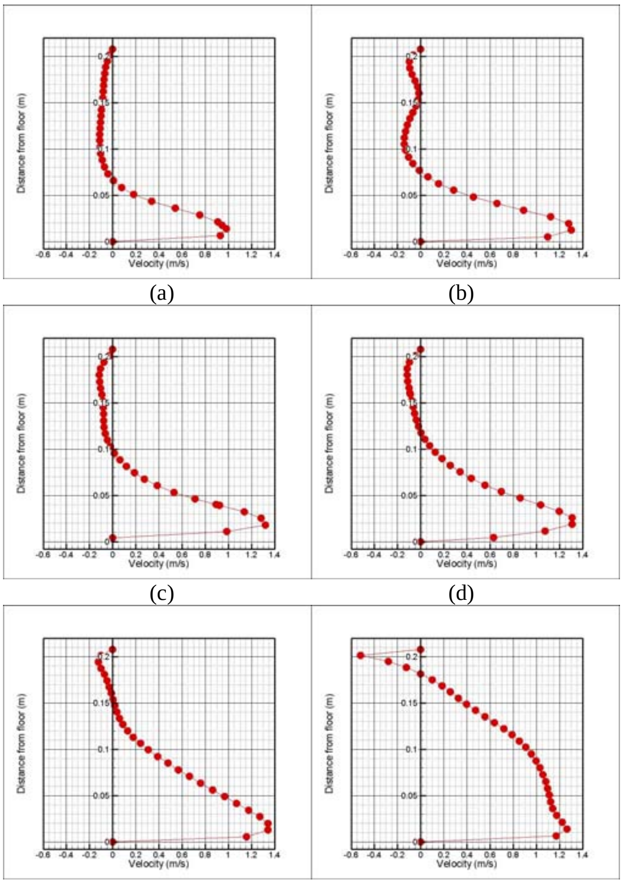


Fig. 8 CFD predicted temperature distribution at time t=13 s of combustion on the centerline plane of fuel source (Q=2.3 kW, V_c=1.47 m/s, V_{inlet}=3 m/s)



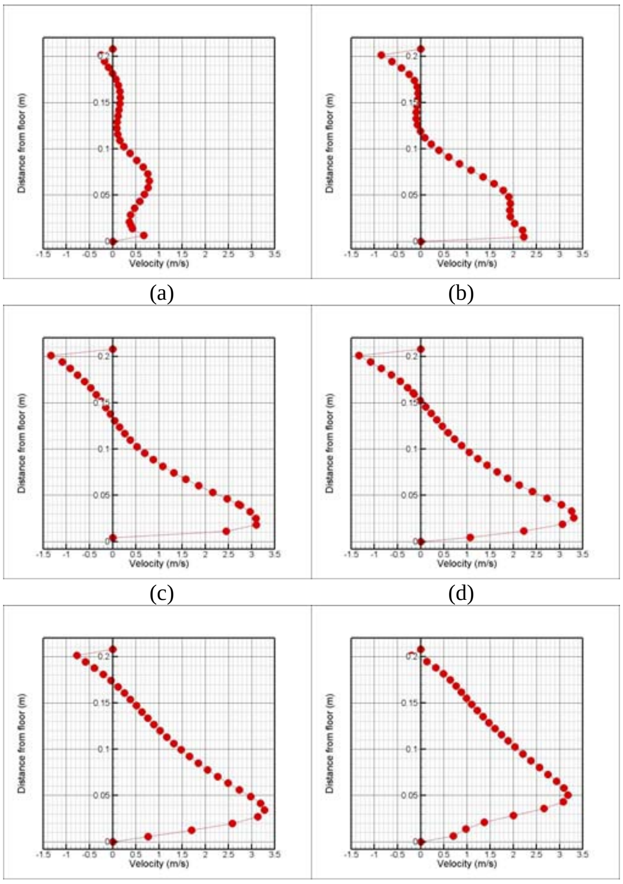
(a) (b)

(c) (d)

(e) (f)

Fig. 9 CFD predicted velocity profiles on the centerline of fuel source at t=13 s for various locations in the station (Q=2 kW V_c=1.41 m/s, V_{inlet}=1 m/s) a) distance from station entrance is 10 cm, b) distance from station entrance is 25 cm, c) distance from station entrance is 40 cm, d) distance from station entrance is 50 cm, e) distance from station entrance is 60 cm, f) distance from station entrance is 70 cm

The velocity profiles for 1 m/s and 3 m/s on the centerline of fuel source at various locations along the station are shown in Fig. 9 and 10. The velocities measured experimentally in the station at different locations are well matched by the CFD simulation results. The prediction of general flow pattern is satisfactory with the smoke visualization tests. The backlayering in velocity is well predicted by CFD simulation.



(a) (b)

(c) (d)

(e) (f)

Fig. 10 CFD predicted velocity profiles on the centerline of fuel source at t=13 s for various locations in the station (Q=2.3 kW V_c=1.47 m/s, V_{inlet}=3 m/s) a) distance from station entrance is 10 cm, b) distance from station entrance is 25 cm, c) distance from station entrance is 40 cm, d) distance from station entrance is 50 cm, e) distance from station entrance is 60 cm, f) distance from station entrance is 70 cm

VI. CONCLUSION

In this study, the fire simulations have been carried out in a reduced-scale underground station model, experimentally and numerically. The effect of ventilation velocity on the time history of temperature distribution is examined. In the experimental study, fire is simulated by burning isopropyl alcohol in a fire pool. In the numerical simulations, k-ω SST turbulence model and non-premixed combustion model are used. It is noticed that, the temperature distribution in the station is the highest without piston effect. Also the lowest temperature is measured in the exit tunnel for the no piston effect fire case. By increasing the ventilation velocity, the fire tilted along the flow direction and point of maximum temperature moves through the tunnel exit and becomes closer to the floor of the station. The burning rate of isopropyl alcohol increases as the ventilation velocity of isopropyl alcohol increased. k-ω SST turbulence model predicts quite well for the flow field simulation. The more adequate combustion model is required for simulating a fire in an underground station

REFERENCES

[1] W.H. Park, D. H. Kim, H. C. Chang, “Numerical predictions of smoke movement in a subway station under ventilation,” *Safety in the Underground Space - Proceedings of the ITA-AITES 2006 World Tunnel Congress and 32nd ITA General Assembly*, 2006.

[2] Y. Wu, M. Z. A. Bakar, “Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of critical velocity,” *Fire Safety Journal*, vol. 35, pp. 363-390, 2000.

[3] H. Xue, J. C. Ho, Y. M. Cheng, “Comparison of different combustion models in enclosure fire simulation,” *Fire Safety Journal*, vol. 36, pp. 37-54, 2001.

[4] J. S. M. Li, W. K. Chow, “Numerical studies on performance evaluation of tunnel ventilation safety systems,” *Tunneling and Underground Space Technology*, vol. 18, pp. 435-452, 2003.

[5] P. Z. Gao, S. L. Liu, W. K. Chow, N. K. Fong, “Large eddy simulations for studying tunnel smoke ventilation,” *Tunneling and Underground Space Technology*, vol. 19, pp. 577-586, 2004.

[6] S. R. Lee, H. S. Ryou, “A numerical study on smoke movement in longitudinal ventilation tunnel fires for different aspect ratio,” *Building and Environment*, vol. 41, pp. 719-725, 2006.

[7] C. J. Lin, Y. K. Chuah, “Smoke management design and computer simulation of an underground mass transit station in Taiwan,” www.ncree.gov.tw/2004tcworkshop/pdf/14.pdf, 2004.

[8] H. Y. Wang, P. Joulain, “Numerical simulation of wind-aided flame propagation over horizontal surface of liquid fuel in a model tunnel,” *Journal of Loss Prevention in the Process Industries*, vol. 20, issues 4-6, pp. 541-550, 2007.

[9] J. S. Roh, H. S. Ryou, D. H. Kim, W. S. Jung, Y. J. Jang, “Critical velocity and burning rate in pool fire during longitudinal ventilation,” *Tunneling and Underground Space Technology*, vol. 22, pp. 262-271, 2007.

[10] C. C. Hwang, J. C. Edwards, “The critical ventilation velocity in tunnel fires – a computer simulation,” *Fire Safety Journal*, vol. 40, pp. 213-244, 2005.

[11] P. H. Thomas, “The movement of smoke in horizontal passages against an air flow,” Fire Research Station Note No.723*, Fire Research Station UK*, September, 1968.

Nuri Yucel was graduated from Istanbul Technical University, Mechanical Engineering Department in 1979, in Turkey. After graduation he worked as a maintenance engineer in Turkish Petroleum Corporation for 2 and half year. He received his M.Sc. degree from Polytechnic Institute of New York in U.S.A, in 1984, and Ph.D. degree in Polytechnic University of New York in U.S.A, in 1988. Yucel is a member of Turkish Chamber of Mechanical Engineers and Society of Turkish Heat Science and Technique. He is the editor – in - chief of the Journal of Turkish Heat Science and Technique. The major field of study of Yucel is numerical and experimental studies in heat transfer and fluid flow in enclosures and channels and internal combustion engines.

He worked as an assistant professor between 1988 and 1991 at Anadolu University in Eskisehir, Turkey. He has been working at Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering since 1991. He has many conference and journal papers.