CFD ANALYSIS OF TUNNEL FIRES

A THESIS

Submitted in partial fulfilment of the
requirements for the award of the degree
of
DOCTOR OF PHILOSOPHY
in
CHEMICAL ENGINEERING

by
SHORAB JAIN

DEPARTMENT OF CHEMICAL ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE - 247 667 (INDIA)
JULY, 2009

INDIAN INSTITUTE OF TECHNOLOGY ROORKEE ROORKEE

CANDIDATE'S DECLARATION

I hereby certify that the work which is being presented in the thesis entitled **"CFD ANALYSIS OF TUNNEL FIRES"** in partial fulfilment of the requirement for the award of the Degree of **Doctor of Philosophy** and submitted in the **Department of Chemical Engineering** of the **Indian Institute of Technology Roorkee, Roorkee** is an authentic record of my own work carried out during a period from July, 2002 to July, 2009 under the supervision of **Dr. Surendra Kumar**, Professor, Indian Institute of Technology Roorkee, Roorkee and **Dr. T. P. Sharma**, Scientist G and Head (Retd.), Fire Research Division, C. B. R. I., Roorkee.

The matter presented in this thesis has not been submitted by me for the award of any other degree of this or any other Institute.

Shorabjain

(SHORAB JAIN)

This is to certify that the above statement made by the candidate is correct to the best of our knowledge.

Date: 30.7.2009

(T. P. Sharma)
Supervisor

(Surendra Kumar)
Supervisor

The Ph. D. Viva – Voce examination of **Mr Shorab Jain**, Research Scholar, has been held on 30.4.2010 at 12.15 PM.

Signature of Supervisors

Signature of External Examiner

ABSTRACT

Majority of transport modes use tunnels, whether it is rail, road or water. Millions of people travel through these tunnels everyday. Recent disasters have shown that consequences of fire are very severe in tunnels. Today, the safety of tunnels has acquired international importance. As the number and length of road and rail tunnels are increasing and more people are using them, an enhanced emphasis is being laid on taking justifiably appropriate fire safety measures. The safety obviously calls for better detection and fighting systems, and also efficient ventilation and smoke control systems.

In order, first to understand various traditional safety features of tunnels, a number of important long distance road transport tunnels of Europe have been studied extensively with respect to their technical specifications including fire safety measures. It is found that the tunnels have a wide variety of features. It is also found that most of the countries either do not have or have minimal formal guidelines of fire safety in tunnels. NFPA – 502, 'Standard for Road Tunnels, Bridges, and Other Limited Access Highways' evolved into a code, only in 1998. Earlier it was, at best, merely a recommended practice. In December 2002, the European Commission however proposed a new directive on 'Safety in European Road Tunnels'. The safety features of the studied tunnels are compared with those of NFPA 502 and with the proposed directive of European Commission. It is found that there are substantial differences in some provisions as given in NFPA 502 and proposed directive of European Commission. In order to provide better safety features inside tunnels, the most stringent measures are proposed in the form of general fire safety guidelines. It is suggested that in future tunnels these observations / recommendation should

be complied. The higher safety standards will ensure less fatalities and less damage to tunnel structure in case of fire.

The aforesaid study of safety components shows that ventilation system is one of very important safety measures inside tunnels used for controlling and extracting smoke in case of fire emergency. In long tunnels, where ventilation is provided by mechanical means, two types of ventilation lay outs exist - longitudinal and transverse. The longitudinal ventilation is provided through jet fans located axially below the ceiling or through jet injection system where the fans are located in a fan room and air is supplied through ventilation shafts. In longitudinally ventilated tunnel fires, smoke and hot gases form a layer below the ceiling and flow in the direction opposite to the ventilation stream. This phenomenon is called back layering.

The ventilation velocity just sufficient to prevent back layering of smoke over the stalled vehicles is the minimum velocity needed for smoke control and is known as the critical velocity. The ability of the longitudinal ventilation system to prevent back layering is the current industry standard to measure the adequacy of the system for smoke control. The ventilation velocity depends on number of parameters such as heat release rate (HRR), tunnel geometry, slope etc. This implies that ventilation system has to be designed for each individual tunnel. The ventilation system can though be designed and evaluated through experimental studies of each tunnel, but that is impractical and expensive. Alternative method is to use mathematical modeling which when coupled with flow visualization techniques provides an excellent means to study the environment inside a tunnel. This should help in designing appropriate ventilation system effectively without the need to conduct experiments.

There exist mainly two types of mathematical models – simple zone models and complex three dimensional field, also called CFD models. To assess the capability and utility of simpler and time efficient zone models for predicting thermal environment inside the tunnel in case of a fire, a comparative study of a zone model and a CFD model has been carried out. For this, a fire scenario inside a naturally ventilated tunnel has been simulated using multi room zone model, CFAST and field model, CFX. For simulation, a tunnel of length 150 m having a rectangular cross section of 80 m^2 has been considered. The temperature and velocity profiles generated by fire, placed at a distance of 20 m from one end of portal have been predicted. The simulation by CFAST has been carried out by dividing the tunnel into 1,2,5,8,10,12 and15 compartments of equal size, where these compartments are joined by openings or vents having same cross section as that of the tunnel. In case of tunnel divided into 15 compartments the fire source position lies at the position of vent; CFAST predicted very high temperatures. The simulations have also been carried out by dividing tunnel into unequal sized compartments such that position of fire is at the center of the compartment. It was found that for accuracy of results, location of fire source inside compartment is an important factor. Computational difficulty is experienced when tunnel is divided into more than fifteen compartments. The CFX and CFAST predictions show that smoke temperature changes with a pattern roughly similar to that of heat release rate. The temperature profiles at selected positions cannot be predicted by CFAST unlike CFX. The detailed features like flame tilt, flow field can only be observed from CFX predictions. It is concluded that zone models alone can not be used for studying fires inside tunnels. On the other hand, the CFD models can be a powerful tool in analyzing problems involving far field smoke flow, impact of fixed ventilation flows etc.

Therefore CFD model has been used to evaluate ventilation strategies in a transport tunnel in case of fire emergency. The aim is to study the smoke movement inside tunnels, and determination of critical ventilation velocity for smoke control in longitudinally ventilated tunnels which are similar to tunnel sections of Delhi Metro Rail corridor, India. The tunnel sections considered have different modes of longitudinal ventilation - ventilation through jet fans and through jet injection system. Both these modes of longitudinal ventilation are evaluated. The CFD program, CFX is used to study the effectiveness of smoke ventilation system to control smoke spread in the event of fire inside the tunnel.

For the first study, where axially mounted jet fans located below the ceiling provides necessary ventilation, the tunnel section considered for analysis is 100 m long, 6 m wide and 9 m high. It is assumed that a fire source producing a constant heat release rate of 4 MW is located at the center of the tunnel. The numerical model used is first verified using experimental results available in the literature. The model is then used to simulate the fire environment inside the tunnel. The results of CFD simulations are compared with those of empirical correlations available in the literature. The effectiveness of smoke ventilation system is then studied. For this the effect of ventilation flow rate, both uniform and non-uniform airflow in tunnel, on thermal environment inside the tunnel is studied. The critical velocity necessary to prevent back layering for the two scenarios are determined. A ventilation scenario where both inlet and exhaust fans are activated is also studied.

It is found that under natural ventilation conditions inside a tunnel, the smoke moves symmetrically along the crown in both directions, and cool entrained air from bottom of tunnel portals move towards the fire source. For uniform induced air flow, it is found that CFD predicted higher critical ventilation velocity (2.18 m/s) than predicted by empirical relations developed by Wu and Bakar (1.52 m/s). For non uniform induced air flow the required critical axial fan velocity is found to be much higher and lies between 8 m/s and 10 m/s. It is also found that the stratified layer of smoke in downstream region is not formed, and no escape can take place from the downstream direction.

In the second study, where air is supplied through jet injection system, the section of tunnel considered is 400 m long, 5.5 m wide and 6 m high. The analysis has been carried out by assuming a variable fire source with a peak heat release rate (HRR) of 16MW, located at the center of the tunnel. Ventilation ducts are located in the ceiling near the tunnel portals and inclined at 10^{o} to the plane of the ceiling through which fans discharge air. The influence of the fire HRR curve slope on the smoke flow dynamics in this realistic tunnel model fitted with inclined fans is investigated. The physical models used are same as those used in previous study. In case of fire two scenarios are studied: (i) fans activated immediately and achieve its full speed after detection of fire. (ii) fans activated at delayed times to take into account the response time of the fans to achieve its maximum speed. The velocity of supply and exhaust fans necessary to remove smoke in 30 sec from the upstream direction is determined.

It is found that the smoke moves symmetrically along the crown in both directions and reaches tunnel portals in about 3 min. It is also found that for this type of tunnel configuration higher supply and exhaust velocities are required to produce the desired critical velocity. The velocities of fan required to produce different desired axial velocity inside the tunnel is determined and is represented in the form of a graph. The exhaust fans do not influence the velocity in upstream area but are necessary for smoke removal in the downstream direction. It is also necessary that fans are activated to full speed within three minutes of starting of fire in order for the ventilation system to be effective for desired smoke removal.

ACKNOWLEDGEMENTS

I express my profound sense of gratitude to my supervisor - Dr. Surendra Kumar, Professor, Department of Chemical Engineering, IIT Roorkee, Roorkee, for his constant inspiration, encouragement and guidance throughout the course of this research work. His esteemed guidance with enriched knowledge, regular encouragement and invaluable suggestions toward excellence during the period has proved to be extremely beneficial. Their intelligence, patience, and understanding of the subject made this work possible.

Besides, I take this opportunity to express my gratitude to Dr. T. P. Sharma Scientist G and Head (Retd.), Fire Research Division, Central Building Research Institute, Roorkee for taking personal interest for my career development in the Institute in research and development activities.

I am deeply indebted to Dr. (Mrs.) Shashi, Assistant Professor, Department of Chemical Engineering, IIT Roorkee, Roorkee for her cooperation, encouragement, constant inspiration and guidance.

I express my sincere gratitude to Dr. I. D. Mall, Professor and Head, Department of Chemical Engineering, IIT Roorkee, and former Heads for their whole hearted cooperation, encouragement and support.

I express my esteem respect to honourable Shri V. K. Mathur, Former Director, Central Building Research Institute, Roorkee for permitting to do this work and Dr. M. O. Garg, Director, CBRI, Roorkee for their support and for providing all necessary facilities at my disposal. They have always been a source of inspiration to me.

Thanks are due to my colleagues Shri R. S. Chimote and Shri Nadeem Ahmed for their help and support in doing this research work. I also wish to convey my thanks to Shri Akhilesh Sharma, Shri Raj Kumar, Shri Narendra who helped me in doing work at the CAD centre, Department of Chemical Engineering, IIT Roorkee.

On completion of the present work, I can't restrict my self from expressing my emotional feeling for my parents, Shri Satish Chandra Jain and Smt Krishna Jain whose blessings made this work possible. I am grateful to my elder sister, Mrs. Malini Agarwal, and my bother-in law, Mr. Manoj Agarwal for their moral support and inspiration for higher education. I also express affection for young ones of the family - Miss Sneha, Master Dhruv. All have missed me during the period.

Last but not the least; I am sincerely thankful to my wife Mrs. Samiksha Jain. Her patience, understanding and constant cooperation and encouragement especially on the frustrated moments during difficult stages of the work, helped me greatly in completing the thesis. My children, Master Rishank Jain and Miss Suditi Jain deserve compensation for the loss of attention to them.

Lastly, my deepest gratitude is due to the All-pervading, All-sustaining Spirit Whose Divine Light and Warmth provided me the perseverance, guidance, inspiration, faith and strength to carry on carrying on even when the going got tough.

Acknowledge Him in all Thy ways and He shall direct Thy paths.

SHORAB JAIN

CONTENTS

LIST OF FIGURES

Note: Figures are numbered on the basis of the chapter where they are referred to.

LIST OF TABLES

Note: Tables are numbered on the basis of the chapter where they are referred to.

NOMENCLATURE

Chapter II

A	Tunnel cross-sectional area (m^2)
A_a	Tunnel annular area (m^2)
A_b	Tunnel blockage area (m^2)
C	Constant (-)
C_p	Specific heat at constant pressure ($kJkg^{-1}K^{-1}$)
D	Characteristic values of length (m)
D_i	Diffusivity coefficient (m^2s^{-1})
Fr	Froude number (-)
Fr_m	Modified Froude number (-)
g	Gravitational force (ms^{-2})
H	Tunnel height (m)
h	Enthalpy (Kgm^2s^{-2})
H_D	Tunnel hydraulic diameter (m)
K	Time constant (s^{-1})
K	Constant (-)
K_g	Tunnel grade correction factor (-)
L	Backed-up layer length (m)
n	Parameter (-)
p	Pressure of gas ($kgm^{-1}s^{-2}$)
$\dot{q}_j^R$	Heat flux due to thermal radiation ($Jm^{-2}s^{-1}$)
Q^*	Dimensionless fire heat release rate (-)
Q'	Convective heat release rate of fire per unit width of the tunnel
Q_c	Convective heat release rate of fire (KW)
Q_{max}	Peak Heat release rate of fire (kW)
R	Gas constant 8.314×10^3 (kgm^2s^{-2} kg-mole^{-1} K^{-1})
r	Parameter (-)
Ri	Richardson number (-)
S_α	Source term in chemical species conservation term ($kgm^{-3}s^{-1}$)
T	Hot layer temperature (K)

t	Time to achieve HRR (s)
T_0	Ambient air temperature (K)
t_d	Time at which decay of fire ends (s)
t_D	Time at which decay of fire starts (s)
t_{max}	Time to achieve maximum heat release rate of fire (s)
u_i	Gas velocity in x_i direction (ms^{-1})
V	Characteristic values of velocity (ms^{-1})
V^*_c	Dimensionless critical ventilation velocity (-)
V_c	Critical Ventilation velocity (ms^{-1})
V_c	Critical Ventilation velocity (ms^{-1})
V_{vent}	Longitudinal air velocity (ms^{-1})
W	Tunnel width (m)
x_i	Distance in i_{th} direction measured from origin of coordinate axis (m)
Y_α	Mole fraction of species α (-)

Greek symbols

λ	Thermal conductivity ($kgms^{-3}\ K^{-1}$)
τ_{ij}	Stress tensor ($kgm^{-1}s^{-2}$)
ΔT	Temperature rise above ambient (K)
$\Delta\rho$	Change in density (kgm^{-3})
$\alpha_{D,L}$	Fire decay rate coefficient for design fires represented by linear growth (s^{-1})
$\alpha_{D,q}$	Fire decay rate coefficient for design fires represented by quadratic growth (s^{-1})
$\alpha_{g,L}$	Fire growth rate coefficient for design fires represented by linear growth (kWs^{-1})
$\alpha_{g,q}$	Fire growth rate coefficient for design fires represented by quadratic growth (kWs^{-2})
ρ	Density (kgm^{-3})
ρ_0	Ambient air density (kgm^{-3})
ρB_i	Body force per unit volume in x_i direction ($Kgm^{-2}s^{-2}$)
ω	Inclination of tunnel (%)

Chapter V

C_p	Specific heat ($kJkg^{-1}K^{-1}$)
D_f	Diameter of flame (m)
g	Gravitational force (ms^{-2})
H_D	Tunnel hydraulic diameter (m)
Q	Heat release rate of fire (kW)
Q^*	Dimensionless fire heat release rate (-)
Q_c	Convective heat release rate (kW)
T_0	Ambient air temperature (K)
T_{cp}	Plume centerline temperature (K)
V^*_c	Dimensionless critical ventilation velocity (-)
V_c	Critical ventilation velocity (ms^{-1})
X	coordinate x axis along length of tunnel (m)
Y	coordinate y axis along height of tunnel (m)
Y_0	Height of virtual origin relative to the base of fire source (m)
Y_f	Height of flame (m)
Z	coordinate z axis along width of tunnel (m)

Greek symbols

ρ	Air density (kgm^{-3})
ρ_0	Ambient air density (kgm^{-3})

Chapter VI

D_f	Diameter of flame (m)
H_D	Tunnel hydraulic height (m)
Q	Fire heat release rate (kW)
Q_{max}	Peak Heat release rate (kW)
t	Time to achieve heat release rate (s)
t_D	Time at which decay of fire starts (s)
t_{max}	Time to achieve maximum heat release rate of fire (s)
V_{avg}	Average axial velocity inside the tunnel (ms^{-1})
V_c	Critical ventilation velocity (ms^{-1})
V_{sv}	Fan velocity (vertical component) (ms^{-1})

X	coordinate x axis along length of tunnel (m)
Y	coordinate y axis along height of tunnel (m)
Y_f	Height of flame (m)
Z	coordinate z axis along width of tunnel (m)

Greek symbols

α	Fire growth rate coefficient (kWs^{-2})
β	Fire decay rate coefficient (s^{-1})

CHAPTER - I

INTRODUCTION

Tunnels are being used throughout the world for transport, the need arises either because of geographical topography of the region or because of pressure on land forcing number of activities to be performed underground. Tunnels play an important role in developing new networks and the number of tunnels in the recent past has increased to significant levels. This has been attributed to improved construction technology whereby tunnels have been used as a cost effective engineering solution to traverse mountainous regions or urban areas with minimum local environment impact. Today tunnels longer than 50 km are a reality. The Sei-kan tunnel in Japan and Channel Tunnel in Europe connecting UK with France are examples of tunnel having length more than 50 kms. Table 1.1 lists some of major road tunnels around the world (Lotsberg, 2009) while Table 1.2 lists some of major rail tunnels across the world (Lotsberg, 2009).

1.1 INDIAN CONTEXT

The oldest rapid transit system in India is the Kolkata Metro, Kolkata (www.kolmetro.com). It extends from Dum-Dum near Netaji Subhas Chandra Bose airport to Tollygunj, the busy north south axis of Kolkata over a length of 16.45 km. The Metro Railway Kolkata was constructed progressively from 1972 to 1995. The Konkan railway project (www.konkanralway.com) has 92 tunnels covering a distance of 83 km and passes through four states of Maharashtra, Goa, Karnataka and Kerala. Delhi Metro Rail Project (www.delhimetrorail.com) truly enlightened the citizens with people friendly

Table 1.1 Major Road Tunnels of the World

S.No.	Tunnel	Country	Length (Meter)	Year of opening
1	Laerdal	Norway	24 510	2000
2	Zhongnanshan (2 tubes)	China	18 040	2007
3	St. Gotthard / 2nd Röhre	Switzerland	16 918	1980
4	Arlberg	Austria	13 972	1978
5	Hsuehshan (twin tunnel + service)	Taiwan	12 942	2006
6	Fréjus	France - Italy	12 895	1980
7	Mont-Blanc	France - Italy	11 611	1965
8	Gudvanga	Norway	11 428	1991
9	Folgefonn	Norway	11 150	2001
10	Kan-etsu (south bound tube)	Japan	11 055	1991

Table 1.2 Major Rail Tunnels of the World

S.No.	Tunnel	Country	Length (Meter)	Year of opening
1	Gotthard Base	Switzerland	57 072	2010
2	Sei-kan	Japan	53 850	1988
3	Channel (Eurotunnel)	England - France	50 450	1994
4	Lötschberg	Switzerland	34 577	2007
5	Guadarrama	Spain	28 419	2007
6	Hakkôda	Japan	26 455	2010
7	Iwate-ichinohe	Japan	25 810	2002
8	Lainzer / Wienerwald	Austria	23 844	2015
9	Îyama	Japan	22 225	2013
10	Dai-shimizu	Japan	22 221	1982

Metro Rail Transit System. The first phase of Delhi Metro Rail project comprising of about 13 km of underground rail network was completed in December 2005. Presently the work on Phase II of project is going on. This project truly paved the way for construction of Metros in other parts of the country for example, Bangalore Metro Rail Project (www.en.wikipedia.org) and Chennai Metro Rail Project (www.janes.com). The work on Bangalore Metro Rail Project has already started. The total proposed length of Bangalore Metro Rail Project comprising of both elevated and underground tunnels is 42 km. Delhi Metro Rail Corporation has prepared Detailed Project Report for Chennai Metro Rail Project and it is proposed to have about 14.29 km of underground tunnel network. A metro rail project 25.25 km long is being planned at Kochi (www.mapsofindia.com/kochi/metrorail-project.html). A metro rail project at Hyderabad is proposed. Besides this, metro rail connecting city of Bhopal and Indore is also under consideration.

As the number and length of the tunnels are increasing and more people are using them, fire incidents inside them are also increasing. Fire inside tunnels behave differently than fires in open area. First, in a tunnel more heat is fed back to the burning vehicles because of enclosed space resulting in increased heat release rate. For example Carvel et al. (2001) concluded that heat release rate of a fire within a tunnel could increase by a factor of 4 compared to that of same material burning in the open. Secondly, the fire inside tunnel interacts with the ventilation airflow and generates aerodynamic disturbance in tunnel flow. Because of this the smoke and toxic gases are carried far away from the fire source and thus poses difficulty in fire fighting and rescue operations.

In case of fire inside the tunnel, the hot gases and smoke moves upward from fire source towards ceiling. After touching the ceiling, they can not move further upward and therefore travel horizontally along the ceiling. The smoke and gases produced from fire are hot and lighter than air, they therefore form a hot layer at the ceiling. The smoke stratifies (forms a discrete layer under the tunnel ceiling) and moves longitudinally away from the fire in both the directions. The smoke progresses and its degree of stratification depend on the airflow in the tunnel. This stratification breaks when smoke is cooled because of convective heat loss to the walls and mixing of smoke with fresh cool air at the bottom of tunnel. Thus, from the beginning of a fire inside the tunnel, the airflow inside the tunnel is modified and becomes highly transient.

1.2 STATISTICAL OVERVIEW OF REAL FIRES

According to statistics from late 1980's the probability of a fire accident in traffic tunnels can be assumed as follows: 1 case per 10 x 10^9 km regarding road traffic and 1 case per 0.5 x 10^9 km regarding railway traffic. With increasing traffic over the years, risk of fire in traffic tunnels also increases. According to French statistics (Peard, 1996), there will be about one or two car fires (per kilometer of tunnel) for every hundred million cars and eight fires for heavy goods vehicle that passes through the tunnel. A recent report by S. Kumar (as cited in Haack, 2007) summarizes fire incidents (road, rail) in UK between 1994 and1999. The report states that the total number of arson fires (all vehicle types) in tunnels varies in UK between 60-90 per year. This would be about 0.1 % of all vehicle fires, about 0.3 to 0.45 % of the vehicle fires with engines running and about 6.7 % to 10 % of the vehicle fires which follow a crash / collision.

A list of few Major Tunnel Fires that occurred in recent past and corresponding fatalities in them is given in Table 1.3. The full details of these fires can be found in the Handbook of Tunnel Fire Safety (Carvel & Marlair, 2005). Though it is concluded that fires inside tunnels are rare, yet fires inside a tunnel have catastrophic effect especially in terms of life safety. The Handbook contains a comprehensive list detailing 160 fires dating back to 1842. The consequences of fire inside tunnels can be extremely destructive and dangerous because the enclosed space hinders the dissipation of heat and smoke. Smoke from a fire reduces the visibility and cause slower evacuation. The toxic gases in the smoke can prove to be fatal. In addition, access limitations for fire fighting & rescue operation, difficulty in ensuring safe escape route to tunnel users from an enclosed space increases severity of incidents. Fires in tunnels not only endanger the lives of tunnel users, they also cause damage to vehicles and tunnel structure with very prejudicial consequences on the capital represented by the tunnel. They often reduce the availability of the tunnel to traffic, at least for a certain length of time. Due to increasing traffic any incidence resulting in closure of tunnel for certain period of time disrupts the normal life and creates panic. Today fires in traffic tunnels have acquired international importance. In spite of grave consequences, a fire can have inside tunnel; most of the countries either do not have or have minimal guidelines for fire safety inside transport tunnels.

1.3 CAUSES OF FIRES

The technical report - Design Fire Scenarios by Alfred Haack (2007) under the aegis of The European Thematic Network FIT 'Fire in Tunnels' analyzed the causes of fires and problems associated with fire fighting operations inside tunnels. They included fires from various countries (e.g. USA,

Table 1.3 Major Tunnel Fires in Recent Past

Tunnel	**Country**	**Date**	**Type**	**Death**	**Injuries**	**Vehicles**
2003						
Guadarrama	Spain	6 Aug 2003	Rail	0	0	Train
Jungango	S. Korea	18 Feb 2003	Metro	189	Many	2 Trains
2002						
Homer	New Zealand	3 Nov 2002	Road	0	3	1 Bus
A86 Motorway	France	5 March 2002	Road	0	2	Train
Tauern	Austria	18 Jan 2002	Road	0	0	1 HGV
2001						
St. Gotthard	Switzerland	24 Oct 2001	Road	11	Some	23 vehicles
Gleinalm	Austria	3 Sept 2001	Road	0	0	Bus
Gleinalm	Austria	7 Aug 2001	Road	5	4	2 cars
Gleinalm	Austria	29 July 2001	Road	0	0	Bus
Howard St.	USA	18 July 2001	Rail	0	0	Train
Schipol Airport	Netherlands	11 July 2001	Rail	0	0	Electrical
Tauern	Austria	10 July 2001	Road	0	0	2 cars
Prapontin	Italy	28 May 2001	Road	0	14	HGV
2000						
Toronto Subway	Canada	8 Dec 2000	Metro	0	3	Train
Laerdal	Norway	27 Nov 2000	Road	0	0	Bus
Kitzsteinhorn	Austria	11 Nov 2000	Funicular	155	Some	Train
Saukopftunnel	Germany	24 Aug 2000	Road	0	0	Car
NYC Subway	USA	2 Aug 2000	Metro	0	0	Electrical
Seljestad	Norway	14 July 2000	Road	0	20	8 vehicles
Cross-harbour	Hong Kong	29 May 2000	Road	0	0	Car
Oslofjord	Norway	2000	Road	0	0	HGV
Montreal Subway	Canada	15 April 2000	Metro	0	0	Electrical
Toulon Motorway	France	1 Feb 2000	Road	0	0	2 vehicles
Tauern	Austria	10 Jan 2000	Road	0	0	HGV

Japan, Great Britain) that occurred between approximately 1970 and 1997 in the analysis of the fire events. A total of 85 fires were analyzed in underground traffic systems. These selected fires were divided up as follows:

- 45 fires in underground railway and suburban railway tunnels
- 11 fires in main-line railway tunnels
- 29 fires in road tunnels.

The study highlighted following points

(1) Main Causes of Fires (Figure 1.1):

Vehicle defects

Arson (metro tunnels),

Rear-end collisions (road tunnels)

(2) Problems with Extinguishing and Rescue Work (Figure 1.2):

Poor visibility for the rescue workers (metro tunnels, road tunnels)

Inadequate two-way radio connections during fire service operations (mainly in metro and road tunnels)

In view of above, it is essential to prevent accidents in tunnels and adequate safety measures should be provided for tunnel users to escape or be rescued by fire brigade in case of emergency. The measures of ensuring safety in tunnels falls under two categories: reduction of probability of an accident & reduction of consequences of events such as fires & accidents. The former consists of aspects like tunnel design, facilities installed in a tunnel such as ventilation systems,

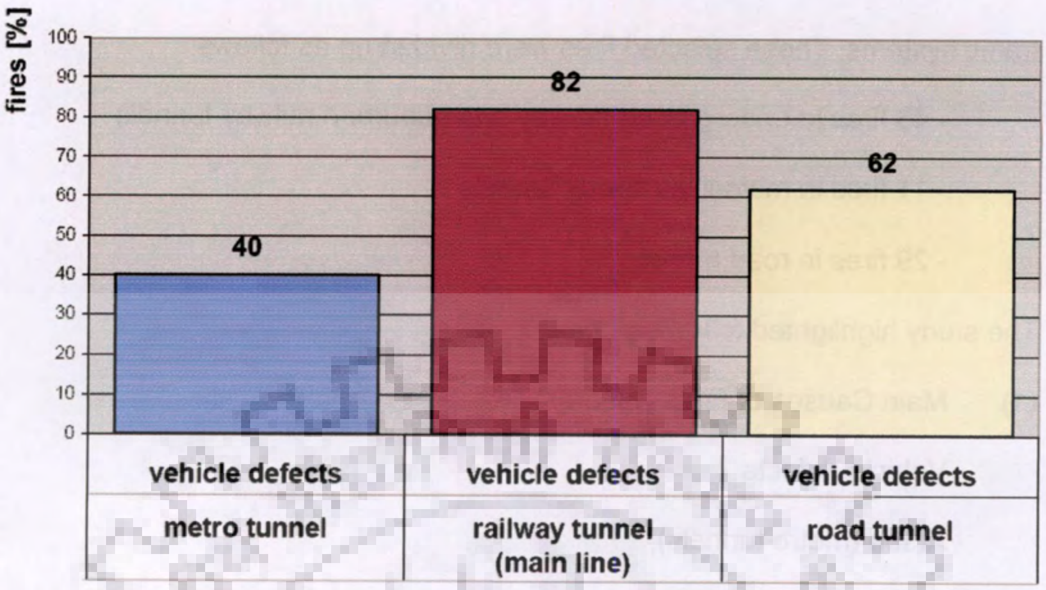


Figure 1.1 Main Causes of the Selected Fires in Traffic Tunnels (Haack, 2007)

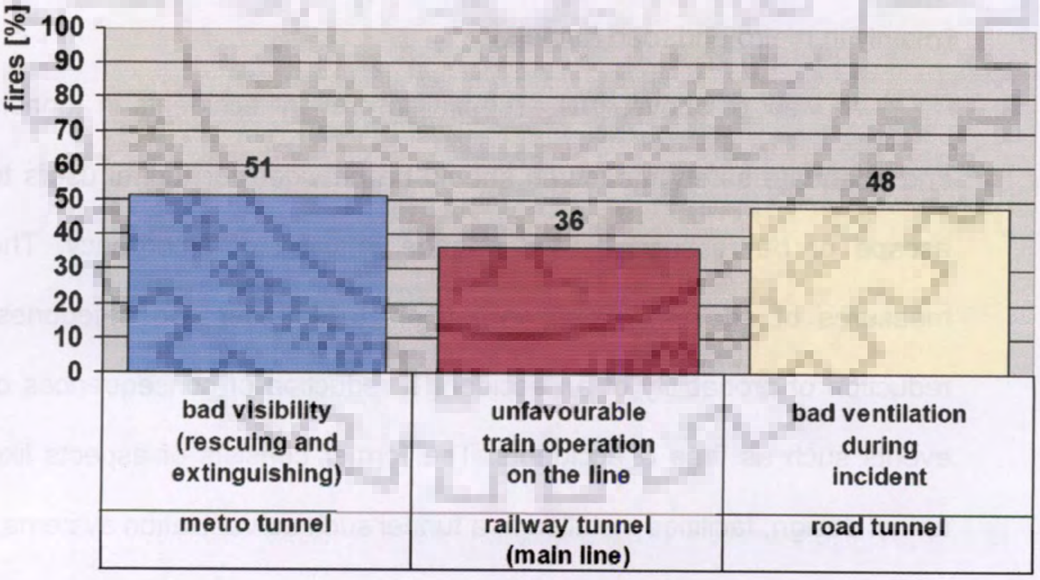


Figure 1.2 Main Problems of Fire-Fighting in the selected Fires in Traffic Tunnels (Haack, 2007)

lightening systems, interior finish, maintenance etc. The latter consists of installing emergency facilities & constructing fire resistant tunnel structures.

The fire safety equipments normally used for extinguishing / controlling fire are fire extinguishers, fire hydrants, sprinklers and ventilation system.

- **Fire Extinguishers**: Fire extinguishers can be used to extinguish a fire at initial stage and its ease of operation makes it important for general use. They are placed at regular intervals at a height for easy operation.
- **Hydrants**: are used by fire fighters for fire fighting operations inside the tunnel. They are installed in all lay-bys in tunnel & near entrance of escape passages apart from both entrances.
- **Sprinkler System**: sprinklers are not used in local tunnels but only at stations. The reason being that water is denser than most hydrocarbon and therefore the hydrocarbons will float on top of water and continue to burn.

Also

(i) Water can cause an explosion in petrol/diesel and other chemical substances if not combined with appropriate additives

(ii) There is a risk that when fire is extinguished, flammable gases may still be produced which may cause an explosion.

(iii) Steam generated would hurt people

(iv) it is difficult to control fire inside vehicles

(v) The smoke layer would be cooled down, lose its stability, and will cover the whole tunnel, thus reducing visibility and subsequent evacuation of people.

- **Ventilation System**: In case of fire inside tunnel, once the smoke fills the whole tunnel it becomes difficult to evacuate passengers without some assistance. For control of smoke, ventilation is used. It is a means by which harmful substances/toxic gases is diluted or removed to prevent harmful substances from injuring the health of tunnel users & maintenance personnel and to maintain good visibility in tunnels. In ventilated tunnel fires, smoke and hot gases may form a layer near the ceiling and flow in the direction opposite to the ventilation stream. The existence of this reverse stratified smoke layer has an important bearing on fire fighting and evacuation operations. Also, ventilation has varied influence on fire size in tunnels. Therefore use of appropriate ventilation is necessary for smoke management.

To study the influence of ventilation on fire size in tunnels and to ascertain the performance of ventilation systems for smoke control, experimental testing of fires in vehicle tunnels has been carried out. Broadly these fire tests are carried out to understand the dynamics of fire inside tunnels. These tests are necessary to assess the hazards arising from tunnel fires and to plan effective mitigation strategies. Reduced-scale experiments are not able to reproduce full-scale features although they provide useful information. Full-scale experiments gives better understanding but are prohibitively expensive. Therefore the mathematical modeling is carried out and a number of predictive models have been developed. These include simple empirical relationships, network models, zone models and complex three-dimensional CFD models. All models have their utility and scope. The simpler models are ideal for repeated applications and are cost effective. On the other hand, three-dimensional CFD simulations are computationally expensive. The objective is to have a well-validated model to study various aspects of tunnel safety design such as ventilation system, smoke management etc.

Zone modeling techniques uses experimentally based empirical expressions to describe various phenomena observed in fire growth and spread. These are one-dimensional models that divide each room into a small number of volumes, such as upper hot layer, the lower cold layer, the fire plume and the compartment boundary. Each of these volumes is assumed to be uniform in its properties. Equations describing the conservation of mass, momentum and energy are solved numerically together with previously developed empirical equations derived from experiments.

In field modeling or CFD Modeling, the computational domain is divided into large number of control volumes and the classical conservation equations of mass, momentum and energy are solved at number of discrete points in time and space (control volume) to determine density, velocity, temperature, pressure and species concentrations . Field or CFD models are based on solving the Reynolds or Favre-averaged Navier Stokes equations using a given turbulence model. Normally a two-equation model like k-ε (Abanto et al., 2007; Fletcher et al.,1994; Jojo et al., 2003; Kumar & Cox, 1985) is used. Recently, Large Eddy Simulation (LES) models have also been used (Gao et al., 2004; Hu et al., 2006). The CFD results can easily be post-processed to visualize areas of concern and can create animation for the training of people to deal with fire emergency. Though this approach has only become possible with the advent of large capacity high-speed modern computers, still CFD simulations are time-consuming.

1.4 OBJECTIVES OF THE THESIS

In the present study, an effort has been made to carry out CFD analysis of tunnel fires and study use of longitudinal ventilation configuration to control and extract smoke from inside the tunnels in case of fire. The tunnel sections considered for analysis are similar to that of Delhi Metro Corridor. The ventilation strategy of both forms of the longitudinal ventilation i.e. jet fans and jet injection system are to be evaluated. The jet fans are located axially below the ceiling and provide both uniform and non uniform induced flow inside tunnel while in jet injection mode; the fans are located in a fan room and provide ventilation through a shaft located in the ceiling. In the tunnel studied, these shafts are inclined at an angle to the plane of the ceiling. The study of smoke movement and determination of critical air velocity to prevent upstream movement of smoke and hot combustion products upstream of fire source for above ventilation scenarios is the objective of thesis. The CFD model used for analysis is CFX developed by AEA Technology (2002).

The main objectives of the thesis are as follows:

(a) Study of fire safety measures in various long distance road tunnels and their compliance with codes and guidelines.

(b) Use of zone model for simulating a fire inside a tunnel and to ascertain its scope and utility vis a vis Field models. A Comparative study of zone model and Field model is to be carried out.

(c) Validation of CFD model using results of experiments published in the literature and evaluation of ventilation strategy of tunnel having jet fans located axially below the ceiling and providing both uniform and non uniform induced air flow.

(d) Evaluation of ventilation strategies of Underground metro tunnel, having jet injection type longitudinal ventilation system where the ventilation shafts are inclined at an angle to the plane of ceiling.

1.5 ORGANIZATION OF THE THESIS

The thesis has been organized into seven chapters. Chapter 1 gives a brief introduction of tunnel fire problems and objectives of the thesis. Chapter 2 gives a detailed review of available literature on smoke ventilation in transport tunnels. This chapter is divided into different sections. Section 1 discusses the principles of ventilation for smoke control. Section 2 reviews literature on full scale, reduced scale and laboratory scale experimental studies conducted on longitudinally ventilated tunnel fires while studies conducted using mathematical modeling is given in section 3. The models reviewed varied from one dimensional empirical model to three dimensional complex CFD models. The use of CFD tool in modeling fires in tunnels is discussed in section 4 of this chapter. Section 5 discusses the use of design fires and design fire scenarios used for representing fire source while modeling of fires in tunnels. Chapter 3 provides a comparative study of Fire Safety Measures in existing long distance road transport tunnels. These measures have been compared with guidelines of existing codes. Based on this, general fire safety guidelines proposed for road tunnels are presented. Chapter 4 gives a comparative study of zone modeling technique using CFAST and field modeling technique using CFX for simulation of a fire inside a naturally ventilated tunnel. In Chapter 5 ventilation strategy of Underground Metro Rail Transport System having jet fans installed axially below the ceiling and providing both uniform and non uniform induced flow have been evaluated using CFD tool. The CFD Model is first validated using results of Steckler Experiment. Chapter 6 evaluates ventilation strategy of Underground Metro Rail Transport System fitted with jet injection system to provide longitudinal ventilation. The Underground Metro Rail Transport System used is similar to a section of Metro in Delhi. Chapter 7 highlights main conclusions of the thesis and recommends directions for future use.

CHAPTER - II

LITERATURE REVIEW

2.0 INTRODUCTION

In the present chapter, efforts have been made to review the literature pertaining to research on smoke modeling resulting from tunnel fires. Broadly, this research includes smoke propagation, its characteristics and its control. The characteristics of smoke propagation include phenomenon like flame development, propagation and sources of heat. These includes effects of obstacles on flame propagation, fire spread between vehicles, measurement of heat release rate (HRR) of burning vehicles inside tunnels, effect of tunnel ventilation and geometry on HRR etc. Smoke control systems require determination of critical ventilation velocity to prevent upstream movement of smoke. For this, it is necessary to understand the behavior of smoke and temperature arising from a fire on a burning train / vehicle inside a tunnel. The behavior of smoke is modified by several factors, including ventilation, fire size (HRR), tunnel inclination and influence of the traffic. Ventilation may be provided by natural means, by the traffic-induced piston, or by mechanical equipment. The mechanical ventilation layouts can be longitudinal or transverse or a combination of these. This chapter reviews both experimental and modeling studies carried out to study behavior of fires inside tunnels, smoke movement and determination of critical ventilation velocity for smoke control in longitudinally ventilated tunnel. Though the focus is on mathematical modeling, yet experimental studies have also been reviewed because these provide necessary insight into the dynamics of fire inside tunnels and have also been used for validating mathematical models. Although every possible effort has been made to include most of the important references, we do not pretend to be exhaustive in its writing. This chapter has been divided into following sections:

Section 1	describes the design principles for smoke control through use of tunnel ventilation system. Different ventilation configurations are discussed.
Section 2	describes in brief the major fire experiments carried out, at large, reduced and laboratory scale.
Section 3	gives state of art on numerical modeling of tunnel fires, more specifically; it describes the simulation of fires in longitudinally ventilated tunnels using empirical models, network models, phenomenological models (zone models / integral models) and field models.
Section 4	describes in brief the basics of CFD techniques and outlines various physical sub models used in fire safety engineering.
Section 5	describes the use of design fire, for representing fire source, in modeling fires inside transport tunnels.

2.1 FIRE & SMOKE CONTROL (Tunnel Ventilation)

Fires behave differently when it occurs inside tunnels. In case of fire inside a tunnel the hot gases and smoke moves from fire towards ceiling where it stratifies (forms a discrete layer under the tunnel ceiling) and moves longitudinally away from the fire in both the directions. The smoke progress and its degree of stratification depend on the airflow in the tunnel. This phenomenon is shown in Figure 2.1. Once the tunnel is full of smoke, it is difficult to evacuate passengers without some assistance. Therefore smoke

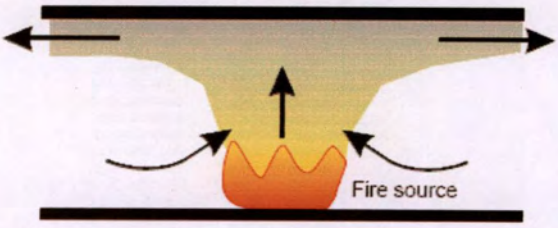
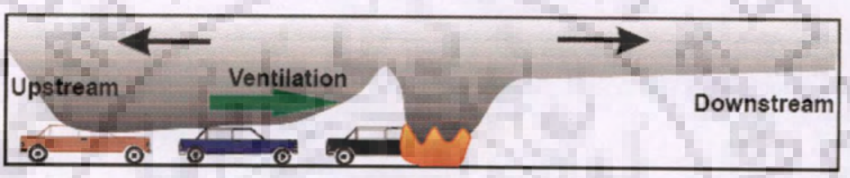


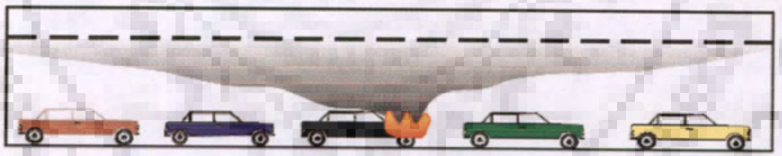
Figure 2.1 Smoke Progress in a naturally ventilated tunnel (Kashef et al., 2003)

(a) Favorable conditions

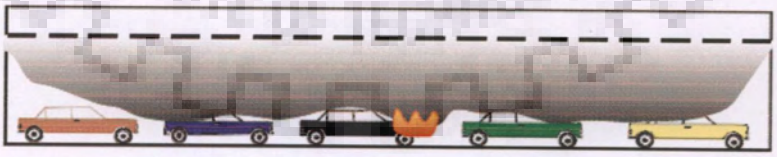


(b) Unfavorable conditions

Figure 2.2 Smoke spread in a unidirectional traffic tunnel (Kashef et al., 2003)



(a) Favorable conditions



(b) Unfavorable conditions

Figure 2.3 Smoke spread in a bidirectional traffic tunnel (Kashef et al., 2003)

control measure is necessary for fire fighting and rescue operations. For control of smoke, ventilation is used. It is a means by which harmful substances / toxic gases is diluted or removed to prevent harmful substances from injuring the health of tunnel users & maintenance personnel and to maintain good visibility in tunnels. Thus, during a fire emergency, the ventilation systems are primarily used to clear tunnel of smoke by removing smoke and hot gases, to assist in the evacuation and rescue of motorists and allow safe access for firefighters inside the tunnel (NFPA 502, 2001).

Figure 2.2 and Figure 2.3 shows, the favorable and unfavorable conditions inside a unidirectional and bidirectional tunnel respectively in the event of fire when ventilation systems are operated. During a fire emergency in a unidirectional traffic condition, the tunnel ventilation system prevents upstream movement of smoke by supplying fresh air on one side of fire site while extracting the smoke away on other side. In case of bi-directional traffic tunnel, the ventilation system is operated in such a way that smoke layer is not disturbed and longitudinal air velocity is kept to a minimum. Smoke extraction is achieved through ceiling openings or through openings located high along tunnel walls.

In addition to smoke removal, ventilation systems maintains tenable environment inside the tunnel. Thus the overall objectives of ventilation are:

- To maintain acceptable levels of contaminants produced by vehicle engines during normal traffic operation.
- Heat removal during normal, emergency conditions and during maintenance operations.
- Emergency smoke removal. Fresh air supply for the evacuating passengers and the fire fighters.

The methods used to control air contaminant and smoke from fires in a tunnel include longitudinal flow, extraction and dilution. Ventilation may be provided by natural means, by the traffic-induced piston, or by mechanical equipment. The choice of type of ventilation system to be used depends on several parameters such as tunnel length, cross-section and grade of tunnel; surrounding environment; traffic volume and construction cost. A brief description of the various types of ventilation systems are given below:

2.1.1 Natural Ventilation System

In natural ventilation system air movements are induced by temperature or pressure gradients caused by meteorological effects (e.g. ambient temperature, or wind between the tunnel's two portals). The difference in elevation of two portals also maintains the airflow. In addition to this, the piston effect of moving traffic also creates an additional airflow. However, since meteorological conditions change with time hence this system cannot be relied solely for maintaining an acceptable environment in the tunnel all the time.

In case of natural ventilation, hot smoke rises from a fire towards the ceiling where it stratifies and subsequently propagates longitudinally away from the fire in both directions (Figure 2.1). In case of grade in tunnel, the buoyant effect of hot smoke and gases pushes smoke from a fire up the grade. The steeper the grade the faster the smoke will move. This traps the vehicle between the incident and the portal at the higher elevation and thus makes evacuation from the tunnel difficult. In such a scenario emergency mechanical ventilation is to be used to extract smoke and hot gases generated from the fire. It is recommended that tunnels longer than 244 m should have mechanical ventilation system (ASHRAE, 2003). Different countries have proposed different guidelines to limit the application of natural ventilation in road tunnels (PIRAC, 1999).

It can be concluded that natural and traffic-induced ventilation is adequate for relatively short tunnels and tunnels with low traffic volume (or density). Mechanical ventilation is necessary for long and heavily traveled tunnels.

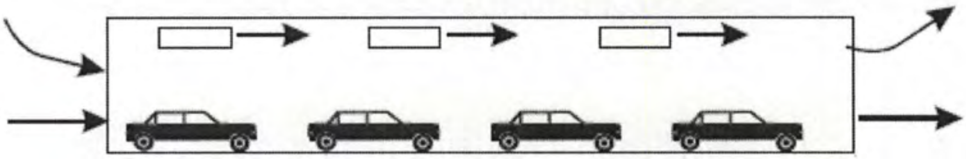
2.1.2 Mechanical Ventilation System

There are two basic types of mechanical ventilation layouts applied in transport tunnels: longitudinal and transverse.

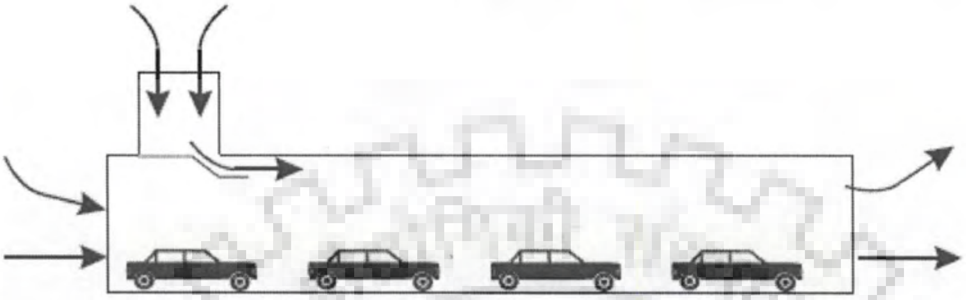
2.1.2.1 Longitudinal Ventilation

The longitudinal ventilation system (Figure 2.4) creates a longitudinal flow of air along the length of the tunnel by introducing or removing air from the tunnel at a limited number of points. The fresh air is supplied at the beginning of the tunnel or tunnel section and heated / polluted air is discharged at the tunnel portal or at the end of tunnel section. Thus the airflow moves the pollutants / heated gases along with incoming fresh air. Longitudinal ventilation can be configured either portal to portal, portal to shaft or shaft to shaft. The ventilation is provided either by jet fans (Figure 2.4a), by injection (Figure 2.4b), or by a combination of injection or extraction at intermediate points in the tunnel (Figure 2.4c and 2.4d).

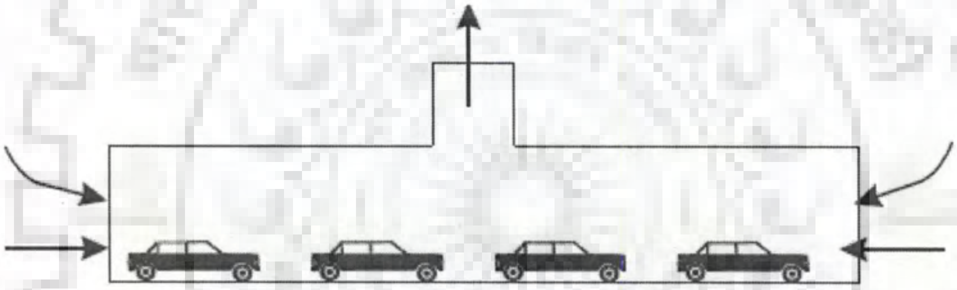
In a jet fan longitudinal ventilation system, jet fans are mounted on the tunnel ceiling in series along the tunnel length. These fans have high discharge thrust and velocity which in turn induces additional longitudinal airflow



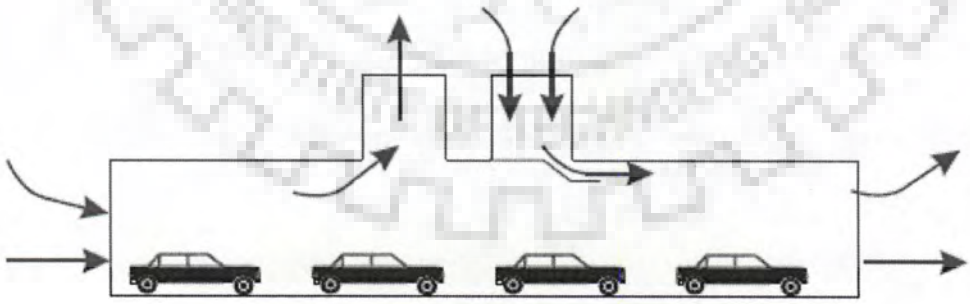
(a) Jet Fans



(b) Jet Injection



(c) one shaft



(d) Two shafts and jet injection

Figure 2.4 Longitudinal Ventilation System (Kashef et al., 2003) within the tunnel. This type of system does not require a large space to house ventilation fans in a separate structure or ventilation building. However, it may require a tunnel of greater height or width to accommodate the jet fans. The limitation is that the fans could be exposed to the high gas and smoke temperature.

An injection longitudinal ventilation system is one in which the fresh air is supplied or the smoke is extracted at a limited number of locations in the tunnel. Air is injected into the tunnel through the high velocity jet of Saccardo nozzle at one end of tunnel where it mixes with the air brought in by the piston effect of incoming traffic. The air velocity within the tunnel is uniform throughout the tunnel length. This system is economical as it uses the least number of fans and does not require extra ducts for the distribution of air. They have been used extensively in railway tunnels. A ventilation system could also be designed with two shafts near the center of the tunnel: one for exhaust and one for supply (Figure 2.4(d)). This arrangement has the advantage of reducing contaminant or smoke concentrations in one half of the tunnel.

The longitudinal form of ventilation is the most effective method of smoke control in tunnels with unidirectional traffic. In the event of a fire in a unidirectional tunnel, it is usually assumed that the traffic ahead of the fire will proceed to the exit portal and the traffic behind the fire will come to a stop. The ventilation system would be operated in such a way that it forces the smoke and hot gases in the direction of the empty tunnel in order to provide a clear and safe environment behind the fire for evacuees and fire fighters.

Figure 2.5 shows the effect of longitudinal air velocity on smoke progress in the fire zone. If the ventilation is weak (Figure 2.5a), the heated air and smoke may flow in the opposite direction.This phenomenon is called back layering. If the ventilation capacity is more (Figure 2.5c) all of the heated air and smoke will flow in the downstream direction. The ventilation velocity just sufficient to prevent back layering of smoke (Figure 2.5b) over the stalled vehicles is the minimum velocity needed for smoke control in a longitudinal ventilation system and is known as the critical velocity (V_c). The ability of the longitudinal ventilation system to prevent back layering is the current industry standard to measure the adequacy of the system for smoke control. If the longitudinal air velocity is greater than the critical velocity, the smoke downstream of the fire will not stratify (Figure 2.5c). In cases where the air velocity is lower than or equal to the critical velocity, the smoke would progress upstream of the fire and would remain stratified (Figure 2.5a and 2.5b).

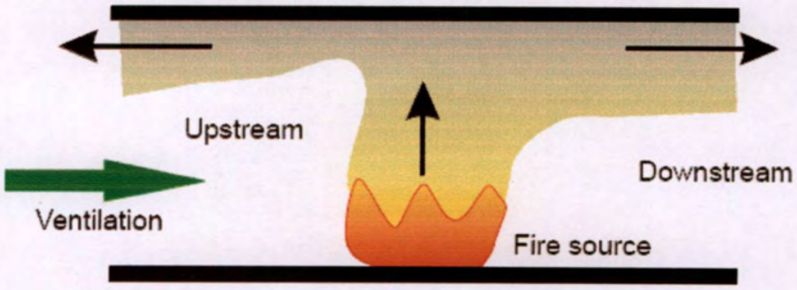
The critical velocity depends on the fire heat release rate (fire size), the slope, and the tunnel section geometry. The empirical relations pertaining to critical velocity are given in section 2.3.

2.1.2.2 Transverse Ventilation System

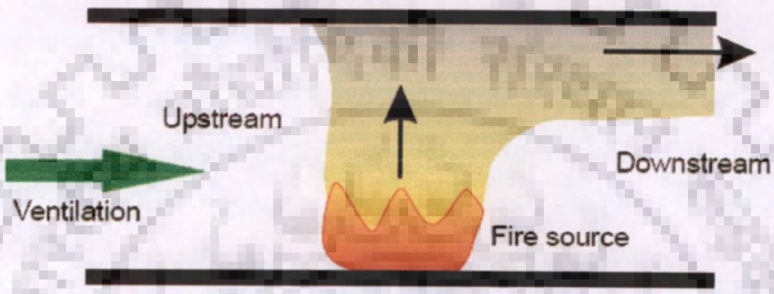
This type of ventilation system is defined by the uniform distribution of fresh air and / or uniform collection of polluted air along the length of the tunnel. The transverse ventilation system can be further classified as full transverse and semi-transverse ventilation system (Figure 2.6).

(a) Full Transverse Ventilation

This ventilation system (Figure 2.6a) comprises both a supply and an exhaust duct to achieve uniform distribution of supply air and uniform collection of vitiated air throughout the tunnel length.



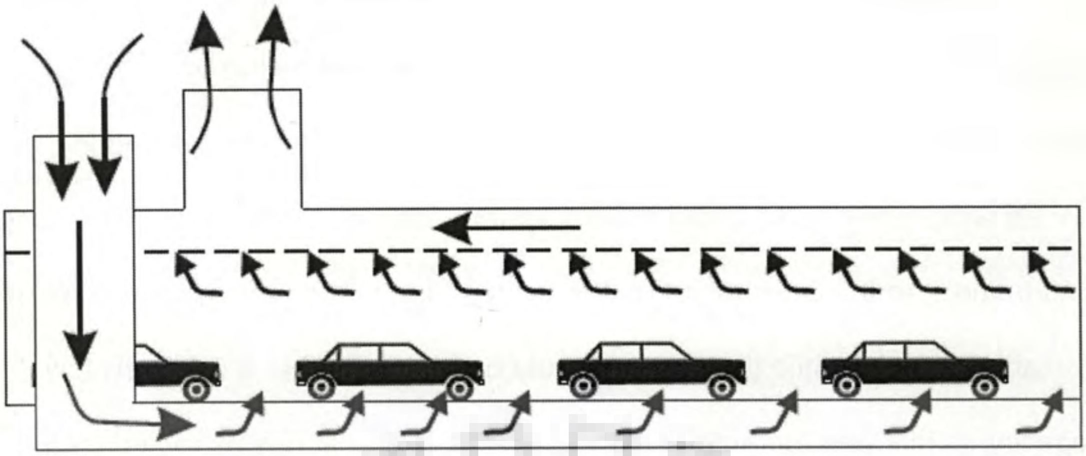
(a) $V_{vent} < V_c$



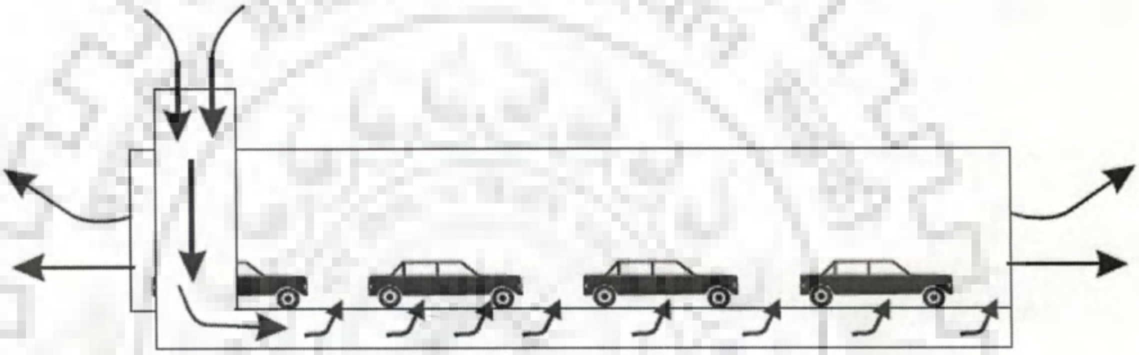
(b) $V_{vent} = V_c$

(c) $V_{vent} > V_c$

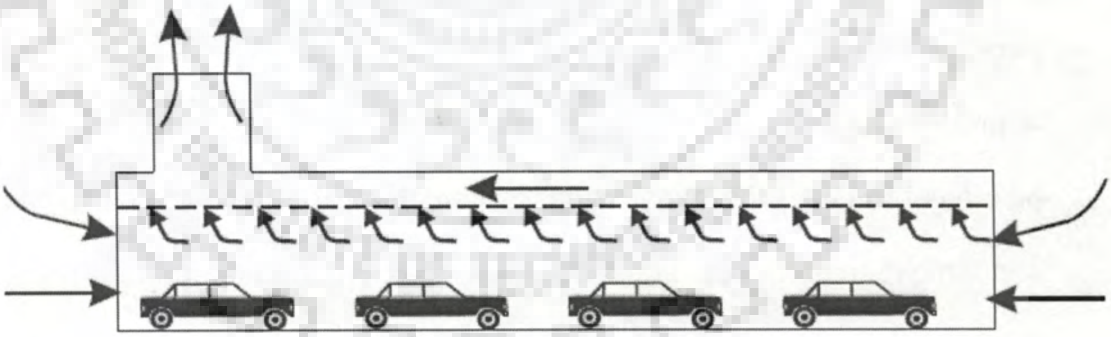
Figure 2.5 Influence of Longitudinal air velocity (V_{vent}) on smoke progress in the fire zone (V_c = critical velocity) (Kashef et al., 2003)



(a) Full transverse



(b) Semi transverse supply



(c) Semi transverse exhaust

Figure 2.6 Transverse Ventilation System (Kashef et al., 2003)

This configuration produces a uniform pressure along the tunnel with no longitudinal airflow being generated except that created by the traffic piston effect. Full transverse ventilation is used in extremely long tunnels and in tunnels with heavy traffic volume. This system has been used primarily in long road tunnels. In the event of a fire, the exhaust fan should attain its maximum available capacity while the supply should be maintained at a relatively low capacity. In this way the smoke remains at the ceiling and is extracted by the exhaust ducts without mixing with the lower fresh air.

(b) Semi Transverse Ventilation

A semi transverse ventilation system can either be a supply (Figure 2.6b) or an exhaust ventilation system (Figure 2.6c). This type of ventilation has the advantage of being less affected by atmospheric conditions since the tunnel airflow is fan-generated.

(i) Supply Air Semi-Transverse System

This type of ventilation system (Figure 2.6b) distributes air uniformly throughout the length of a road tunnel in a duct fitted with supply outlets spaced at predetermined distances. Thus fresh air is introduced at the vehicle exhaust pipe level to dilute the exhaust gases immediately. There are no exhaust fans and smoke and gases are exhausted through tunnel portals. This system produces a uniform level of pollutants and temperature throughout the tunnel because air and vehicle generated pollutants and heat enter the roadway area at same relative rate.

If a fire occurs in the tunnel, the supply air initially dilutes the smoke. Subsequently, it should be operated in reverse mode so that fresh air enters the tunnel through the portals to create a respirable environment for fire fighting efforts and emergency egress.

(ii) Exhaust Air Semi-Transverse System

This type of ventilation system (Figure 2.6c) collects air uniformly throughout the length of a road tunnel in a duct fitted with exhaust outlets spaced at predetermined distances. The fresh air is supplied through tunnel portals. The exhaust semi-transverse system produces maximum level of pollutants and temperature at the exiting portal.

2.1.3 Handbooks, Standards and Guidelines

There are number of books which describe tunnel ventilation. A brief list of these books along with their publishers is given in Table 2.1.

The Subway Environment Design Handbook (SEDH) (United states, 1975) is an outcome of the development of the SES numerical code. This document describes the ventilation and environmental control in transit (metro) tunnels and underground stations. The SEDH provides a detailed description of the evaluation and design process.

About the time the SEDH was under development, the National Fire Protection Association (NFPA) in the United States initiated to develop a fire protection standard for transit (metro) tunnels. In 1983 the first NFPA standard for transit systems was published as NFPA 130 'Standard for Fixed Guideway Transit Systems' (NFPA, 1983). This standard has been updated on a regular basis and is currently known as NFPA 130 'Standard for Fixed Guideway Transit and Passenger Rail Systems' (NFPA, 2000).

Table 2.1. Handbooks addressing tunnel ventilation

S No	Title	Edition	Date	Reference	Pertinent chapters
1	Subway Environmental Design Handbook. Vol 1 - Principles and applications	1st	1975	United States Deptt. of Transp., 1975	
2	Tunnel Engineering Handbook	1st	1982	Bickel et al., 1982	Chapter 19 - Tunnel ventilation Chapter 20 – Fire Protection
3	Tunnel Engineering Handbook	2nd	1996	Bickel et al., 1996	Chapter 19 - Fire life safety Chapter 20 – Tunnel ventilation
4	ASHRAE Handbook HVAC Applications		2003	Owen, 2003	Chapter 13 - Enclosed vehicular facilities
5	Fire Protection Handbook	19th	2003	NFPA, 2003	14.3-Fixed guideway and passenger rail systems 14.4 -Rail transportation systems 14.7-Fire protection for road tunnels

In the area of road tunnels, NFPA published for a number of years, beginning in 1972, a tentative standard NFPA 502T 'Standard for Limited Highways, Tunnels, Bridges and Elevated Structures' (NFPA, 1972). In 1981 it was converted to a recommended practice entitled NFPA 502 'Recommended Practice on Fire Protection for Limited Access Highways, Tunnels, Bridges, Elevated Roadways and Air Right Structures' (NFPA, 1981). This document was totally revised in the 1990s and ultimately emerged as an NFPA standard: NFPA 502 'Standard for Road Tunnels, Bridges and other Limited Access Highways' in 1998 (NFPA, 1998). The World Road Association (formerly the Permanent International Association of Road Congresses), now known simply as PIARC, has been publishing technical reports on tunnels and tunnel ventilation for the last 25 years in conjunction with their quadrennial World Road Congresses. The PIARC Technical Committee on Road Tunnel Operation (designated as (C3.3)) and its working groups have published several important specific documents on tunnel fire safety and ventilation as listed in Table 2.2.

Table 2.2. World Road Association publications on tunnel ventilation

SNo	Publication	Reference
World Road Association (PIARC)		
1	Report to the XVth World Road Congress, Mexico City, Mexico. 1975	PIRAC,1975
2	Report to the XVIth World Road Congress, Vienna, Austria, 1979	PIRAC,1979
3	Report to the XVIIth World Road Congress, Sydney, Australia, : 1983	PIRAC,1983
4	Report to the XVIIIth World Road Congress, Brussels, Belgium, 1987	PIRAC,1987
5	Report to the XIXth World Road Congress, Marrakech, Morocco, 1991	PIRAC,1991
6	Report to the XXth World Road Congress, Montreal, Canada, 1995	PIRAC,1995a
PIARC Committee on Road Tunnels		
7	Classification of Tunnels, Existing Guidelines and Experiences, 05.03.B, 1995	PIRAC,1995b
8	Road Safety in Tunnels, 05.04.B, 1995	PIRAC,1995c
9	Fire and Smoke Control in Road Tunnels, 05.05.B, 1999	PIRAC,1999
10	Systems and Equipment for Fire and Smoke Control in Road Tunnels, 2005	PIRAC,2005

PIRAC - Permanent International Association of Road Congresses (now World Road Association (PIARC))

In addition to the above, many countries have published tunnel guidelines and standards for use in their country. A partial list is shown in Table 2.3.

2.1.4 Recommended Practices / Design Criteria

The design principles along with types of ventilation have been described above. However, there are no universally accepted quantitative criteria of air temperature and velocity for the protection of motorists and firefighters during emergency situations. Some design criteria and recommended practice useful

Table 2.3 National Guidelines and Standards

SNo	Country	Publication	Reference
1	Austria	Design Guidelines Tunnel Ventilation	Planning Guidelines 2002
2	France	Safety in the Tunnels of the National Highways Network	Ministry of Public Works 2000
3	Germany	Guidelines for Equipment and Operation of Road Tunnels	RABT 2002
4	Italy	Safety of Traffic in Road Tunnels with Particular Reference to Vehicles Transporting Dangerous Materials	Ministry of Public Works 1999
5	Japan	Tunnel Ventilation Design Guidelines	PWRI 1993
6	Netherlands	I. Technical Standards for the Provision and Installation of Ventilation of Road Tunnels II. Technical Standards for the Provision and Installation of Fire Protection for Tunnels III. Ventilation of Road Tunnels	RWS 1991 RWS 1998 KIVI 1993
7	Nordic countries	Ventilation of Road Tunnels	NVF 1993
8	Norway	Norwegian Design Guide - Road Tunnels	Directorate of Public Roads 1992
9	Spain	Manual for the Design, Construction and Operation of Tunnels	IOS-98 1998
10	Sweden	Tunnel 99 - General Technical Specification	Swedish National Road Association 1999
11	Switzerland	Ventilation of Road Tunnels, Selection of System, Design and operation	ASTRA 2001
12	U.K	Design Manual for Roads and Bridges, Volume 2, Section 2, Part 9, BD 78/79: Design of Road Tunnels	Highway Authority 1999
13	U.S	Standard for Road Tunnels, Bridges and Other Limited Access Highways Standard for Fixed Guideway Transit and Passenger Rail Systems	NFPA 502, 2001 NFPA 130, 2000

are as follows:

(i) NFPA 502 standard (NFPA, 2001) outlines the design objective for the emergency ventilation system for egress routes which are as follows:

- A stream of no contaminated air is provided to motorists on a path of egress away from a fire.
- During emergencies, evacuees should not be subjected to air temperatures that exceed 60°C.
- There should not be very high ventilation rates which makes difficult for people to walk. Motorists under emergency conditions can tolerate velocities as high as 11 m/s.

(ii) It is recommended that the optical density or light attenuation of smoke in the emergency evacuation path must be maintained below a limit so as not to block the sight of emergency exit signs. For ex., NFPA 130 states that illuminated signs should be discernible at 9.1 m, while for doors & walls they should be discernible at 6.1 m. It is also to be ensured that that sufficient clear height should be maintained to facilitate escape & for fire fighters to take action.

(iii) Heselden (1976) recommended that there should not be fresh air jets openings in the ceiling because it will break smoke stratification. A fresh air supply through openings located at the bottom of the sidewall is more favorable to maintaining smoke stratification.

(iv) It is also recommended that emergency ventilation systems should be designed based on a design fire size. The design fire should correspond to the type of vehicles that are expected to use the tunnel. The fans should be designed to withstand elevated temperatures in the event of a fire and should "remain operational for a minimum of 1 hour in an air stream temperature of 250oC" (NFPA, 2001).

2.2 EXPERIMENTAL STUDIES

Till 1960s, experimental studies were carried out inside tunnels to study fire safety inside mine tunnels. In the early 1960s, many transport tunnels were being constructed through out the world, especially in the Alps. To address the question "what may happen in case of fire inside these transport tunnels", experimental testing of fires in vehicle tunnels began. Broadly these fire tests are carried out to understand the dynamics of fire inside tunnels and their interaction with fire protection measures like ventilation systems, sprinklers, tunnel lining etc. These tests also provides input data like initial conditions of tunnel environment, wind movements, ventilation flows, heat release rate of burning objects etc. for use in numerical modeling of fires inside tunnels. The sections below describe in brief the major fire experiments carried out both at large, reduced and laboratory scale.

2.2.1 Large Scale Fire Tests

Many large scale fire tests have been conducted. An overview of these large tests is:

(i) *The Ofenegg Tunnel Fire Experiments, 1965*

A series of fire tests were carried out in an abandoned railway tunnel, near Ofenegg, Switzerland in 1965 (Haerter, 1994) to ascertain the effect of an accident caused by a fuel tanker inside a road tunnel. The tunnel was 190 m long having a cross sectional area of 23 m^2. The tunnel was modified and a duct having vents at 5 m intervals along one side of the tunnel was made. The objective of these tests was to assess the necessary ventilation capacity during a fire. In all, 12 tests were carried out with different sizes of pool fires.

The main findings of the tests were as follows:

- Longitudinal ventilation increases burning rate of fire.
- The velocity and thickness of the smoke layer is larger for large fires.
- Loss of stratification was seen in case of tunnel subjected to longitudinal ventilation.
- Maximum temperatures were achieved within 1-2 minutes from ignition.
- It is not possible to survive within 30 - 40 m of a large pool fire with any ventilation configuration and the chances of survival downstream of the fire are substantially reduced with longitudinal ventilation.

(ii) The Zwenberg Tunnel Fire Experiments, 1976

In the early 1970's, many road tunnels ranging from 5–13 km were being constructed in Austria. The Austrian government carried out tunnel fire experiments inside an abandoned railway tunnel to gain better understanding of fires inside tunnels (Pucher, 1994). The tunnel was 390 m long and 20 m^2 in cross section. During these tests various ventilation systems were tested In all 30 fire tests (25 of a 6.8 m^2 petrol pool, 3 of a 13.6 m^2 petrol pool, one of a 6.8 m^2 diesel pool and one test with a mixed load of wood, car tyres and sawdust) were carried out. Temperature, gas concentration, visibility and degree of combustion were measured.

The main findings of these tests were as follows:

- Temperatures at the ceiling reached as high as 1200°C.
- Higher longitudinal air velocities destroyed smoke stratification.
- In case of transverse ventilation, the exhaust fans should be set to maximum exhaust, while the power of supply fans should be reduced by 20-30% to maintain stratification.

- The position of supply vents (above or below) is not crucial.
- For the smaller fires it was possible to extract all the plume gases over a tunnel length of approximately 260 m with maximum extraction and minimum supply of fresh air.

(iii) The Glasgow Tunnel Fire Experiments, 1970

These experiments were carried out in a 620 m long, 7.6 m wide, 5.2 m high disused railway tunnel in Glasgow (Heselden & Hinkley, 1970). Five sets of experiments were conducted. The fire load in each case was made up of 1, 2 or 4 trays of kerosene fuel, each tray (1.2 × 1.2m) having an approximate thermal output of 2MW. Temperature and Smoke movements were recorded. The main findings were:

- The smoke layer was approximately 1-2 m thick and it thickened with time. For the largest fire, the maximum smoke layer thickness was 3-4 m.
- There was a "plug" of smoke formed at the end of the tunnel as it encountered the crosswind outside. The smoke filled the entire height of the tunnel and tended to be drawn back into the tunnel by the fire-induced airflow.
- The air below the main smoke layer was not smoke free; there was some mixing of smoke with underneath fresh air.

(iv) PWRI Tunnel Fire Experiments, 1980

In 1980, tunnel fire experiments were carried out by The Japanese Public Works Research Institute (PWRI) to assess the capacity of the emergency facilities. A total of 16 full scale fire tests were carried in a 700 m long fire gallery and 8 full scale fire tests were carried in a 3.3 km road tunnel. Petrol pools (10 tests), six passenger cars and six buses (one vehicle per test) constituted the fire load (Mizutani et al., 1982). Temperature, gas concentrations, smoke, etc. were monitored in the gallery / tunnel subjected to natural and longitudinal ventilation. The main findings were as follows:

- At low longitudinal velocities (1 ms^{-1}) smoke stratification is partially destroyed and at high longitudinal velocities (2 ms^{-1}), it is completely destroyed.
- The heat release rate of a fire increased at higher longitudinal ventilation velocities.
- Sprinklers were not able to extinguish totally the fires. It only reduced the heat release rate. On the other hand, the use of sprinklers caused a reduction in smoke density near the ceiling and an increase in smoke density in the lower part of the tunnel.

(v) The EUREKA EU-499 “Firetun” Test Series, 1990-1992

The largest tunnel fire experiments with the largest scope till date was the EUREKA “firetun” test series. These tests were carried out between 1990 and 1992 by teams of fire researchers representing Austria, Finland, France, Germany, Italy, Norway, Sweden, Switzerland and the UK. (Studiengesellschaft Stahlanwendung e.V., 1995). Most of the fire tests were carried out in an abandoned tunnel near Hammerfest, Norway in 1992. The Hammerfest tunnel is a 2.3 km long mine tunnel having an irregular cross-section (approximately square), which varied between 30-40 m^2. Fire load included cars, train carriages, wooden cribs, heptane pools, a “simulated truck load” and a HGV fully laden with a cargo of furniture. Some fire tests were carried out at in a disused railway tunnel in Germany and at VTT in Finland. These tests were done using wooden crib as fire load. In addition to it some pool fire tests were carried out by the Institut National de l'Environment Industriel et des Risques (INERIS) France in 1992. The objectives of the project were to gather information on:

- Fire phenomena
- Escape, rescue and fire-fighting possibilities.
- The effect of the surrounding structure on the fire
- Reuse of the structure.
- Accumulation of theory (improving understanding of fire, modifying models, etc.)
- Formation, distribution and precipitation of contaminants

The data recorded included mass loss, temperature, gas concentrations, smoke density, airflow velocity, etc. Main conclusions drawn from this test series were:

- The maximum temperature during most of the vehicle fires ranges between 800- 900°C. In case of HGV, the maximum temperature reached 1300°C. There was significant reduction in maximum temperatures even at short distance from the fire location. Downstream temperatures were greater than upstream temperatures (Haack, 1995).
- The heat release rate of railway carriages was found to lie between 15 & 20 MW. The heat release rate of HGV was over 100 MW (Haack, 1995).
- A single railway carriage exhibited a HRR of 45 MW (Steinert, 1994).
- All road and rail vehicles registered a fast development in the first 10-15 minutes (Haack, 1995).
- Growth rates of vehicle fires varied from "medium" to "ultra fast" (Ingason, 1995a).

- Ventilation influences fire growth and burning pattern (Malhotra, 1995).
- The HRR of fire is significantly increased by a free supply of air (Malhotra, 1995).
- Controlled ventilation was found to be useful in smoke management (Malhotra, 1995).
- Increased longitudinal ventilation destroys stratification downstream of the fire (Malhotra, 1995).

(vi) The Memorial Tunnel Fire Ventilation Test Program (Mtfvtp), 1993-1995

The Memorial tunnel fire ventilation experiments were carried out by the Federal Highway Administration for the Boston Central Artery Tunnel project between 1993 and 1995 in a disused 850 m long two-lane road tunnel near Charlestown, West Virginia, USA (Massachusetts Highway Department / Federal Highway Department, 1996). It was the largest tunnel fire experiments in terms of scale and studied various ventilation configurations - natural, semi-transverse, fully transverse and longitudinal, for their ability to control / extract smoke. The objective was the testing of ventilation configurations. In all, 98 pool (diesel) fire tests having heat release rate from 10 MW to 100 MW were carried out. The main conclusions from the test series for the case of longitudinal ventilation are:

- Longitudinal ventilation using jet fans was found to be effective in controlling smoke for fires up to 100 MW. However, this configuration was found to be appropriate only for unidirectional tunnels.
- Longitudinal air velocity was dependent on the number of active fans and the thrust. It was found to be independent of the configuration of the fans.

- Longitudinal airflow was reduced by 10% for a 10 MW fire while it was reduced by 50-60% for a 100 MW fire.
- The critical velocity to prevent back layering of smoke was found to be in the range 2.5-3 ms^{-1} for 100 MW pool fires in the Memorial Tunnel.

(vii) <u>'Project Safety Test': Tests in The 2nd Benelux Tunnel, The Netherlands, 2001.</u>

There were number of accidents in the Alpine road tunnels between 1999-2001. Therefore a series of 26 tests were conducted in the 2nd Benelux Tunnel, Rotterdam, The Netherlands in 2001 (Directorate General for Public Works and Water Management, 2002). The objective of these tests were to study

- Spread of heat and smoke from fire.
- Influence of ventilation on fire size.
- Influence of sprinklers.
- Capabilities of fire detectors during tunnel fire accidents.

These tests were conducted using different fire loads such as fuel pools, cars, van, stacked loads representing HGV fires and small fuel basins. The tunnel was subjected to both natural and longitudinal ventilation. The conclusions drawn from the study were as follows:

- Conditions were lethal within 6 m of passenger vehicle and within 12 m of HGV fire.
- There was poor visibility because of smoke even at distances of 100 – 200 m from the fire source.
- High ventilation velocities retarded the development of fire caused by passenger car while it enhanced the development of fire caused by HGV fire.

- ❖ Sprinklers substantially reduced air and vehicle temperatures in the vicinity of fire. There was no formation of steam.
- ❖ Escape route signage became quickly invisible in smoke. It was recommended that signs should be situated at low levels.

(viii) Fire Tests in The Toumei-Meishin Expressway Tunnel, Japan, 2001.

Shimizu tunnel was being constructed between Toumei and Meishin in Japan in 2001. During construction phase, a series of tests were undertaken to understand fire behavior and smoke control in this tunnel and also to provide data for comparison with computational fluid dynamic study. The tunnel was a three lane 1.119 km long roadway having semicircular cross section (8.5 m high, 9 m radius and cross sectional area = 115 m^2). Fire load included pool fire and a bus (Kunikane et al., 2002a, b). The conclusions drawn from this study were:

- ❖ Backlayering distance of smoke was more than that expected in a two lane tunnel.
- ❖ Longitudinal ventilation does not have a predominant effect on heat release rate in this tunnel which has a large cross section.
- ❖ The maximum convective heat release rate of a fire in a large tunnel was greater than expected in a two lane tunnel.

(ix) The Runehamar Tunnel Fire Test Series, 2003

These tests were conducted in a 1.6 km long disused two lane road tunnel in Norway in 2003 (Lonnermark & Ingason, 2003; Ingason & Lonnermark, 2003). The tunnel has a cross sectional area of about 47– 50 m^2. At the location of fire experiments, a 75 m of tunnel was lined with protective lining. In all, four fire tests were carried out. The objectives of the test program were to study

- Fire development in a HGV cargo loads.
- Influence of longitudinal ventilation on heat release rate and growth rate of fire.
- Production of toxic gases.
- Fire spread between vehicles.
- Fire fighting possibilities.
- Temperature development at tunnel ceiling.

The peak heat release rates of the four fires used were 203, 158, 125, 70 MW. Two fans were positioned near the tunnel portal which would generate longitudinal air flow of about 3 m/s. The conclusions drawn from the study were as follows.

- HGV cargoes produced heat release rate comparable to that of fuel tanker fire.
- Maximum temperatures achieved in a HGV cargo fire was above 1300 °C.
- When fire reaches its maximum heat release rate, ventilation flow was reduced and back layering was observed.

2.2.2 Tests in Operational Tunnels

Most of the tunnel fire tests mentioned above was carried out in abandoned or disused tunnels. This is because fires with high heat release rates can damage tunnel lining and structure. However in these fire tests (except in the Memorial tunnel) the tunnel cross section was significantly smaller than most operational tunnels, which have normally two or three lanes of road. Therefore it is desired that experiments should be carried out in these real sized tunnels. Hence, some small scale fire (car and pool fires) tests have been carried out in new and operational road tunnels to test the ventilation system capabilities or as a fire-fighting exercise (Perard, 1992; Perard & Brousse, 1994). Very few details of these tests are available hence these are not discussed in detail. However for the sake of completeness, the tunnels (French road tunnels) in which these tests were carried out are as follows.

- Des Monts Tunnel, 1988.
- Nogent-Sur-Marne Covered Trench, 1988/89.
- Frejus Tunnel, 1992.
- FFF Tunnel, 1990.
- Monaco Branch Tunnel 1992.
- Grand Mare Tunnel, Rouen.

The overall lessons learned from all these fire tests as given by Perard (1992) were:

- A car fire in a tunnel is not very big. It is possible to pass the fire location on foot or by car.
- Cold smoke tests are not representative of real fires.

However, these tests helped in

- Increased knowledge of smoke behavior in various configurations.
- Some quantitative data on temperature and smoke movement.
- Experience gained by the tunnel operators and emergency services.

2.2.3 Small Scale Fire Tests

Fire tests in full size tunnels are expensive. For example, the cost of the EUREKA "Firetun" test series was about US$10,000,000 while that of MTFVTP cost was in excess of US$40,000,000 (PIRAC, 1999). Thus it is prohibitively expensive to carry out tests in full sized tunnels. Hence reduced scale and small scale experiments are carried out. These help in understanding behavior of fire and smoke in tunnels and also help in analyzing specific model design of tunnel at affordable price.

However, for these tests to be meaningful, a similarity should exist between the scale model and full scale tests. For this to happen, it is found that Froude Number, Reynolds number, Richardson number and Grashof number should be the same in both the scale-model and in full model. In reality however, this is not possible and only Froude number is conserved. This assumption is reasonable as variations in the Reynolds number are not particularly significant for highly turbulent flows. Some reduced scale test series reported in literature and which are relevant to this study include:

(i) Pool Fire Tests at The Londonderry Occupational Safety Centre, Australia

A series of five kerosene pool fire tests was carried out in a 130 m long, 5.4 m wide and 2.4 m high "mine roadway" tunnel near Londonderry, NSW, Australia (Apte et al., 1991). The two exhaust fans placed at one end of tunnel provided the necessary longitudinal ventilation. The tests were carried out using three different sizes of pools: 0.57 m, 1 m and 2 m diameter pools. The ventilation velocities were varied from minimum of 0.5 ms^{-1} to a maximum of 2 ms^{-1}. The experiments were carried out to test a numerical model. Some of the observations from the study include:

- HRR of the 1 m diameter fire decreased by 25% on increasing the ventilation rate from 0.5 to 2.0 ms^{-1}.

- There was significant back layering at 0.85 ms^{-1}. No backlayering was observed at an air flow of 2 ms^{-1}.

(ii) Reduced Scale Fire Tests in The HSL Test Tunnel, Buxton, UK, 1993

These tests were conducted within a "mock-up" of a Eurotunnel HGV shuttle carriage in the test tunnel at the Health & Safety Laboratory (HSL) near Buxton in 1993 (Bettis et al.,1993, Bettis et al.,1994a; Bettis et al., 1994b). The objective of the experiments was to provide data for validation of a CFD simulation of the interaction between the tunnel's longitudinal supplementary ventilation system (SVS) and a back-layering smoke flow. The tunnel was 366 m long, 2.56 m wide and had a 5.6 m^2 arch shaped cross-section. The fire load included wooden crib and kerosene pool fires which generated between 2 and 18.6 MW heat output. Three fans were housed at one end of the tunnel which provided longitudinal ventilation in the tunnel. It was found that at low rates of heat release, the critical velocity varied as the one-third power of the heat release rate, but at higher rates of heat release, the critical velocity became nearly independent of heat output over a wide range of fire sizes. The other findings of this study were:

- Temperatures near the ceiling reached 1000°C.
- Backlayering of smoke was prevented at ventilation rates above 1.25 ms^{-1}.
- No clear relationship between the critical velocity required to prevent back layering and the heat output of the fire can be deduced.
- The air velocity required to prevent any back layering was less than predicted by existing theories.

(iii) Test Series carried out by The Swedish National Testing And Research Institute (SP), 1995

These tests were conducted in 11 m long, 1.08 m wide and 1.2 m high fire gallery at SP (Sveriges Provnings) in Sweden in 1995 (Ingason, 1995b). A series of 18 fire tests were carried out. The pool fire tests were carried out using heptane (11), methanol (2) and xylene (5) as fuels. Two sizes of square fuel pans were used; 0.09 m^2 and 0.16 m^2. The tests were carried out to investigate the effect of under-ventilation on the HRR of a fire. Comparable fire tests were also carried out in the open air. The following observations were made:

- Restricting the airflow caused a decrease in HRR.
- The mass burning rate (and hence the HRR) was much higher in the tunnel than in the open for heptane and xylene fuel pools. However, the mass burning rate for methanol fires was slightly lower in the tunnel than in the open.
- Burning rate of heptane & xylene fuel pools decreased with increasing wind velocity.
- With restricted airflow, the flames (from heptane pools) were deflected by about 30-45°.

(iv) Test Series carried out by The Swedish Defence Agency (FOA)

A series of 24 fire tests were carried out in a 100 m long "blasted rock tunnel", approximately 3 m wide by 3 m high. The tunnel was open to the air at one end and the other end had a large chimney. There was no mechanical ventilation system installed in the tunnel. Different ventilation rates were achieved by restricting the inflow of air. The experiments were carried out using fire load of heptane pools (12), methanol pools (2), kerosene pools (2), polystyrene cups in cardboard boxes (2), wooden cribs (3), heptane pools contained within a dummy vehicle (2) and a car (Ingason et al., 1997). The tests were carried out to provide experimental data for testing of CFD codes. The conclusions drawn were:

- The fire tests showed that there exist some correlation between the degree of ventilation and the heat release rate of the fire. This effect was found to be more apparent for solid fire loads than for liquid pools.
- The HRR of a small car reached 4 MW for a short period of time.

2.2.4 Laboratory Scale Experiments

Apart from reduced scale tests, a number of laboratory scale tests have also been carried out. However, the similarity of lab scale experiments to full scale fires is not good. When using reduced scale experiments, Froude number should be conserved. Bettis et al. (1994c) found that scaling criteria may not apply for tunnels having diameter less than 1 m. For example, a pool of hydrocarbon fuel will exhibit laminar-turbulent flame behavior when diameter of tunnel is less than 1 m, whereas larger pool fires will exhibit turbulent behavior. This does not imply that laboratory scale experiments are meaningless. These experiments also gives useful results and helps to understand better some features of tunnel fire dynamics at a fraction of cost of full scale experiments. Some of important laboratory scale experiments reported in literature are as follows:

(i) Experiments carried out at US Bureau of Mines

Lee et al. (1979) conducted laboratory-scale experiments in a 13.6 m long, 0.27 m^2 cross-section wind tunnel having an oak-lined burning working section of length 7.1 m. The measured data included flame spread rate, ventilation flow characteristics, heat fluxes, gas temperatures and gas composition. Their results showed that the airflow induced by a given fan in the presence of a tunnel fire could be less than half the value attained in the same system in the absence of the fire. They devised a simple criterion to ascertain whether or not backflow would occur. This criterion was based on correlations of the reversed layer velocity, a density-modified Froude number (just upstream of the fire), and a Froude number based on far upstream conditions.

Chaiken et al. (1979) also conducted similar experiments, using a coal-lined working section. The objective was to investigate the same parameters as above, and also to investigate the response of fires to scaling. As observed by Lee et al. (1979), reverse flow was again observed in the upper level of the duct, upstream of the fire. He also used Froude number to estimate the critical velocity required to prevent backflow. The results suggested that the ratio of the fire zone length to the duct diameter should be included as a non-dimensional scaling parameter. However Grant et al. (1998) observed that this conclusion was probably specific to similar experimental and prototype configurations where the length of the fire zone represents a significant proportion of the total tunnel length.

Heselden (1976) referred to research in the area of coal mines in his discussion of road tunnel fires, and evolved an expression relating the velocity required to prevent back layering to the thermal output of the fire. He considered the effect of a longitudinally ventilated fire upon the tunnel environment downstream of the fire site.

(ii) A Scale Model to Investigate Smoke Movement in The Paris Metro (1991)

These experiments were carried out in a 3 m long, 0.15 m radius semicircular tunnel made of Pyrex. The objective was primarily to investigate backlayering for the Paris Metro tunnel system (Vantelon et al., 1991). The fire source was a porous burner 2 cm in diameter and ventilation was obtained using an extraction fan at one end of the apparatus. Ventilation flows up to 25 cms^{-1} were produced. The conclusions drawn were:

- ❖ In case of fire, some back layering will occur in the Paris Metro tunnels even if the usual ventilation velocity of 1.5 ms^{-1} is used.
- ❖ In the absence of forced ventilation (natural airflow at about 0.2-0.4 ms^{-1}) the smoke from a fire is likely to remain stratified, so a smoke-free layer will exist near to the floor.

(iii) A Wind Tunnel Study to Investigate the Influence of Ventilation on Pool Fires (1995)

These experiments were carried out in a 21.6 m long wind tunnel, having square cross section (0.09 m^2) at the Japanese Fire Research Institute (Saito et al., 1995). The objective was to investigate burning rate, ceiling temperature and backlayering phenomenon. The influence of longitudinal ventilation on burning rate for pool fires was studied using different diameters of pools and different ventilation velocities. For this, 31 methanol pools of 10, 15, 20 & 25 cm diameters were tested with ventilation rates ranging from 0.08 to 1.03 ms^{-1} and five n-heptane pools of 15cm diameter were tested with ventilation rates ranging from 0.43 to 1.3 ms^{-1}. The back layering phenomenon was studied using different sizes of methanol pools, different ventilation rates and different tunnel slopes. The conclusions from this study were:

- ❖ Air velocity affects the burning rates of liquid pools in tunnels.
- ❖ The burning rate of a pool fire inside a tunnel was found to be higher than in the open air.

- The ceiling temperature of the tunnel fire was found to be inversely proportional to wind velocity.
- The relationship between critical velocity and heat output rate was almost linear.
- When back layering is long (twenty times tunnel height) the back layering length is only weakly dependent on the slope of the tunnel (up to 10°) or the fire size. When back layering is short it was found to be dependent on both fire size and tunnel slope.

(iv) Critical Velocity Experiments in a Laboratory Scale Model of The HSL Tunnel (1995)

A series of fire tests was carried out in a 1/10 scale model of the HSL tunnel which was 366 m long and have a cross section of 5.6 m^2. The apparatus was mounted in such a way that the tunnel can be inclined up to 10°. The fire source used in the test series was a propane burner, which can produce heat release rates of 2.8-14.1 kW. The longitudinal airflow was produced using compressed air at one end of the apparatus (Wu et al., 1997a). It was observed that slope has a modest effect on the critical velocity.

The same apparatus was then modified to study the effect of tunnel geometry on critical velocity (Wu et al., 1997b). For this, in addition to above tunnel configuration, experiments were also carried out in two different tunnel cross sections: 250 mm x 250 mm and 500 mm x 250 mm. The results from the three series were compared and following conclusions were drawn:

- The critical velocity required to control smoke varies with the mean hydraulic diameter of a tunnel (ratio of 4 times the tunnel cross-sectional area to the tunnel perimeter).

- A universal formula for critical velocity may be possible (subject to further work).

In addition to above, there had been some other tunnel fire tests carried out at laboratory scale. For example, Xue et al. (1993) carried out experiments to obtain data for model validation. They performed tests in an 8.45 m long circular duct of 0.25 m radius. A premixed gas burner produced heat output of 3 kW and two tests at velocities of 0.46 and 0.92 ms^{-1} were conducted. Megret et al. (1995) described a new lab scale apparatus to study smoke control in tunnels, although no experimental results have been published to date. Oka et al. (1996) tried to reproduce tests of a large scale laboratory test conducted by Bettis et al. (1994a). They built a one-tenth scale model in the laboratory of the geometry. They scaled the fire sizes and ventilating flows using Froude modeling. They found that the correlation of results with the large scale experiments was extremely close. The critical velocity varied as one-third power of heat output for small fires but tends to a near constant value when flaming extended over the height of the tunnel. Vauquelin & WU (2006) carried out experiments on scale models and CFD calculations to study the influence of tunnel width on critical velocity (for a given tunnel height H). Two different experimental reduced scale models were used: the first one was a thermal model using a propane gas flame to simulate the fire and the second one was a densimetrical model in which the fire induced smoke was represented by a continuous release of an isothermal buoyant mixing. They found that for aspect ratios (W/H) greater than unity, the critical velocity decreases when the width increases, as predicted by theory, but for low values of the aspect ratio (i.e. when W < H) and for high enough fire heat release rates, the critical velocity significantly increases with tunnel width. However, the critical velocity decrease for aspect ratios greater than unity did not appear to be in accordance with Thomas theory ($V_c \sim W^{-1/3}$). They also performed CFD calculations to describe the influence of the lateral confinement on smoke plume spreading and on critical velocity.

Most of the small scale and some of the larger scale experiments have been carried out in parallel to numerical or computational studies. These are described in section 2.3 of this chapter.

2.2.5 Remarks

It can be seen from the above mentioned different experimental studies that the conclusions drawn from some of the studies appears to contradict the conclusions drawn from other studies. It is observed that most of these tests were conducted with uncontrolled oil-pool fires, which develop and behave differently to vehicle fires. In oil pool fires, the HRR is constant while in vehicle fires, the HRR varies with time. It is also observed that in most of the tests the fire source is centrally placed while this may not be the case in reality. The smoke spread in such a case is not known. However, several aspects of tunnel fire behavior which has been observed in different experimental test series are summarized below:

- Thick smoke layer is produced inside the tunnel which advances faster than walking speed.
- Longitudinal ventilation tends to destroy smoke stratification.
- Smoke does not remain stratified even in naturally ventilated tunnels.

- Longitudinal ventilation enhances the burning rate of fuel. This is more evident in vehicle fires. However, the burning rate of some pool fires is reduced by increased longitudinal ventilation.
- All vehicle fires exhibit a 'fast' rate of fire development.
- The heat release rate of car fires in tunnels may be significantly larger than in the open air.
- Temperatures above 1000°C are common in tunnel fires.
- High temperatures are only evident in the immediate vicinity of car fires.

2.3 NUMERICAL MODELING OF FIRES

Fires are, in general, very complex in nature. This complexity arises because the flow behavior associated with fires is three dimensional, turbulent and influenced by buoyancy forces. A prior understanding of the behavior of fire and smoke movement in tunnel environments is necessary to assess the hazards arising from tunnel fires and to plan effective mitigation strategies. Today, fire models are increasingly being used for various aspects of tunnel fire safety design such as ventilation, smoke management, fire suppression and egress analyses. A number of predictive models have been developed. These range from simple empirical relations to most complex three-dimensional CFD models. All models have their utility and scope. The simpler models are ideal for repeated applications and are cost effective. On the other hand, three dimensional CFD programs are costly and simulations are computationally expensive. The objective is to have a well-validated model which could help in designing appropriate safety systems.

A number of studies on simulation of fires in longitudinally ventilated tunnels have been reported in the literature. The various models developed or used in these studies belong to any one of the following categories:

- Empirical Models
- Network Models
- Phenomenological Models – Integral / zone models
- Field Models or CFD Models.

2.3.1 Empirical Models

The simple empirical models comprise of one or two algebraic equations that are used to determine critical velocity needed to prevent upstream movement of smoke from a fire in a tunnel. These equations do not predict downstream conditions. A detailed review of critical velocity is given by Grant et al. (1998).

The values of the critical velocity for various tunnels have been obtained mainly from sets of equations derived by applying Froude number preservation, combined with some experimental data. The most commonly used models are those proposed by Heselden (1976) and Danziger and Kennedy (1982), the latter was incorporated in the US Department of Transport Subway Environment Simulation Programme (United States Department of Transportation, 1975).

The theory of critical ventilation velocity is based on Froude number preservation and is presented below.

2.3.1 Critical Ventilation Velocity-Theory Based on Froude Number Preservation

The Froude number is defined as

$$Fr = \frac{V^2}{gD} = \frac{\text{inertia force}}{\text{gravity force}} \tag{2.1}$$

where, g is the force due to gravity, V and D are the characteristic values of velocity and length. Froude no. modeling is often used as a scaling technique in fire situations. Ideally, Froude number, Reynolds number, Grashof number should be conserved while scaling. However since in these problems Reynolds number is sufficiently large, turbulent conditions prevail and buoyancy forces are dominant, hence only Froude number is conserved. It is sometimes combined with density ratio of smoke to include the effects of stratification. It is then called the Richardson number or modified Froude number and is defined as:

$$Ri = \frac{1}{Fr}\frac{\Delta\rho}{\rho} \tag{2.2}$$

Thomas (1970) was one of the earliest who applied this technique to study the effect of ventilation velocity on fire plumes in ducts. He proposed a simple relationship that determines critical velocity necessary to prevent upstream movement of smoke. He suggested that the flow character depended on the ratio of buoyancy to inertial force over a cross section of the tunnel. This ratio could be described by a global parameter having the form of a modified Froude number

$$Fr_m = \left(\frac{gH\Delta T}{V^2 T}\right) \tag{2.3}$$

where, H is the tunnel height, V is the ventilation velocity, T is the hot layer temperature and ΔT is the temperature rise above ambient. Thomas suggested that when $Fr_m = 1$ the magnitude of buoyant force = inertial force and back layering will not occur. He then substituted the expression relating ΔT and Q_c, the fire heat release rate (convective component of heat release) and proposed a relationship between ventilation velocity and the heat release rate as

$$V_c = k\left(\frac{gQ'}{\rho_o C_p T}\right)^{1/3} \tag{2.4}$$

where, V_c is the critical ventilation velocity, Q' is the convective heat release rate per unit width of the tunnel, ρ_o is the ambient air density, C_p is the specific heat capacity of air, K is a constant of order unity and T is the smoke temperature. The above expression indicates that critical velocity increases as heat release rate increases irrespective of the fire size.

However, Bettis et al. (1993, 1994a) in their experiments carried out in a full-scale colliery-shaped tunnel at the Health and Safety Laboratory in Buxton, UK found that at low rates of heat release, the critical velocity did vary as the one-third power of the heat release rate, but at higher rates of heat release, the critical velocity became nearly independent of heat output over a wide range of fire sizes. These finding were also found to be correct from the CFD simulations done by Lea (1995). Lea carried out CFD simulations to study the critical velocity versus heat release relation. The most comprehensive large-scale tunnel fire tests, The Memorial Tunnel Fire Ventilation Test Program carried out in the USA by Parsons Brinnckehoff (Massachusetts Highway Department / Federal Highway Department, 1996) also showed that the theory based on the Froude number preservation tend to over predict the critical ventilation velocity for the sizes in the 50-100 MW range by approximately 5-17%.

Therefore the models based on the Froude number preservation are not suitable to predict the critical ventilation at high heat release rate. Another limitation of Froude number preservation approach is that it is assumes that the buoyancy force is mainly a function of tunnel height and therefore uses tunnel height as characteristic length. It does not consider the effect of the tunnel geometry on the critical velocity.

Hinkley (1970) used the same theory as Thomas to derive a formula for calculating the velocity of hot gases traveling along the roof of a shopping mall. Here he assumed that the magnitude of the ventilation velocity was equal to that of the hot gas velocity and was given by

$$V_c = K\left(\frac{gQ_cT}{\rho_o C_p T_o^2 W}\right)^{1/3} \tag{2.5}$$

where, T_o is the ambient temperature and K is a constant equal to 0.8. Grant et al. (1998) concluded that the Hinckley's relation of critical velocity had little foundation on experimental data from fires in tunnels or corridors because this expression had not been tested by him for the case of fires inside ventilated tunnels. However, based on the Hinkleys theory, Heselden (1976) derived another formula for calculating critical ventilation velocity.

$$V_c = CK\left(\frac{gQ_cT}{\rho_o C_p T_o^2 W}\right)^{1/3} \tag{2.6}$$

Heselden based on the test of Glasgow Tunnel fire experiments assumed that K is equal to unity and fixed C at 0.8. Heselden determined C from only from three sets of experimental data. Heselden also tried to compare the Ofenegg tunnel fires data (Haerter 1994) to predictions using above equation but the results were not found to be in agreement.

Danziger and Kennedy (1982) devised a simple model based on Froude number approach to calculate critical ventilation velocity. The expression proposed by him is

$$V_c = k_g K \left(\frac{g Q_c H}{\rho_o C_p T \ A} \right)^{1/3} \tag{2.7}$$

where

$$T = \left(\frac{Q_c}{C_p \rho_o A V_c} \right) + T_o \tag{2.8}$$

Here A is the cross sectional area of the tunnel, K is a constant set to be equal to 0.61 and k_g is the tunnel grade (ω) correction factor to be applied for fires in sloping tunnels. It is defined as

$$k_g = 1 + 0.0374 * (\omega)^{0.8} \tag{2.9}$$

They determined this figure on the basis of experimental work carried out at US Bureau of Mines on fires in small-scale wood-lined ducts (Lee et al., 1979).This model is being used by Subway Environmental Simulation code (SES) (United States Department of Transportation, 1975) for predicting ventilation flows in tunnel networks. The SES fire model predicts that the critical velocity rises as the fire heat output increases. The appearance of T in the denominator of above equation ensures that at large heat outputs the critical velocity tends to an asymptote of near-constant velocity. This behavior showed some similarity to that observed in the Buxton fire trials (Bettis et al. 1993).

It can be seen that the values of K proposed by Danziger & Kennedy's (0.61), Thomas (1) and Heselden (0.8) are different because the experimental set up on which these predictions are based are different and varies in terms of tunnel shape, size and fire scenario.

The SES fire model is normally used in tunnel ventilation system design. The above model is also incorporated in D'Albrand & Bessiere's (1992) VENDIS-FS tunnel network code, developed at the Institut National de l'Environment Industriel et des Risques (INERIS), as well as Te Velde's (1988) zero-dimensional code for simulating flow in a vehicle tunnel incorporating slip roads.

Vantelon et al. (1991) developed a simple model for predicting the length of a backed-up smoke layer (L). The model is based on a very small scale experiment conducted in a 1.5 m long semicircular pipe of 15 cm radius. Vantelon found that the following relationship gives a good correlation of his data:

$$\frac{L}{H}\,\alpha\left(\frac{gQ}{c_p\rho_0 T_0 V^3 H}\right)^{0.3} \qquad (2.10)$$

In Vantelon's experimental work, the backed-up layer length was only varied by about one pipe height at each value of Q. The heat outputs taken were very small; less than 1 kW and the measurements were made at only one scale. Grant et al. (1998) concluded that there is no experimental basis for including H as a parameter in equation 2.10. Also the above relationship may not be applicable to any other tunnel system. These major deficiencies led to the conclusion that the work is of little value.

Oka et al. (1996) tried to measure critical velocities for a model tunnel (same as that of HSL, Buxton) fire using large range of fires sizes including flames with lengths much smaller and much larger than the tunnel height. They explained the variation of critical velocity with heat output by examining the buoyancy head produced by fire plumes. They used results obtained by

McCaffrey (1979) for a free plume which suggest that the temperature, and hence the density difference, above a large fire is roughly constant for very large fires for all heights and heat outputs. They proposed a dimensionless correlation for critical velocity which is as follows:

$$V_c^* = K * \left(\frac{Q^*}{0.12}\right)^{1/3} \quad \text{for } Q^* \leq 0.12 \tag{2.11}$$

and

$$V_c^* = K \text{ for } Q^* \geq 0.12 \tag{2.12}$$

where, the value of K varies from 0.35 to 0.31. For a fire on floor without significant blockage K is 0.35 and for fire that extends to full width of tunnel K = 0.31

and, the dimensionless HRR, Q*, and the dimensionless critical velocity, V_c^*, are given by

$$V_c^* = \frac{V_c}{\sqrt{gH}} \tag{2.13}$$

$$Q^* = \frac{Q_c}{\rho_0 C_p T_0 g^{1/2} H^{5/2}} \tag{2.14}$$

Wu and Bakar (2000) showed that the above dimensionless analysis cannot correlate the experimental results from the five tunnels of different shape and size into a simple form. Therefore, the formulae proposed by Oka and Atkinson should not be used for tunnels with a shape different from the arched Colliery shape. They suggested that the problem with the dimensionless analysis method proposed by Oka and Atkinson is that the tunnel height is used as the characteristic length in the analysis. Their experimental results demonstrated that for a tunnel having the same height but different width, the critical velocity varies with the tunnel width. They concluded that tunnel height is therefore not suitable as the characteristic length. They then proposed a modified form of the above correlation which uses tunnel hydraulic height instead of height, and the proposed limit of Q* to be 0.20. Thus the formulae proposed by them is

$$V_c^* = 0.40 * \left(\frac{Q^*}{0.20}\right)^{1/3} \quad \text{for } Q^* \leq 0.20 \tag{2.15}$$

and

$$V_c^* = 0.4 \text{ for } Q^* \geq 0.20 \tag{2.16}$$

where the dimensionless HRR, Q*, and the dimensionless critical velocity, V_c^*, are given by

$$V_c^* = \frac{V_c}{\sqrt{gH_D}} \tag{2.17}$$

$$Q^* = \frac{Q_c}{\rho_0 C_p T_0 g^{1/2} H_D^{5/2}} \tag{2.18}$$

Kang (2006) studied the effect of enclosure obstruction inside a tunnel numerically and concluded that the hydraulic diameter, if computed on the basis of annular area instead of tunnel cross section gives better results. He proposed to replace height and area in equation 2.7 and 2.8 with hydraulic height (based on annular area) and area of annular region respectively. His findings were based on blockage ratios of 25, 50 and 65%. However he concluded that these findings are subject to experimental validation. The annular area (A_a) and tunnel hydraulic height (H_D) is defined as

$$A_a = A - A_b \tag{2.19}$$

$$H_D = \frac{2 x (A_a)}{2 x \sqrt{A} + \sqrt{A_b}} \tag{2.20}$$

where A_b is the frontal cross sectional area of obstruction.

2.3.2 Network Models

The principle underlying these models is that tunnel is represented by a network of tunnel sections which intersects at junctions. Each section is assumed to have uniform local fluid properties. The equations for conservation of energy are applied for each section, and mass conservation equations are applied at each junction. From these conservation equations, a system of algebraic equations is obtained which are solved iteratively. Thus in these models, the downstream time-development of smoke movement and its relative concentration can be computed. Examples of network models used in the design of ventilation systems for tunnel networks SES (Parsons Brinkerhoff, USA), VENDIS-FS (INERIS, France), MFIRE (Chang et al. 1990), HOTFLOW (Mott MacDonald, UK), CAMATT (CETU, France), EXPRESS' AIR (Hydratec, France), and SPRINT (HBI Haerter, Switzerland). All these models have been validated to some extent against full-scale fire test data.

2.3.2.1 Remarks

These models have limited scope and do not predict downstream conditions. These models show that critical velocity varies with one third power of heat release rate which are not applicable for predicting critical velocity at high heat release rates. The network models are simple and easy to use for designing tunnel ventilation system. These models are able to predict pressure distributions, airflows, temperatures and smoke propagation. However, they are not able to simulate the complex 3-D fluid flow and heat transfer phenomena involved in vehicle fires. Smoke stratification and back layering are also not predicted. Thus these models only provide a global view of the effects of fire in a tunnel network. The results from these one dimensional models can be used to determine boundary conditions for more complex modeling methods.

2.3.3 Phenomenological Models

Phenomenological models provide more detail than the simple empirical models, but less than that can be available from CFD techniques. Solutions of these models are obtained by solving a set of ordinary differential equations. The two approaches used in Phenomenological models are integral method or zone model method. In each approach the domain of interest is split into several distinct regions. In the integral approach, one-dimensional integral equations for mass, momentum and energy conservation are approximated over each of these regions. The resulting ordinary differential equations are then solved using commonly available techniques. On the other hand, zone models divide each room into a small number of volumes, such as upper hot layer, the lower cold layer, the fire plume and the compartment boundary. Each of these volumes is assumed to be uniform in its properties. That is, the temperature, smoke and gas concentrations within each layer are assumed to be exactly the same at every point. Heat and mass transfer between the layers are processed through the plume. Equations describing the conservation of mass, momentum and energy are solved numerically together with previously developed empirical equations derived from experiments such as plume models, vent flow equations, ceiling jet correlations, radiation and combustion models. A comprehensive discussion of this type of model may be found in reviews by

Budnik & Walton (1990); Cox (1995); Thomas (1995) and Altinakar et al. (1997). These models run very quickly on a personal computer. The main differences among the various models as summarized by Charters (2005) are as follows:

- Three layers (hot, mixing/shear and cool) or two layer (hot and cool)
- Fundamental conservation equations applied to control volumes or integration of differential equations
- Transient or steady state
- Bespoke tunnel fire model or modified compartment model
- Ventilation system employed

There are very few published phenomenological models applicable to tunnel fires. The earliest models developed for tunnel fire applications are those of Hwang et al. (1977), Daish & Linden (1994) and Charters et al. (1994). Hwang et al. method employs an integral approach, whilst Charters method employs zone model of tunnel fires. Daish & Linden's model is a hybrid of the two, using an integral method only for the hot stratified layer, coupled with an algebraic relation for the thickness of the mixing layer which exists below the hot layer. Chow (1996) used multi room model 'CFAST" (Jones 1990) and Suzuki et al. (2002) used a self developed Multi layer zone model applicable for single room to study tunnel fires which was later modified by Xiaojun (2008). Another two layer multi compartment model based on modification of CFAST, namely TUFISI (Tunnel Fire Simulation) was developed by Altinakar et al. (1997).

Hwang et al. (1977) model divided the tunnel into three regions: a plume whose angle of inclination was affected by the imposed longitudinal ventilation, a plume ceiling impingement zone referred to as a 'turning-region' and a hot ceiling layer. For each of these regions number of assumptions was taken. These are necessary to yield closed forms of the governing equations. For example, the plume was initially assumed to be two-dimensional, vertical and with the fuel being instantaneously burned at its base. They estimated entrainment of ambient air from plume theory, and assumed properties to be uniform across the plume. Inside the turning-region, the fluid density was assumed to be constant. Similarly, all properties were assumed to be uniform over the cross-section in the ceiling layer, and they estimated entrainment rates again. No feedback from the computed upstream conditions to the plume region was considered. The modeling of the plume supplies initial conditions for calculating the developing upstream and downstream hot layers. The only comparison which Hwang et al. made between model predictions and measurements was with a single data point due to Eisner & Smith (1954). They found that the backed-up layer length was grossly underestimated.

Daish & Linden's (1994) assumed that the flow composed of a hot plume and ceiling layers which travels in both up and downstream. In the region downstream from the fire, they assumed that a layer of air at ambient temperature exists at low level. The fire plume was considered to behave according to classical theory and its trajectory was assumed to be unaffected by imposed ventilation. However, several workers had observed that the plume can be inclined at substantial angles to the vertical. For example, Hwang et al. (1977) simple model for the plume-ceiling impingement region indicated that the possibility of having inclinations between 45° and 60° to the vertical. They evaluated this model by comparison with the data obtained from the comprehensive Buxton fire trials. The temperature, depth and propagation distance of the hot layers were predicted. They concluded that the model was able to reproduce existence of any backed-up flow. The evolution of temperature in the hot layer and the downstream distance at which the hot layer breaks down to well-mixed conditions was well predicted provided that fire was not too large and the ventilation not too strong. The position at which breakdown of the hot layer was deemed to have occurred in the experimental trials was, however, somewhat arbitrary, because in reality this layer was not clearly defined in experimental data. The theoretical model also exhibits same behavior of continued rise of critical velocity with fire size as predicted by simpler models based on Thomas's work.

Charters et al. (1994) developed a multi-zone, three layer zone model, FASIT (Fire growth And Smoke movement In Tunnels) to simulate fire growth and smoke movement in tunnels subjected to either natural ventilation condition or longitudinal ventilation. The layers can be described as hot, mixing and cool. The layers are mixed on the basis of a Richardson number (dimensionless ratio of buoyancy and inertia) criterion combined with an empirical entrainment coefficient for buoyancy-driven flows. Charter et al. stated that three layer approach was better than two layer approach as used in modeling of compartment fires because of stronger mixing effect resulting from horizontal flow in tunnels. The model predicts temperature, depth and concentration of each layer's at various times. The authors compared their model to temperature profiles measured downstream of a wood fire in a tunnel. The FASIT model captured the qualitative flow behavior but generally over-predicted the measured temperatures by about 100°C. This model was also validated from experimental test results of Ofenegg Tunnel fire tests.

Chow (1996) used fire zone model 'CFAST', a multi-room zone model (Jones and Forney, 1990), to simulate a tunnel fire and compared the results with a self developed field model. The model CFAST i.e. "Consolidation Model of Fire growth And Smoke Transport" was developed at NIST laboratory. The CFAST model divides the room(s) into a hot upper layer and a cool lower layer. The physical properties of each layer, such as gas temperature and species concentrations were assumed to be uniform. A tunnel has very large space though the ceiling height is usually not high. Chow commented that the equations for fire plume and ceiling jet employed in most zone models can be employed for this ceiling height. The simulation was done by dividing the tunnel into ten small compartments and compared the results with a self-developed field model. The vertical variation of temperature at steady state was used to compare the results. He found that average smoke temperature has been well predicted. The CFAST model only gives the mean temperature and species of upper layer and lower layer. Detailed distribution of temperature and species were not known. He recommended that zone model; CFAST can be used for studying probable fire environment in the tunnel. Later on Jones along with Matsushita and Baum (Jones, 1992) added the horizontal momentum equation to CFAST while studying smoke movement in corridors.

Altinakar et al. (1997) modified CFAST model and developed a multi-compartment, two-layer zone model called TUFISI (Tunnel Fire Simulation). In the CFAST model, mixing between the upper and lower layers was incorporated using an empirical mixing coefficient based on the local Richardson number. The frictional pressure losses along the tunnel were also added in the model. The TUFISI model was validated using measurements from the Ofenegg Tunnel tests and from natural and longitudinal ventilation tests conducted as part of the Memorial Tunnel Fire Ventilation Test Programme (MTFVTP).

Recently Suzuki et al. (2002, 2004) developed a new zone modeling approach called multilayer zone model (MLZ) to predict vertical distributions of temperature in a fire compartment. They used this model to simulate a tunnel fire and compared it with experiment. For simulation, the tunnel space was divided into a number of layers in vertical direction and regions in longitudinal direction. The physical properties like temperature and species (CO, CO2, etc.) are assumed to be uniform in every zone like two-zone model. The experimental study was carried out in small cylindrical tunnel equipment, the diameter and length being 0.3 m and 20 m respectively. The temperatures predicted by the model in the lower part of the tunnel were found to be lower than the experiments and the thickness of upper hot layer was found to be thicker by 0.05–0.1 m than experiments (Suzuki et al., 2004).

Xiaojun (2008) proposed modifications in the Suzuki's model for better modeling of the tunnel fires. He proposed modifications in the governing equations, radiation heat transfer models, flow rate through openings, etc. Suzuki's model assumed that the fire plume flow does not mix with the upper layer as soon as it penetrates the layers interface but continues to rise till it hits the ceiling, and all the heat released by the fire rises into the top layer in fire region. But Drysdale (1998) predicted that about 30% of the fire's heat was radiated back. So Xiaojun assumed that 70% (fuel dependent) rises to the top layer and other 30% was transferred to each layer by radiation. He introduced a four-surface radiation heat transfer model. The modified model predictions were compared with a full scale experimental results and CFD model-FDS (McGrattan, 2005) predictions. This model predicted better results (temperature and smoke propagation time) when compared with results of experiments and predictions of FDS. However, some discrepancies were observed in lower layer temperatures between model and experiment. He concluded that more studies should be done on MLZ and FDS model in the future.

2.3.3.1 Remarks

These models are based on significant simplifications and have limitations. These models are suitable for rooms where it can be assumed that the products of combustion cover ceiling instantaneously. However, this might not be the case in tunnels. It is also not certain whether these models can be used for tunnels having large lengths and cross-sections and to study the effects of vehicles and jet fans.

2.3.4 Field Models

Unlike the zone model where the region of interest is divided into two or three control volumes, in a field model, the region of interest is divided into large no of small volumes and the equations representing the conservation of momentum, energy and species concentration, etc. are solved at a point within each volume. The advantage of these models is phenomena such as combustion, radiation or turbulence can be incorporated in these models.

In addition, the graphical post processors allow visualizing important features of flow field.

However, this technique has its limitations also. A number of assumptions and approximations are to be made when defining the physics of flow problem which ultimately affects the accuracy of CFD simulations. For example, simplified models of combustion, turbulence and radiation are used in fire engineering applications. Numerical errors may arise as a result of using coarse computational grid.

One of the most important physical models that affects CFD predictions while modeling fire phenomenon is turbulence. Apart from this, the other parameters that influence CFD results are the effects of the fire itself, heat transfer between the fire, smoke, tunnel structure and objects within the tunnel. Most commercial CFD programs, for example Phoenics (CHAM), STAR-CD (CD Adapco), Fluent (Fluent), CFX (Ansys) etc solve Reynolds-averaged Navier-Stokes (RANS) equations. This approach does not resolve the turbulent motions but provides the time-averaged characteristic quantities of the flow. Therefore the grid need not be very fine. The grid should be fine enough to capture the important time-averaged features of the flow. A turbulence model is used that takes account of the effect of turbulence on the flow. Recently, another approach based on 'Large Eddy Simulation' (LES) technique is being used (Gao et al., 2004). This approach solves the Navier-Stokes equations for all except for the smallest turbulent motions. The finest eddies are either ignored or modeled. This approach therefore requires far more grid cells than the RANS approach. An example of CFD code using LES approach is 'Fire Dynamics simulator (FDS), developed by NIST of USA.

CFD technique for tunnel ventilation and smoke control is been widely used and many papers have been reported at conferences and in publications over the last few years. These papers focus on the capabilities and validity of the CFD modeling. The use of CFD for actual tunnel ventilation design or safety studies has also been the subject of few papers.

The earliest literature on CFD studies of tunnel fire problems refers to a unsteady two-dimensional model of a fire in a corridor developed by Ku et al. (1976). This model was modified by Brandeis & Bergmann (1983). The JASMINE code was developed and used extensively by the Fire Research Station, UK. Kumar & Cox (1986, 1988) developed and used JASMINE to validate test results of Zwenberg tunnel while Tuovinen et al. (1996) used it to compare experimental results from Ofenegg tunnel test No.1. As part of Phase IV of Memorial Tunnel Ventilation Test Program (Massachusetts Highway Department and Bechtel/Parsons Brinckerhoff, 1999), Bechtel/Parsons Brinkerhoff developed and validated SOLVENT code (Innovative Research Inc. / Parsons Brinckerhoff Inc., 2000). Some experiments of MTVTF have been validated by Vistnes (2004); Borello et al. (2002) and Miles et al. (1999). McGrattan & Hamins (2002) used FDS model to estimate the thermal environment of the Howard Street Tunnel Fire in Baltimore, Maryland, following the derailment of a freight train and the burning of spilled tripropylene and the contents of surrounding rail wagons. For this, FDS was validated against temperature data from the MTFVTP. Bettis et al. (1994c) and Lea (1994, 1995) validated HSL 'phase 2' and HSL 'phase1' tests respectively using FLOW3D. Recently Hu et al. (2007) validated the latest version of FDS, version 4.0 with four full-scale road tunnel tests. An important accomplishment of CFD was the simulation and validation of reduced-scale experiments done to investigate the King's Cross escalator fire (Simcox et al., 1992; Moodie & Jagger, 1992). Yuan & You (2007) used CFD to evaluate and optimize ventilation strategy of subway side-platform of Tianjin Metro, China while Yang and Lee (1999) investigated several design options for the emergency operation modes in newly-established

Taipei Railway Underground station. Schabacker et al. (2001) studied safety in a railway tunnel with natural ventilation system. The most comprehensive sensitivity studies on tunnel fires using CFD were conducted by Woodburn and Britter (1996a, b). Atkinson et al. (1996), Fletcher et al. (1994) and Rhodes (1994) studied smoke movement, temperature, and back layering phenomenon in tunnels. Rhodes (1996) investigated the use of CFD techniques in tunnel fire modeling. Grant et al. (1998) have presented a review on fires in tunnels by paying special attention to the determination of the critical velocity of back-layering. Chen (2000) reviewed the progress of research on smoke propagation in tunnels, paying particular attention to the characteristics of smoke propagation along the tunnel both up and downstream of fire. Kashef et al. (2003) reviewed the modeling of movement and behavior of smoke in tunnels. These and few other studies in ascending order of published year are reviewed and presented below.

Ku et al. (1976) developed a two-dimensional computer model for solving the transient flows induced by a fire in room / corridor geometry. Combustion was not modeled. The fire was represented by a volumetric heat source He compared the results with small-scale experimental configurations and found that they were in close agreement except at regions near the fire source. This was thought to be caused due to the coarseness of the computational grid used, inaccuracies in the turbulence model and the omission of radiation model.

Brandeis & Bergmann (1983) improved the above model by incorporating combustion model in it. Their model examined the effect of forced ventilation on the transient development of a hydrocarbon spill fire. Combustion was modeled assuming a single step reaction between hexane and air. The geometry of the model was chosen to be similar to the Caldecott Tunnel (because of fire that occurred inside this tunnel in 1982). The model domain taken was 100 m in length and 7.6 m in height and has a 5° slope. The domain was divided into 1575 cells. A parametric study was conducted to study effects of varying ventilation configuration, combustion reaction rate and external wind velocity. The study concluded that combustion affected the flow field more than the external wind and ventilation configuration. It was found that with increasing reaction rate, the volume of hot combustion products (smoke) increased. As external wind velocity is increased, the area of the flame front diminished, and was limited to the downstream side of the fire. Stratification was also observed. The effect of the tunnel slope was found to be negligible. The results were found to be qualitatively correct but quantitative conclusions could not be drawn due to the coarseness of the grid and other simplifications in the model.

The CFD code 'JASMINE' (Analysis of Smoke Movement in Enclosures) was developed at the Fire Research Station, UK. This model is similar in principle to that of Brandeis & Bergmann (1983), except that the JASMINE model is fully three dimensional. Kumar and Cox (1986) validated JASMINE with experimental data obtained in fire tests conducted in the Zwenberg tunnel (Feizlmayr, 1976) located in Austria. It was thought that the CFD technique may replace, in part or whole, the role of full-scale testing once sufficient validation data have been collected.

This model used buoyancy-modified k-ε turbulence model. The combustion of fuel was modeled as a one-step reaction between hexane and oxygen. The calculation of the combustion reaction rate incorporated the effect of turbulent mixing by using either the Spalding eddy break-up model (1971a) (steady-state calculations) or the Magnussen et al. model (1978) (transient calculations). The loss of heat due to convection and radiation were lumped together in a local empirical heat transfer coefficient. This approximation allows radiative heat transfer from the smoke to its surroundings but does not model radiative heat exchange between neighboring cells in the smoke layer. This refinement is included in a later study by Kumar & Cox (1988). The transient simulations showed that the temperature stratification and gas concentrations reached steady conditions around the same time that the heat release rate reached its steady-state (maximum) value. They measured the 80°C temperature contour during the fire tests to relate to potential thermal life hazard criteria. It was found that after two minutes, the stratification of the 80°C contour was in agreement with the experimental measurements.

They also conducted steady-state simulations for tunnel fires under natural and forced longitudinal ventilation conditions. The temperatures, CO_2, O_2 and fuel concentrations predicted by the model were found to be in reasonably good agreement with the experimental results except in the vicinity of the fire. This was attributed to the use of simple combustion model. Gas composition predictions were reasonable in the forced ventilation cases but poor for the naturally ventilated run. This disagreement in near fire region was attributed to grid being coarse, inaccuracies in the simple turbulence-chemistry interaction theory. However, the model was thought to be of sufficient accuracy to be useful for the study of the tunnel environment in the case of ventilated fires. The recommendations of the study were that an improved radiation treatment needed to be incorporated and that a time-dependent conduction equation was necessary at the solid boundaries.

Kumar & Cox (1988) then modified the JASMINE code by incorporating in it the six-flux radiation model and a wall function for rough surfaces (concrete block or brick face). The predictions from the improved, special purpose version of JASMINE known as TUNFIRE were compared to predictions made by JASMINE and experimental data obtained during fire tests conducted in the Zwenberg tunnel. It was found that there was not much improvement in the predictions. The authors suggested that this may be the result of using a coarse grid in the vicinity of the fire and a simple combustion model. In regions far away from fire source this was attributed to sensitivity of wall surface characteristics and boundary layer development.

One of the most acclaimed applications of computational fluid dynamics to a tunnel fire configuration is the HARWELL-FLOW3D simulations of the steady-state flow done by Simcox et al. (1992) to investigate the King's Cross escalator fire that took place on 18th November 1987. The aim of the computation was to ascertain the manner in which the smoke and combusting volatiles spread. To simplify the problem, combustion was not modeled and the fire was treated as a time-dependent heat source. Radiation effects were ignored and all solid walls were assumed to be adiabatic (no heat loss). Turbulence was modeled using standard k-ε model. The results were found to be in good qualitative agreement with the pattern of smoke damage to the ceiling. The temperature contours were found to be geometrically similar to the observed damage to the ceiling finishes. It showed that the method provided a good indicator of the path taken by flames and combustion products. The most important result of the simulations was the prediction of the 'trench effect' at the fire source. In this phenomenon, the flames touched the bottom of the wooden escalator which resulted in transferring of heat to the higher surfaces. This behavior prevented the early realization of the seriousness of the incident which finally led to a catastrophic flashover. This prediction was later confirmed by one-third scale tests carried out by the Health & Safety Executive (Moodie & Jagger, 1992).

A research project was launched by Woods of Colchester, Mott MacDonald, UK and the University of South Bank with the objective of understanding the aerodynamic characteristics of road tunnels ventilated by jet fans. The investigations were conducted on a series of single, centrally-mounted jet fans in a circular tunnel section. Armstrong et al. (1993) presented comparisons of the results of analytical, experimental and computational (CFD) smulations done using using FLOW3D (1990, 1991 and 1992). They found that CFD results and the results of analytical work, developed by Gasiorek (1991), showed excellent agreement with the experimental data.

Bettis et al. (1994c) used FLOW3D to simulate two tests of the one-third scale HSL 'phase 2' (i.e. obstructed geometry) tests (Bettis et al., 1993). The results were compared with those of experiments. To assess the use of CFD as a design tool, the CFD modelers were not made aware of the experimental behavior in advance. The CFD results were found to be qualitatively correct but quantitatively inaccurate. The critical velocity (V_c) was found to be under predicted for a given fire size (Q). Bettis et al. (1994b) also expressed reservations to the use of Froude modeling in tunnel fire experiments. They concluded that CFD was still an immature technology and can not be used as a design tool. On the other hand Lea (1994, 1995) conducted simulations of the HSL 'phase I' (unobstructed geometry) tests (Bettis et al., 1994a) using

FLOW3D. Radiation was not modeled and buoyancy-modified k-ε turbulence model was used. It was found that the CFD predictions agreed qualitatively with the HSL trials. Again, the critical velocity (V_c) was found to be under predicted for a given fire size Q. He suggested that this may be because of significant blockage created by the fire plume and subsequent tilting of the plume in the downstream direction, reducing the tendency for upstream smoke propagation. The effect of tunnel aspect ratio (H/W) and the fire size Q on V_c was also studied. It was found that as the tunnel width increased, V_c became more dependent on Q and that V_c increased with width (W) for a given value of Q. He reported that this behavior was contrary to the results from simpler modeling approaches.

Fletcher et al. (1994) compared the results of a CFD model developed at the University of Sydney, to experimental data for pool fires conducted in a ventilated tunnel. The CFD model included buoyancy-modified k-ε turbulence model and incorporated both combustion (mixture fraction concept) and radiation (discrete transfer method) sub-models. The experiments were performed in a 130 m long, 5.4 m wide and 2.4 m tall tunnel. It was found that the back layer predictions were in good agreement with the experimental results. However, the downstream temperature was over predicted. The authors concluded that temperature stratification and smoke back layer were highly sensitive to turbulence and soot radiation modeling.

Rhodes (1994) carried CFD simulations to investigate smoke movement and temperature profiles resulting from prescribed fire scenarios in tunnels. Three different tunnels were modeled, one level road tunnel, and two railway tunnels with gradients of 0.9 percent and 1.5 percent. The fire produced heat release rate between 5 and 50 MW. Simulations were carried under natural and longitudinal ventilation conditions. The results were compared with behavior of the simulated smoke layer of tunnel fire experiments found in the literature. The author concluded that both the experimental results and the CFD model demonstrated that in the absence of longitudinal ventilation, smoke would stratify. Longitudinal ventilation caused mixing and forced the smoke layer to de-stratify. The stability of the smoke layer is not affected by tunnel gradient.

Woodburn & Britter (1996a, b) used FLOW3D to model one of the HSL experiments and carried out sensitivity studies in region around the fire and in region downstream of fire. This study is perhaps the most complete CFD studies so far reported on tunnel fires. In the fire area, the back-layering phenomenon was investigated. In the downstream area, smoke propagation along with ventilation was investigated.

In region upstream of fire, they concluded that ventilation velocity profile, HRR of fire, and turbulence model is required for simulation. They found that the use of a partially developed ventilation velocity profile rather than a uniform profile led to a longer back-layering length. They employed both standard and buoyancy-modified k-ε turbulence model. They found that the modified turbulence model accurately predicted the extent of the upstream layer whereas the standard k-ε model predicted zero back-flow. The value of back layer length was found to be sensitive to variations in heat release rate.

The downstream flow regime was found to be independent of the turbulence model selected, instead it was most sensitive to variations in the prescription of natural convective and radiative heat transfer and also to the wall roughness. They also indicated that the wall temperature at downstream boundary was insensitive to the roughness of the wall, contrary to the results of Kumar and Cox (1988).

Atkinson et al. (1996) studied the critical longitudinal air velocity required to control the upstream motion of combustion products and smoke during a tunnel fire. They conducted experimental tests at the Health and Safety Laboratory (HSL) and compared with FLOW3D predictions. Tests were performed in a 366 m long, 2.5 m high arched tunnel with fires ranging from 0.3 MW to 20 MW. The CFD model used buoyancy-modified k-ε turbulence model along with eddy break-up combustion model. Radiation was not modeled. The results from the tests indicated that the critical longitudinal velocity did not necessarily depend on heat release when the latter became large. CFD simulations also showed the same behavior. The critical velocity predicted was found to be lower than those suggested by the experimental data. This was thought to be caused due to turbulence model employed. It was also found that back layer length was under predicted in case of the obstructed fire tests. The back layer length in case of unobstructed tunnel was found to be sensitive to small changes in the imposed velocity field.

Tuovinen et al. (1996) conducted numerical simulations, using two different CFD codes, JASMINE and FLOW3D, and compared their results with the experimental results from Ofenegg tunnel test No.1. They conducted a parametric study of fires inside an arbitrary tunnel of 300 m length using JASMINE. The parameters studied were fire size, tunnel width, ventilation and tunnel grade. They investigated the evacuation procedures inside the tunnel in the event of a fire by analyzing the movement of the temperature and smoke region as a function of time.

In the JASMINE simulation, buoyancy modified k-ε turbulence model and Magnussen's "Eddy Dissipation Concept" combustion model were used. Heat losses to the walls were modeled using a lumped heat transfer coefficient that accounted for both radiation and convection. The FLOW3D simulations incorporated similar sub-models as JASMINE except that the tunnel walls were considered smooth and the fire was modeled as a volumetric heat source. It was found that the temperature predictions of JASMINE showed good agreement with the experimental results near the fire source while at distances away from fire source, FLOW3D predictions yielded better results. The authors reported, "the precision of the predicted values was sufficiently realistic to allow the model to be used for fire safety engineering design."

The parametric study showed that the fire growth rate and width of the tunnel had the largest impact on the possibility of escape. In case of sloping tunnels, it was found that the evacuation of people uphill from regions near the fire was difficult. They also found that the natural vent shafts helped in smoke extraction and recommended the use of shafts every 100 m in the road tunnel for clearing smoke. They also recommended that escape routes parallel to the road tunnel should be provided and interconnected at short intervals of 100 m.

Rhodes (1996) investigated the use of CFD techniques in tunnel fire modeling. The author noted that it is difficult to validate CFD codes against full-scale experiments because of excessive precision required to instrument large-scale fire tests so that these can provide useful results for validating. He therefore recommended that parametric studies should be performed to test for the sensitivities of assumptions. In this way a "reasonable" correlation with experiments could be achieved. He considered the problem of predicting the critical longitudinal air velocity in an arbitrary tunnel. Simulations were performed and the critical velocity predicted by the numerical model was compared to the values calculated from established empirical equation. The predictions made by the numerical model were concluded to be in approximate agreement with the empirical model.

Yang and Lee (1999) investigated several design options for the emergency operation modes required to maintain a "smoke-free" escape route condition for the passengers and firefighters in a newly-established underground railway station, the Taipei Railway Underground. The entire emergency procedure was tested for compliance with NFPA 130 (1995) design criteria. The investigation was carried out using the 1 D Subway Environment Simulation (SES) computer programs (DOT, 1997). The simulation results were validated by a full-scale hot smoke test with a 16 MW fire. Different modes of ventilation were tested and investigated numerically. Yang and Lee reported that the measured airflow velocities correlated well with the SES simulation results at some stairs and deviated significantly at other spots. They attributed the deviations to the fact that the SES program is 1 D. They recommended using a 3D CFD simulation in these cases to ensure accurate results.

The Memorial Tunnel Fire Ventilation Test Program (Massachusetts Highway Department and Bechtel/Parsons Brinckerhoff, 1996), described in section 2.2.1 of this chapter, consisted of a series of full-scale fire tests conducted in an abandoned road tunnel near Charleston, West Virginia. Different tunnel ventilation configurations (pure longitudinal and transversal ventilation system along with mixed configurations) were analyzed to assess their performance in managing the smoke and temperatures produced by the various design fires within the tunnel.

As part of Phase IV of The Memorial Tunnel Ventilation Test Program (MTVTP) (Massachusetts Highway Department and Bechtel/Parsons Brinckerhoff, 1999), Bechtel/Parsons Brinkerhoff (PB) teamed with Innovative Research, Inc. (IRI), a CFD vendor, to develop and validate a customized CFD code specifically for tunnel application. The code developed was SOLVENT (Innovative Research Inc. / Parsons Brinckerhoff Inc., 2000). They represented fire as a volumetric heat source. The details of combustion process were not modeled. The model used buoyancy-augmented k-ε turbulence model along with six-flux radiation model. The validation concluded that for the longitudinal ventilation system:

- The CFD model was able to closely predict the airflow generated by a jet fan ventilation system.
- The effects of the fire in the far-field region of the tunnel (more than 61 m away from the fire) were well predicted while the predictions in the near-field region were not in agreement with the test data.
- The bulk flow predictions upstream and downstream of fire were within 10 – 15 % of the measured values.
- The temperature and velocity vertical profiles correlated reasonably well with the test data except for the case of 100 MW fire test,
- The model predicted the occurrence of a steady back layer condition. The extent of the backlayer was found to be sensitive to the heat release rate. The depth of the back layer below the ceiling was over predicted.

Detailed conclusions for various ventilation configurations studied are given in Memorial Tunnel Fire Test Ventilation Program Comprehensive Test Report.

Miles et al. (1999) validated the CFD fire model TUNFIRE against Tests 607, 610 and 615B from the MTFVTP. These tests involved longitudinal ventilation and pool fires of 20, 50 and 100 MW respectively. They made comparisons between predictions obtained using different computational grids, highlighting that agreement with experimental results depends on the modeling strategy used and experience and expertise of the modeler.

Chen (2000) reviewed the progress of research on smoke propagation in tunnels, paying particular attention to the characteristics of smoke propagation along the tunnel both up and downstream of fire. He describes a gravity current approach (GCA) to predict the smoke propagation behavior in tunnels. The GCA approach is based on the fact that the smoke propagates with a constant speed, like the movement of gravity current above a flat surface. This approach is compared to CFD approach and also with Froude number preservation approach (FNPA). The GCA approach showed that the propagation speed of smoke in both up and downstream of fire is constant in contrast to result of FNPA approach which shows that in the region upstream of fire, the critical ventilation velocity is generally proportional to the one third power of the heat release rate (HRR). The results from GCA approach of downstream smoke compared favorably with the CFD results. Chen concluded that for GCA approach to be used successfully sufficient knowledge of the flow characteristics should be known a priori.

Hwang and Edwards (2001) used a CFD program 'CFD2000 by Adaptive Research' to model floor-level fires in a ventilated tunnel. The fire is simulated by a diffusion flame of propane (mass fraction unity) issuing from a circular hole on the tunnel floor. To realistically simulate the burning process, the fuel reacts with air in a 4-step finite rate reaction scheme. The standard k-ε turbulence model is used. They studied re-circulating flow patterns, movement of the ceiling-layer front, and distributions of gas temperature and velocity under various fire parameters. The result of computations showed that profiles of the gas velocity and temperature along channel cross sections are similar to those experimentally observed. It was found that finite-rate reaction scheme used resulted in very long computations. They recommended use of simpler reaction scheme (such as the mixture-fraction model). They also showed that CFD analysis can be used to determine a correlation between smoke reversal length and tunnel ventilation and thus can be used as a design tool for fire protection.

Schabacker et al. (2001) studied safety in a railway tunnel with natural ventilation system. Their study investigates conditions within the tunnel near a burning train. They employed two different approaches: 1-D analysis using the code SPRINT, developed by HBI, and 3D CFD analysis using the commercial code FLUENT. The predictions of a one-dimensional simulation of the time - dependent smoke and temperature distribution near a burning train are compared with the results obtained from a three-dimensional CFD simulation. They found that the 1-D approach gave correct predictions of the relevant air-quality parameters in the tunnel and the escape conditions of the passengers. They further concluded that for more complex tunnel geometry, including for example cross-passages or rescues areas, 3-D CFD should be employed because it will lead to more comprehensive and reliable results and also the high risk regions will only be revealed by the 3-D CFD simulations.

Borello et al. (2002) conducted CFD analysis of flows in tunnels with longitudinal ventilation in normal and critical operating conditions. The fire simulation deals with the 10 MW T606A experimental test-case of Memorial Tunnel Fire Ventilation Test Program. Combustion is modeled as a volumetric heat source model. A comparative study of two radiation approaches, one is radiation modeling using IMMERSOL model (Spalding, 1971b) and second is neglecting effects of radiation but reducing HRR by 30% to account for radiation losses was also carried out. The predicted results were found to represent the main flows characteristic such as temperature and velocity fields. The back-layering effects were also predicted. It was also found that modeling of radiation using radiation model gives better values of temperature and velocity near the fire.

Vistnes (2004) validated Phoenics 3.5 using the Memorial Tunnel Fire test data. Test 501 of the MTFVTP was used for validation which examined the effectiveness of natural ventilation in ventilating a 20 MW fire. The CFD model used k-ε turbulence model and incorporated buoyancy effect on turbulence. Radiation was not modeled; instead the radiative component of the heat release rate (30%) was deducted from the total heat release rate. The study concluded that adiabatic conditions specified for walls and solids resulted in gross over-predictions and therefore recommends using fixed temperature conditions for walls and solids within the domain. He found that after a time of 10 minutes, the predicted conditions within the tunnel were in good agreement with the test data while for earlier times the spread of the hot layer was under-predicted. This under prediction was thought to be caused due to the uncertainty in the heat release rate measured in the fire test.

Gao-shang et al. (2006) analyzed the characteristics of tunnel fire using commercial CFD software PHOENICS 3.5. They studied spreading rules, characteristics of concentration field and temperature field of smoke flow under different longitudinal ventilation speeds. The tunnel section considered is similar to the 7 km long Xue-feng Mountain Tunnel, the second longest tunnel in China. The average space between the cross passages is 250 m and they therefore used a 300 m section for analysis. The fire source considered was 20 MW, equivalent to HRR of passenger car. Both Steady and transient simulations during fire in a tunnel were carried out. They found the longitudinal critical velocity to be 3 ms^{-1}. They also found that the initial temperatures were highest on the side wall which then spreads to the middle of the tunnel and so they concluded that the passengers should evacuate along the midline of the tunnel. They also concluded that smoke movement from the beginning to the growth can be obtained from transient simulation of the tunnel fire which provides the theoretical background for evacuation design in tunnel fire.

Hu et al. (2007) validated the latest version of FDS, version 4.0 (McGrattan & Forney, 2005) developed by the National Institute of Standards and Technology (NIST), USA with four full-scale road tunnel tests. The tests were conducted in YuanJiang Road Tunnel in China. The experiments were conducted with two sizes of pool fires, 1.8 and 3.2 MW, with two different fire surface heights, 0.2 and 1.7 m from the floor level. Longitudinal ventilation velocities were also varied. The temperature distribution of smoke ceiling jet flow, upstream and downstream of the fire, below the ceiling was measured. It was found that that the temperatures predicted by FDS 4.0 were near to the measured data. The temperature predicted was found to be very close to the measured value in regions near the fire. The back-layering length predicted by FDS 4.0 also seemed to agree fairly well with that deduced from the experiments. However, it should be noted that these results are valid for relatively small tunnel fires. For a medium fire such as a bus fire or for a large fire such as a burning heavy good vehicle (HGV), the configurations of flame, plume, and ceiling jet are different from that of a small fire.

Recently Migoya et al. (2008) developed a simplified hybrid model (UPMTUNNEL) for the simulation of accidental fires in road tunnels with longitudinal ventilation. This model has characteristics of both field and zone models. The tunnel was divided in two zones: the plume, located upstream from the point at which the smoke hits the ceiling, and a diffusion zone extending downstream. They analyzed both these regions assuming steady-state conditions. The plume was described by one-dimensional conservation equations for turbulent flows. The results were then compared with two different 3D models (CFD codes) and full-scale experiments. The CFD codes used were FLUENT and PHOENICS, in which the eddy break-up and k-ε-g turbulent combustion models have been used respectively. The experiments were performed in a 2605 m long longitudinally ventilated tunnel located under a runway of Madrid-Barajas airport. They found good agreement between the results of the UPMTUNNEL model and those of the three dimensional models and full-scale measurements. They also found that UPMTUNNEL results were in better agreement with experiments than the 3 D models. However this model could not predict backward movement of smoke. It was concluded that this can calculate, for a given fire power and ventilation speed, the downstream fire conditions, and therefore estimate the safety of the people and tunnel equipment in downstream region of the fire. This methodology was used to design the ventilation control system in the Lorca tunnels in the Southeast of Spain.

2.3.4.1 Remarks

The studies reviewed above indicate that CFD is being widely used to investigate flow in tunnel fires, design of tunnels and conducting safety studies. The commercialization of the CFD codes, such as JASMINE, FLOW3D, PHOENIX, FLUENT, FDS and availability of fast computers has enhanced the power of computation. It has been found that though quantitative accuracy has varied but good qualitative behavior has been achieved from CFD programs. The accuracy of CFD results is found to depend on the following:

- Use of appropriate sub models like combustion, turbulence, radiation, buoyancy etc.
- Validity of the model i.e. the ability of model to predict the real scenario.
- Expertise of the modeler required for setting up and running the model.

The studies reviewed above indicate that reliable CFD results can be obtained if the following points revealed by these studies are carefully treated:

- The flow in tunnel fire is essentially two-dimensional in both up and downstream of fire, while in the fire region, the flow is three-dimensional.
- In the fire region, the flow is fully turbulent and is influenced by radiation, turbulence and combustion models used. The inclusion of complex combustion models increases computation time. For regions away from fire source, combustion may not be modeled; instead the fire can be represented as a volumetric heat source model. Radiation should not be

ignored. If radiation is not been modeled, than at least fire heat output should be reduced to account for radiative loss.

- In region upstream of fire, the buoyancy modified k-ε turbulence model predicts well the extent of the upstream layer. However, standard k-ε turbulence model is found to be sufficient to predict downstream region.
- The ventilation velocity profile influences the length of smoke in upstream region.
- The boundary condition at the exit portal and the wall roughness has negligible effects on the adiabatic smoke propagation in downstream region.
- The heat loss to the walls has an important effect on smoke movement. A conservative approach is to assume no heat loss at walls i.e. an adiabatic condition. This approach describes the worst case scenario.
- The smoke movement behavior from the beginning to the growth can be obtained with transient simulation of the tunnel fire. These results can help in designing evacuation means and estimating evacuation time.
- In most of the studies, the tunnel shapes and sizes vary considerably and so do the fire sizes. Caution should be exercised in comparing results obtained under dissimilar geometries and dissimilar fire scenarios.
- Instead of full tunnel length, only a section of the tunnel can be simulated to optimize the cost of computations.

It is clear that a number of assumptions are made while using CFD tool and the user should be familiar with the various physical models, boundary conditions to be used in a particular flow problem before its use. It is then only the user can get meaningful results from the CFD programs and that too only after carefully treating the various points mentioned above. In view of above, the next section describes in brief the basics of CFD along with various sub models used in Fire Safety Engineering.

2.4 CFD MODELING APPROACH

As described earlier, a CFD model requires that the enclosure of interest be divided into small rectangular control volumes or computational cells. The CFD model computes the density, velocity, temperature, pressure and species concentrations in each control volume based on the solution of conservation laws of mass, momentum, and energy at discrete points in time and space.

This section explains the basics of CFD technique and outlines various sub-models used in fire safety engineering as guided by Gobeau et al. (2002). General books on CFD, like those written by Ferziger and Peric (1996), Veersteg and Malalasekera (1995) and Anderson (1995) provides good reading material on CFD techniques.

2.4.1 Overview

CFD can simulate fire and smoke movement in three-dimensions, for steady state and time-dependent applications. The set of equations, which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These are partial differential equations (PDE's), which have no known general analytical solution but have to be solved numerically.

The governing equations for the flow field used in any CFD code are mentioned below (AEA Tech., 2002).

Mass conservation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j\right) = 0 \tag{2.21}$$

Momentum conservation

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_j u_i\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho B_i \tag{2.22}$$

Energy conservation

$$\frac{\partial}{\partial t}\left(\rho h\right) + \frac{\partial}{\partial x_j}\left(\rho u_j h\right) = \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j}\left(\frac{\lambda}{c_p}\frac{\partial h}{\partial x_j} - \dot{q}_j^R\right) \tag*{2.23)}$$

Species conservation

$$\frac{\partial}{\partial t}\left(\rho Y_\alpha\right) + \frac{\partial}{\partial x_j}\left(\rho u_j Y_\alpha\right) = \frac{\partial}{\partial x_j}\left(\rho D_i \frac{\partial Y_\alpha}{\partial x_j}\right) + S_\alpha \tag{2.24}$$

Equation of state

$$p = \rho R T \tag{2.25}$$

where

ρ represents density, u_i represents gas velocity in x_i direction, τ_{ij} represents stress tensor, B_i represents body forces in x_i direction, h represents enthalpy, λ represents heat conductivity, C_p represents specific heat capacity at constant pressure, $\dot{q}_j^R$ represents heat flux due to thermal radiation, Y_α represents mole fraction of species α, and S_α represents source term in chemical species conservation term.

The steps involved in undertaking a flow analysis using CFD code are as follows:

- Geometry and domain creation
- Selection of physical sub-models
- Specification of boundary conditions
- Discretisation of the mathematical equations
- Monitoring the iterative solution process
- Analysis of the solution obtained

2.4.2 Geometry and domain creation

The first step is to create a geometrical model of the problem to be solved.

This implies, to represent by a set of surfaces the confines of the space, i.e. the walls of rooms, corridors, staircases, fixtures and fittings, furniture, etc.
The geometrical model specifies the 'computational domain' i.e. it defines the region to be modeled. The computational domain is so chosen that the flow conditions are known inside it. In some cases, where the external flow may influence the flow at the boundary of the domain, the domain has to be extended beyond the actual geometry of the model. For example, surrounding buildings might need to be included when interested in computing the internal flow of a naturally-ventilated building. Examples of a computational domain are a room, a floor or an entire building.

For faster convergence, it is necessary that the computational domain is kept as simple as possible. It thus becomes necessary to decide how much of the system detail to be included and whether there is a need to include or simplify the fine details, such as furniture in the model. If fixtures and fittings have marginal influence on the flow behavior, then it is better to exclude these from the domain. For example, while modeling underground station, furniture can be excluded and staircases can be represented by inclined planes.

2.4.3 Selection of physical sub-models

The fluid flow equations mentioned above cannot be solved directly. At present there exist certain gaps in understanding of important physical processes like turbulence, kinetics, combustion, radiation etc. Therefore certain assumptions in the form of simplified physical sub models are used. The simplified physical processes are represented by a set of approximate equations, derived either from empirical formulae or from physical reasoning.

Common examples of sub-models relevant to fire applications include turbulence models, radiation models, combustion models, buoyancy and heat transfer to walls. These sub-models are then incorporated in the simulation. These sub models defines physics in a simpler manner thereby reducing the number of equations to be solved and hence decreases the simulation time. A brief description of these physical sub models are presented below:

2.4.3.1. Combustion Models

Heat is released in fires through the complex combustion processes. There are two approaches to model the distributed heat release from a fire, namely the volumetric heat source model and various combustion models.

In volumetric heat source model (VHSM), the fire source is represented by a volumetric source term in the governing equation for enthalpy i.e. this model uses a volumetric energy source for the heat release rate. This approach does not predict the combustion process; instead heat is assumed to be uniformly distributed over a prescribed volume. Therefore the size and shape of source is to be defined in such a way that these correspond to the expected characteristics of the flaming region in which combustion occurs.

However, in reality, the distribution of heat and smoke produced by combustion is not uniform. Also the shape and extent of the volume is often unknown. Heat and smoke result from chemical reactions between the fuel and air. The chemical reactions are influenced by their quantities. These reactions depend on the quantities already consumed by the chemical reactions, as well as on the flow characteristics, including turbulence, which drive the fuel-air mixing. Ventilation also affects mixing in the combustion zone and thus the trajectory of the fire plume. The presence of surfaces such as walls also affects plume trajectory.

To model the above phenomenon, a number of combustion models have been developed which solve additional transport equation for the concentration of reacting and inert gaseous species. The combustion models gives more realistic representation of the fire source since it accounts for turbulent mixing of fuel and air, followed by their chemical reaction and heat release in the fire plume. Most of these models assume an instantaneous diffusion controlled single, one-step reaction between fuel and oxidant. The reactions take the form: fuel + air ---> product. More elaborate schemes include chemical kinetics effects (Drysdale, 1998) by employing a combination of an Arrhenius expression and some formulation describing turbulence interactions.

The most commonly used combustion model is the 'eddy-break-up' model (Spalding, 1971b) which assumes an instantaneous reaction - the rate of which is proportional to a computed turbulent time-scale. Since the model considers a single step reaction, hence it cannot model the numerous competing chemical reactions occurring in reality. During combustion it assumes that the fuel is completely oxidized. Thus, it does not provide an accurate prediction of rates of products including smoke. Another model is the 'laminar flamelet' representation of combustion chemistry (Carlsson, 1999). This method separates the combustion model from the turbulence model. This assumes that the turbulent diffusion flame can be described to consist of small, microscopic elements that have a structure similar to undisturbed laminar flame. These small elements are called flamelets. The chemical reactions are assumed to be fast. These assumptions are used to write chemical species concentration, thermo-chemical scalars such as enthalpy, temperature etc. in terms of one single conserved scalar, the mixture fraction. This has been successfully applied to simple gaseous hydrocarbon flames. Xue et al. (2001) have compared different combustion models in enclosure fires.

Gobeau and Zhou (2004) compared both these approaches for a fire which started in a large open area in a building under construction. They found that the predicted transport of smoke were similar. This is in agreement with the findings of Xue et al. (2001), but not with those of Kumar and Cox (2001). This is believed to be due to the different characteristics of the fires investigated: Kumar and Cox modeled a fire located in a doorway - the plume of which was deflected by the jet of air going through the doorway. The combustion model successfully reproduced the tilted fire plume whilst the volumetric heat source approach imposed a straight plume and as a result failed to predict the distribution of temperature and air entrainment rates into the plume. Kumar and Cox (2001) also stress the importance of defining an appropriate fire area in the case of a combustion model.

Gobeau and Zhou (2004) also investigated the sensitivity of the volumetric heat source model to the specification of the plume volume, namely its height and shape. They found that predictions were different near the fire source while the differences were insignificant further away.

2.4.3.2 Buoyancy

In fire modeling where the flows are accompanied by heat transfer, the fluid properties, in particular density, are functions of temperature. There are two approaches commonly used. One is to assume constant density, also called Boussinesq approximation and another approach is to use a compressible model which calculates air density using the perfect gas law. The

Boussinesq approximation assumes a constant density in the most of the terms of the momentum equations and treats it as a variable only in the gravitational term where it assumes that it is linearly dependent on the temperature. This approach is valid for very small temperature gradients, of the order of a few degrees or a maximum few tens of degrees Celsius. This model is sometimes used to predict the transport of smoke from a fire in large volume spaces. This is done because it is assumed that this assumption will be valid in most parts of the domain except in the vicinity of fire and that the difference in the end result will therefore be small.

2.4.3.3. Turbulence Models

Several approaches have been proposed to model turbulence which can broadly be classified into three categories:

(i) Direct Numerical Simulation (DNS)

In this approach all the turbulent motions are resolved by solving directly the Navier-Stokes equations that govern fluid flows, without additional modeling. This approach requires a very large number of grid cells and is thus impractical for industrial applications. It is therefore not used in Fire Safety Engineering.

(ii) Large Eddy Simulation (LES)

Here all but the smallest turbulent motions are resolved by the Navier-Stokes equations. The finest eddies are thus either ignored or modeled. Although this approach allows one to employ fewer grid cells than DNS, yet it is considered impractical for fires and smoke movement in complex spaces. It is recently been applied to fires but is limited for the moment to simple scenarios.

(iii) Reynolds-Averaged Navier-Stokes (RANS) Models

The Navier-Stokes equations are time-averaged and thus the equations so obtained do not aim to resolve the turbulent motions but provide the time-averaged characteristic quantities of the flow. Since the turbulent motions are not resolved in this model, hence it is not necessary to have very fine grid. The grid should be fine enough to capture the important time-averaged features of the flow. It is therefore less computational expensive than the above two models, hence is widely used in Fire Safety Engineering.

The turbulence models most likely to be employed for fire and smoke movement in complex spaces are therefore of RANS type. Amongst these, the most widely used is k-ε model, in which the kinetic energy of turbulent fluctuations (k) and rate of dissipation of turbulence energy (ε) are solved for each computational domain using partial differential equations.

The limitation of RANS models lies in the modeling of the turbulent Reynolds stresses: unknown terms appearing in the equations as a result of the time-averaging process. The k-ε model is based on the assumption that the Reynolds stresses are linearly related to the local mean strain rate. This assumption is, however, not strictly valid for the following situations: buoyancy, streamline curvature, acceleration/deceleration, impingement and three-dimensionality (Leschziner, 1992) which are obviously all features of fires in enclosed spaces.

Therefore, modifications were suggested to the k-ε model to take into account the buoyancy effects on turbulent mixing, for example Rodi (1980) added additional terms related to the buoyancy forces to the k and to the ε equations to correct the behavior of the model. Woodburn and Britter (1996 a,

b) proposed modifications to the k-ε model to account for effect of buoyancy and wall damping on the turbulence. They added a buoyancy-related term in the equations of the model so that the resulting computed Reynolds stresses behave more as expected in the presence of buoyant forces.

The equations employed in a k-ε model uses a set of constants which have now well-established values. Though these constants have been determined empirically by tuning against experimental data in simple specific configurations, they have often proven acceptable for a wide range of flows.

This model is applicable to flow or flow regions with high turbulent Reynolds Number and cannot be applied to near walls where viscous effects become dominant. For places near walls, where viscous effects dominates two approaches are commonly used namely The Wall Function method and The Low-Reynolds-Number method. The Wall Function method uses empirical formulas that impose suitable conditions near to the wall without resolving the boundary layer, thus saving computational resources. This assumes that at wall distance y, outside viscous sublayer, the velocity components parallel to the wall follows logarithmic law of the wall and turbulence is in local equilibrium so that production of turbulence/turbulent kinetic energy becomes equal to dissipation of turbulence energy (Mankbadi, 1995). In the Low-Reynolds-Number method calculations are carried out right up to the wall.

2.4.3.4. Radiation Models

Radiation accounts for a large percentage of heat release from the fire source. Radiation thus plays an important role when temperatures are above 600 K. The main sources of radiation are CO_2 & H_2O, both of which emit energy in discrete bands, and soot, which emits radiation at all wavelengths. Radiative heat transfer occurs between the emitters and receivers, i.e. between solid surfaces, soot / gas phase mixtures of flames and smoke aerosols.

There exists numerous models to define radiation, however the ones that are common and are frequently used are:

- Fractional heat loss due to radiative heat transfer to the surroundings
- Radiative heat transfer at the walls
- Six flux modelling
- Discrete Transfer modelling
- Monte Carlo simulations

These models, except the first one, provide a prediction of the radiative heat transfer by solving a set of differential equations. The value so obtained is then set as a source term in the transport equation for temperature, more precisely the enthalpy and energy equation.

(i) Fractional Heat Loss

This method assumes that certain percentage of heat (approximately 20 - 35%) is lost due to radiation. This means that the heat output of the fire in the model is set as the remaining (convective) percentage of heat.

(Ii) Radiative Heat Transfer at the Walls

This method accounts for the radiative heat transfer between the smoke and the walls. This transfer is expressed as a function of the temperature of the wall & smoke and the emissivity of the smoke. It is applied between the wall and the fluid cell next to the wall.

(iii) Six Flux Modeling

This model assumes that participating medium is gray and employs spherical harmonic P1 approximation for solution of Radiative Transfer Equation

(R.T.E). This method, assumes that the radiant flux across each of the faces of the grid cells is uniform. This simplifies greatly the set of equations to be solved to calculate the radiative source term. The accuracy of six flux models is highly directionally dependent; the radiation is assumed to be transmitted along the co-ordinate axis only.

(iv) Discrete Transfer Modeling

This model tries to solve the discrete radiative rays. Only representative rays are solved. The directions of the rays are specified a priori. The solution for any particular ray is restricted to the path between two boundary walls rather than being partially reflected at walls and being tracked to extinction. The accuracy of the discrete transfer model is dependent on the ray directions chosen as well as the number of rays. The discrete transfer method is not ideally suited to the body fitted grids likely to be seen in fire and smoke movement applications in complex spaces. This is because it is computationally expensive, especially for situations where a large number of rays are required to obtain an accurate solution.

(v) Monte Carlo Simulations

A number of rays are "emitted" in (pseudo-) random directions. The rays are then traced until they hit an obstacle, hit a wall, or disappear out of the computational domain. The quality of the heat transfer calculations is dependent on the number of rays. The computational cost is high for flows because a large number of rays are required in order to obtain accurate solutions. It is therefore fairly unlikely that such a method will be applied to fires and smoke movement in large complex spaces.

The accuracy of radiation models is dependent on the knowledge of absorption and scattering coefficients of the fluid medium and emissivities of solid surfaces. The emissive powers and absorptivities further depend on the composition of the soot/gas mixture.

It has been found that while describing fire as volumetric heat source model, the simplest method of accounting for radiation loss is using the Fractional Heat Loss method. This approach however only accounts for the radiative loss of the flaming region and ignores other radiative heat transfer, in particular the transfer from hot smoke to walls and the transfer within the smoke. Woodburn & Britter (1996a, b), simulated a fire in a tunnel and found that including radiative heat transfer at the walls improved the agreement between the predicted temperatures with the measurements in the hot gas layer, although the model ignored the radiative transfer within the gas. However, employing more sophisticated approaches to account for radiation in combination with a volumetric heat source model is unlikely to increase the accuracy of the results due to the limitation of this model which assumes a uniform heat distribution in the flaming region.

Any radiation model can be combined with a combustion model. Kumar and Cox (2001) compared an eddy-break-up model with and without radiation. The radiation model used was a simple six-flux model with a grey gas assumption to calculate the emissive power of hot gases. This model takes into account the radiative heat transfer within the gas. Its inclusion generated a redistribution of the heat energy within the gas and consequently the radiation model gave better agreement with experimental data.

2.4.4 Specification of boundary conditions

Boundary condition implies defining any mechanism which is external to the computational domain, but which influences the behavior of the flow inside computational domain. These include:

- The flow that enters or leaves the domain at openings such as doors, windows or vents, portals;
- The transfer of mass, momentum and heat at walls;
- The source or sink of mass, momentum or heat due to an external event, example fire.

There are various types of Boundary conditions which are as follows:

- Dirichlet boundary conditions: These prescribe variable values, for example these specify velocity and temperature as flow enters the computational domain;
- Neumann boundary conditions: These prescribe gradients of variables. These are used, for instance, to set a plane of symmetry in the domain by specifying that gradients of all flow variables normal to the plane are zero;
- A combination of both previous methods;
- Volumetric sources or sinks of heat, mass and momentum. For example fire can be represented by volumetric heat source approach.

Two other physical considerations are essential while defining boundary conditions. These relate to the wall friction and wall heat transfer. A wall function is applied at the walls which supplies the shear stress and near wall turbulence, and heat transfer coefficient. These functions accounts for the roughness of the walls and lead to lower velocities in the case of rough walls.

The wall temperature may be assumed to be constant during the transient, or further modeling into the wall is carried out to predict the temperature rise at the surface. Thus two extreme possibilities exist to model heat transfer at the walls. One possibility is to assume zero heat transfer i.e. an adiabatic wall. The other possibility is to assume a constant wall temperature i.e. maximum rate of heat transfer. The adiabatic wall condition implies a faster rate of lateral smoke transport and can be used as a conservative estimate.

It is not necessary that all of the required boundary conditions are well-defined. For example turbulence parameters as flow enters the computational domain may be unknown; uncertainty may exist in wall heat transfer coefficients; fire sources and fire growth rates may be ill-defined; events external to the selected computational domain, such as pressure distributions arising from natural or forced ventilation may affect flow inside the domain.

2.4.5 Discretisation of the mathematical equations

The set of equations obtained cannot be solved analytically because the equations are highly non-linear and strongly coupled. These equations are to be solved numerically. This is achieved by a discretisation method. A discretised equation is an algebraic relation connecting the values of variable for a group of grid points. The discretisation of the differential conservation equations involves relating the values of quantities such as fluxes across the faces of the cells, to the unknowns of the problem, which are typically the variable values at the centers of the cells. CFD codes offer the user a choice of numerical schemes that express the convective terms and transient terms of the conservation equations as a function of the unknowns. They all incur numerical errors as they include approximations.

Thus, in this process, the computational domain is divided into small grid or mesh cells and the differential equations are applied to each cell. These differential equations are approximated to algebraic equations using various methods. The most common ones are the finite difference, finite volume and finite element methods. Most of the commercial CFD codes are based on the finite volume method, for example CFX4, CFX5, FLUENT etc. Hence a large system of algebraic equations at discrete locations in space and in time is obtained where the unknowns are the flow variables, such as velocity or temperature. The spatial locations are linked to the computational grid, either the grid cell centers or corners. The temporal locations correspond to the boundaries of the time steps which subdivide the continuous time into intervals. A solution algorithm is then used to solve this system of algebraic equations using an iterative procedure.

The most frequently encountered error due to the discretisation of the convective terms is known as 'numerical diffusion'. Its effect is to increase rates of diffusion of heat, mass and momentum, generally leading to an over-prediction of mixing. Numerical diffusion is most severe from use of discretisation schemes which are referred to as 'first-order accurate'. 'Higher-order accurate' schemes are likely to provide better results, but they tend to give more difficulty in obtaining a solution. First-order accurate schemes include those known as 'upwind', 'hybrid', 'exponential' and 'power law'. Higher-order accurate schemes include those known as 'QUICK', 'CCCT', 'TVD' and second order upwind. More details can be found in Patankar (1980) and Versteeg and Malalasekera (1995).

Time discretisation schemes are also of different orders of accuracy and are either explicit or implicit; depending respectively on whether the results at previous time steps are used in the solution of the current time step, or not.

2.4.6 Monitoring the iterative solution process

An approximate solution to the algebraic equations so obtained is solved by iterative algorithms (Patankar, 1980). These start by assuming an initial value to the flow solution. They gradually iterate to a final result which satisfies the imposed boundary conditions, while ensuring that mass, momentum and energy are conserved both locally for each grid cell and thus globally over the whole computational domain, resulting in a 'converged' solution. For unsteady problems this process is repeated at each of many time steps until the total time required has been covered.

2.4.7 Analysis of the solution obtained

The approximate solution obtained consists of the values of the variables of interest at all the grid cells and at the time steps chosen by the user. The 'post-processor' is used to analyze this large amount of data. A post-processor is a piece of software that can read a file containing the data from a CFD simulation and translate it into visualizations: for example, contours of temperature or smoke concentrations; three-dimensional iso-surfaces; plots of velocity vectors. The results predicted from CFD should be compared with experimental results or empirical correlations to ascertain the accuracy of results.

2.4.8 Remarks

Thus in a nutshell, the various sub models that can be helpful in fire safety engineering are as follows:

- Turbulence: A modified k-ε turbulence model which includes modifications for buoyancy effects in both k and ε equations should be used.
- Buoyancy: The Boussinesq approximation should not be used for modeling the effect of buoyancy.
- Boundary condition: Forced ventilation should be prescribed as a boundary condition.
- Heat transfer to walls: Heat loss to the walls can have important effect on smoke movement. A conservative approach is to assume no heat loss at walls, i.e. an adiabatic condition, but this approach will under-estimate smoke concentrations at low levels.
- Radiation: Radiation should be taken into account, at least by reducing the fire heat output to account for the radiative loss. In the far-field, more sophisticated radiation models should be used such as six flux model.
- Fire specification: A volumetric heat source or combustion model can be used when the fire is not influenced by proximate walls or ambient air flows and providing the fire source is well specified. Where the fire plume might be affected by proximate walls or local flow conditions which cannot be adequately determined in advance, a combustion model is preferred to a volumetric heat source model.

While modeling fires inside tunnels, one of the most challenging parts is to define traffic related fire source. Here the source of combustion is complicated since how the vehicle burns and also the calorific value is unknown.

One approach is to assess the range and combustibility of different traffic types in different fire scenarios and estimate a reasonable fire development and fire size. When incorporating this design fire scenario inside the CFD model the vehicle is represented as a blocked region.

A number of experiments have been conducted to assess the HRR of different vehicle fires and corresponding design fires and design fire scenarios have been defined. A short note on design fires and design fire scenarios for vehicles is given in the next section.

2.5 DESIGN FIRES AND DESIGN FIRE SCENARIOS

As described in last section, the fire source, which corresponds to a vehicle fire, can be specified in terms of the variation of heat release rate (HRR) of vehicle fire under different fire scenarios. Also, the peak HRR of fire gives an idea of the maximum heat load expected inside tunnel, based on which the ventilation system for smoke control is designed. Thus design fires and design fire scenarios are essential inputs to assess the fire safety design of new tunnels and evaluation of fire protection measures in existing tunnels.

ISO/TR 13387 which covers "Design fire Scenarios and Design Fires" (ISO, 1999) for buildings and which can equally be applied to tunnels defines 'design fire' as an idealization of a real fire that might occur. Design fires in tunnels are usually defined in terms of peak Heat Release Rate (HRR). It has been found from full-scale tests in the Runehamar tunnel (Lonnermark & Ingason, 2005; Ingason & Lonnermark, 2005; Lemaire, 2003) that the HRR can reach over 100 MW in less than ten minutes. This means that the fire growth rate is very important in determining whether the people caught in the fire can escape. Therefore, while defining design fire, the peak HRR is now combined with the fire growth rate.

Many large scale fire tests have been conducted, which showed that the peak HRR of vehicle fires inside tunnels varies between 1.5 MWs up to 202 MWs for road vehicles and from 7 MWs up to 43 MWs for rail vehicles. Ingason (2006) summarizes peak HRR of vehicles and coaches and time to achieve peak HRR from available literature and experiments. Broadly, the peak HRR and time taken to achieve peak HRR for different type of vehicles as suggested by Ingason (1995c) is given in Table 2.4.

Table 2.4 Peak Heat release rate for different type of vehicle fires

Type of Vehicle	**Peak HRR (MW)**	**Time to peak HRR (min)**
Single passenger car	1.5 - 9	10 -55
Bus	30	< 10
HGV trailor	13 -202	10 - 20
Rail coaches / subway car	7 - 43	5 - 80

ISO/TR 13387 also defines 'design fire scenario' as an interaction of the design fire with its environment, i.e. impact of the fire on the geometrical features of the tunnel, the ventilation and other fire safety systems in the tunnel, occupants etc. A design fire scenario would thus define the ignition source and process, the growth of fire on vehicle ignited within the tunnel, the spread of fire, the interaction of the fire with its enclosure and environment, and its eventual decay and extinction. The work package, WP2 of UPTUN project, a European 5th framework research project (Haack, 2007) which aims to provide a methodology for upgrading fire safety in existing tunnels, suggests following phases of design fire scenario (Figure 2.7):

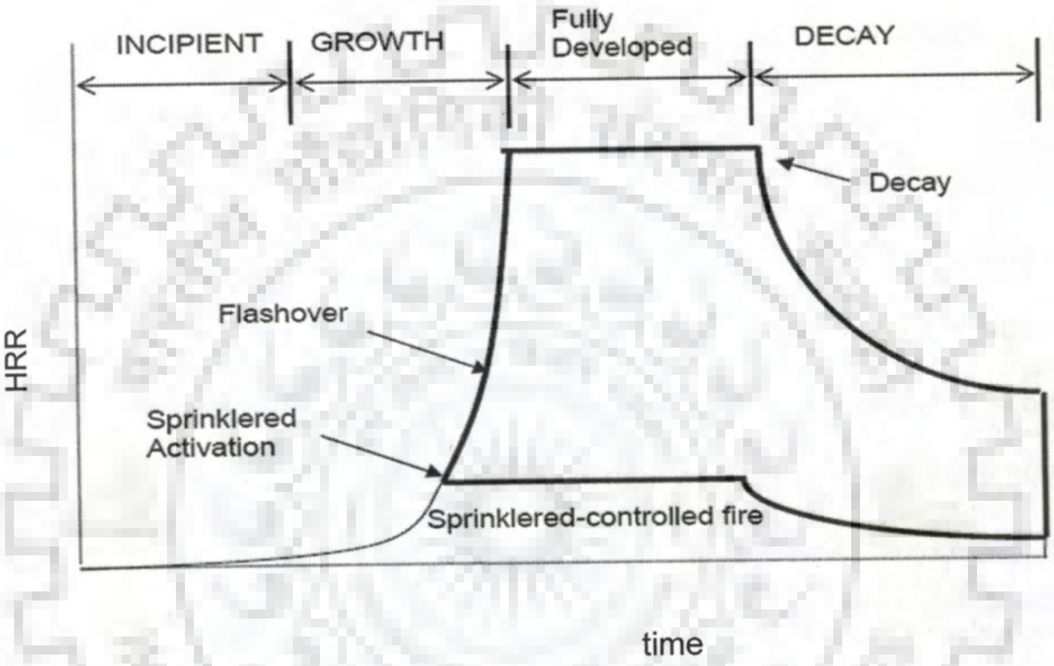


Figure 2.7 Schematic of a design fire scenario

a) incipient phase - characterized by a variety of fire sources, such as smoldering or flaming fire

b) growth phase - covering time of fire propagation up to flashover

c) fully developed phase - characterized by a substantially steady burning rate

d) decay phase - covering the period of declining fire severity

e) Extinction - when no more energy is being produced.

This work package (WP2) has also proposed design fire scenarios and associated design fire curves for road, rail and metro tunnels. The proposed design fire curves have all the phases of design fire scenario as mentioned above. It is desired to represent these design fire curves of vehicles in tunnels mathematically so that they can be used in fire models for fire specification. These curves can be defined in different ways depending on the fire growth rates such as linear growth ($\alpha\ t$), quadratic growth ($\alpha\ t^2$) or exponential growth ($\alpha\ (1- e^{-t})$) rate. These growth functions are combined with a peak HRR value and decay functions ($\alpha\ -t$ or $\alpha\ e^{-t}$). Few of various important mathematical methods to define fire curve for tunnels as summarized by Ingason (2006) is given below:

(i) Linear growth

This curve is based on The French tunnel recommendations (Lacroix, 1997) for fire ventilation. This assumes a time dependency of HRR with a linear growth from zero to time t_{max}, a constant maximum value till time t_D and finally a linear decrease from the maximum value to zero at time t_d.

$$Q = \alpha_{g,L}\, t \qquad 0 < t \le t_{max} \tag{2.26}$$

$$Q = \alpha_{g,L}\, t_{max} = Q_{max} \qquad t_{max} < t < t_D \tag{2.27}$$

$$Q = Q_{max} - \alpha_{D,L}(t - t_D) \qquad t_D < t < t_d \tag{2.28}$$

(ii) Quadratic growth

This curve (Ingason, 1995c) assumes a quadratic growth from zero to time t_{max}, a constant maximum value to the time t_D and finally an exponential decrease from the maximum value to zero at infinity.

$$Q = \alpha_{g,q} t^2 \qquad 0 < t \le t_{max} \tag{2.29}$$

$$Q = \alpha_{g,q}\, t_{max}^2 = Q_{max} \qquad t_{max} < t < t_D \tag{2.30}$$

$$Q = Q_{max}\, e^{-\alpha_{D,q}(t - t_D)} \qquad t_D < t \tag{2.31}$$

The design parameters, growth rate coefficient ($\alpha_{g,q}$) and decay rate coefficient ($\alpha_{D,q}$) proposed by Ingason (1995c, 2006) for different type of vehicles while using the quadratic method are given in Table 2.5.

Table 2.5 Proposed design parameters for creation of design fires for traffic tunnels (Ingason, 1995c)

Type of vehicle	**Q_{max}**	**$\alpha_{g,q}$ (MWmin^{-2})**	**$\alpha_{D,q}$ (min^{-1})**
Car	4	0.036	0.06
Bus	30	0.36	0.042
Truck	15 - 130	-	-
Train (Steel)	15	0.036	0.06
Subway car (Aluminium)	35	1.08	0.06

(iii) Exponential growth - fuel control

Ingason (2005, 2006) has proposed a method to estimate the HRR given as a single exponential function of time instead of three functions of time. It is based on original work by Numajiri and Furukawa (1998) and it is only thought for fuel controlled fires. The design parameters are the peak HRR (Q_{max}) and the parameter n, which is arbitrary chosen parameter with no physical meaning. Other parameters are r and k, which are calculated based on the information given. For details refer Ingason (2006).

$$Q = Q_{max}\, n r (1 - e^{-kt}) \qquad t \geq 0 \qquad (2.32)$$

2.6 MOTIVATION FOR THE STUDY

While serving with Central Building Research Institute, and engaged in several national programs related to fire safety in building enclosures and infrastructure, that too in a laboratory unique in our country; I got an opportunity of getting introduced to some scientific challenges in designing / evaluating ventilation systems of very large aspect ratio enclosures. The Delhi Metro, which is by and large the most important infrastructure in India that started during this period and lack of Indian Standards, fascinated me to study fire safety inside these tunnels. It prompted me to evaluate ventilation strategies of the tunnel configurations similar to Metro corridor using modeling techniques.

The first phase of the project under execution comprises of 62.5 km of route length of which about 13 km is underground and is called the Metro corridor. On 24th December 2002, the first line of Delhi Metro was opened and the Phase I of the project was completed in December 2005. At present Phase II is in progress. It is proposed that the four phases planned till date shall be completed by 2020. After completion of these four phases, the Delhi Metro will be larger than the London Metro.

The study of ventilation system of Delhi Metro showed that the ventilation configuration used is jet injection type. The tunnel ventilation system consists of ventilation shafts located at each end of the station. Each ventilation shaft has a fan room in which there are fully reversible ventilation fans. The physical shape of ventilation shaft plenum where it connects from the fan room to the tunnel is such that it resembles and performs like an impulse nozzle. These nozzles are inclined at an angle to the plane of ceiling. In addition to this jet fans are located in the track ways at certain places to assist in controlling air movement within the track ways. These are either wall mounted or ceiling mounted.

This inspiration was articulated after proper literature review and identification of gaps in the form of the objectives of the thesis as given in chapter 1.

2.7 CONCLUDING REMARKS

In this chapter, the literature related to various aspects of movement of smoke and toxic gases resulting from fires in tunnels and subsequent use of ventilation for smoke control has been reviewed. Besides discussing the basics of the smoke control using ventilation system, important experimental studies have been reviewed. Thereafter, the role of different mathematical models (empirical formulas, network models, zone models and CFD models) for predicting thermal environment inside tunnels and in the design and analysis of ventilation systems are analyzed with respect to their scope and limitations. Special emphasis is given to field models (CFD) as these are interest to us in the research work. The appropriate physical models and boundary conditions required to construct a CFD model for fire modeling is then discussed. In the end, design fire and design fire scenarios used for representing fire source of vehicles are discussed. Finally motivation for the study is given.

CHAPTER – III

COMPARATIVE STUDY OF FIRE SAFETY MEASURES IN LONG DISTANCE ROAD TRANSPORT TUNNELS

3.0 INTRODUCTION

This chapter studies the important safety features of long distance road tunnels. It is observed during study of various tunnels that all the tunnels have different fire safety features depending upon their requirements. It is also observed that most of the countries either do not have or have minimal guidelines for fire safety inside transport tunnels. Till late 80's there had been no formal guidelines for the safety features to be installed in the tunnels. NFPA – 502, 'Standard for Road Tunnels, Bridges, and Other Limited Access Highways' was only a recommended practice. It evolved into a code in 1998. In December 2002, The European Commission proposed a new directive on 'Safety in European Road Tunnels'. There exist differences in some provisions as given in NFPA 502 and proposed directive of European Commission.

In view of above, it is desired to formulate general fire safety guidelines which should be complied in future tunnels for improved safety. For this, few of world's longest and important road tunnels are studied with respect to fire protection measures installed in them. The tunnels included Autostrada tunnels, Italy; St Gotthard Tunnel, Switzerland; Seelisberg Road Tunnel, Switzerland; Frejus Road Tunnel, France & Italy; The Mont Blanc Tunnel, France & Italy. These tunnels are so chosen that they are representative of almost all of tunnels built. For example, Autostrada tunnels have varied length from 100 m to 3 km, St. Gotthard tunnel is the longest and the busiest tunnel having a parallel safety tunnel, The Seelisberg Tunnel is the longest twin bore tunnel in the world, The Frejus Tunnel has a railroad tunnel at the same location and is managed by two different countries. The two tunnels St Gotthard and The Mont blanc Tunnel were renovated with improved safety features after fire incidents in them. The fire protection measures in these tunnels have been studied under following parameters - Traffic signal system, Lightening system, Ventilation system, Fire Detection system, Fire Protection system, Control Rooms and Escape & Refuge. These are then compared with NFPA 502 and proposed directive of European Commission, based on which an attempt is made to formulate general safety guidelines for road tunnels.

A brief description of these tunnels is as follows:

a) Autostrada Tunnels near Genoa, Italy

These tunnels (Egilsrud, 1984) are on the main autostrada (or super highway) which runs along the Mediterranean coast passing through LaSpezia, Rapollo, Nervi, Genoa, Savona, Albenga, Imperia, San Remo and Ventimiglia to the French border. There are similar tunnels on three autostrades leaving Genoa and Savona for Turino and Milano. The autostrades have over 500 km of roadway in tunnels along these routes. There are approximately 90 tunnels, from less than 100 m to as long as 3000 m. Most of the tunnels are between 100 and 600 m.

b) St. Gotthard Tunnel

It is the longest road tunnel (Egilsrud, 1984; www.gruxa.ch/ gotthard_road.htm) in the world-16.918 km, portal to portal. The tunnel lies completely within Switzerland and passes between the Swiss Cantons of

Ticino on the south and Uri on the north. Tunnel construction started in 1969 and opened to traffic in 1980. The connecting roads north and south of the tunnel each have four lanes of traffic; the tunnel has only two lanes. The tunnel with an average of 19000 vehicles a day is one of Europe's busiest north – south routes. This tunnel has a parallel safety tunnel. On October 24, 2001, a serious accident inside the tunnel claimed 11 lives. Temperatures reached up to 1,800 degrees Fahrenheit causing damage to the tunnel structure resulting in closure of tunnel for months.

c) Seelisberg Tunnel

The Seelisberg Tunnel (Egilsrud, 1984) is a twin bore, 9.25 km long tunnel on the Swiss National N-2 motorway connecting Basel and Chiasso. It is the longest twin – bore tunnel in the world. The St. Gotthard, Mont Blanc, and Frejus tunnels are longer, but they have only a single bore with opposing traffic. The Seelisberg Tunnel is 35 km from the St. Gotthard tunnel on the same route. The general installations of fire safety and traffic controls in this tunnel are similar to St. Gotthard Tunnel, except that there is no safety tunnel.

d) Frejus Tunnel

The Frejus Tunnel (Egilsrud, 1984; www.tunneldufrejus.com/english/presentation.htm) lies between Lyon, France, and Turino, Italy and is the shortest route between Brest, Paris, and Lyon in France and Turino, Milano, and Rome in Italy. There is a railroad tunnel at the same location through which the fastest trains between Paris and Rome passes. The tunnel is about 13.5 km, making it one of the longest road tunnels in the world.

e) Mont Blanc Tunnel

The Mont Blanc Tunnel (Egilsrud, 1984; http://news.bbc.co.uk/2/hi/europe/1858436.stm; Bettelini et al., 2001; www.mrtunnel.com/page3.htm) was built jointly by France and Italy and opened in 1965. It is 11.6 kilometers long and passes under the Alps near the highest mountain in Europe, Mt. Blanc. Because of its location, it has become a very heavily traveled route between France and Italy. The longest part of the tunnel is on French territory: 7,640 m, with 3,960 m in Italy. Two operating Entities were created, the STMB (Societe du Tunnel du Mont Blanc) in France, which became ATMB (Autoroute et Tunnel du Mont Blanc), and SITMB (Societa Italiana del Traforo di Monte Bianco) in Italy, each operating half of the tunnel.

Modifications were done in this tunnel in 1979, 1980, 1981-82. However a devastating fire in 1999 led to a radical reassessment of safety needs in the Mont Blanc Tunnel. The tunnel was opened in 2002 after renovation.

A comparative study of these tunnels with respect to length, purpose, location, type of construction, lightening system, traffic signal system, ventilation systems, fire detection & suppression systems etc. is presented in Table 3.1.

Table - 3.1 Comparative Study of Long Distance Road Tunnels

S.N.	Parameters	Auto Strada Tunnels	St. Gotthard Tunnel (SGT)	Seelisberg Road Tunnel	Frejus Road Tunnel	The Mont Blanc Tunnel	NFPA 502 (NFPA, 2001)	European Commission proposed directive [7,8]
1	Year of Opening		1980	1980	1980	1965, renovated in 2002		
2	Location	Near Genoa, Italy	Switzerland, Between Ticino and Uri	Swiss National N-2 motorway	Between Lyon, France & Rome, Italy	Between France & Italy, Passes under Alps		
3	Modes of Transport	Road	Road	Road		Road		
4	Tunnel Length	100- 3000m > 90 tunnels	16.918 km	9.25 km	13.5 km	11.6 km		
5	Tunnel Cross - section	Circular arch design [1]	Arched ceiling 4.5m high [1]	At portals, horse shoe cross section , in middle- circular cross section [1]	Horse shoe cross section. Max height = 4.30 m [3]	Horseshoe cross section -- 46 m^2 [6]		
6	Inclination of Tunnel / Road way gradient	Nil	Very gentle, approx 1.4% [2]	0.45 % in north, 0.6% in south [1]	Gentle 0.54 % [1]	Maximum 2.4% [5,6]		Tunnel gradient above 5% shall not be allowed
7	Type of Bore	Twin – both for traffic [1]	Twin one for traffic and other for safety tunnel [1]	Twin [1]	Single [1]	Single [1]		Single tube tunnels should only be built if long term forecasts shows moderate traffic volume
8	Tunnel Lining	Reinforced concrete/sto	Reinforced concrete [1]		Reinforced concrete. On	Reinforced concrete [1]		

For [1,2,6,7,8], …., pl. see end of table

		ne/concrete blocks [1]			French side wall panels are fire retardant, On Italian side they are concrete [1]			
9	No of Traffic Lanes	Four/six [1]	Two [1,2]	Two [1]	Two [1,3]	Two [1]		
10	Width of Roadway	12 ft / 3.6m [1]	7.8 m [1]	7.5 m [1]	9 m [3]	7 m [5]		
11	Width of Sidewalk on each side of road	2 ft ~ 0.6 [1]m	0.7 m [1]	0.8 m [1]	0.9 m[1]	0.8 m [5]	On each side of cross passageways a walkway with a minimum width of 1 m shall be given	
12	Tunnel drainage –(Gutter width between tunnel wall & sidewalk)	8-10 inches [1]	Traffic bore side walls have a void space for handling seepage[1]	The walls have a curved panel mounted free of structural lining to provide space for seepage [1]	Tunnel wall panels are free of wall to allow seepage [1]	------	A drainage shall be provided. The design shall be such that spills of hydrocarbons cannot propagate along length of tunnel	

For [3,5], …., pl. see end of table

13	Traffic Signal System	No [1]	Placed every 250m [1]	Yes [1]	Every 600 m [1,3]	Every 1200 m [4]		
14	Lightening Systems	Mostly Fluorescent & low pressure Sodium fixtures In some instances high press. Sodium lights also used [1]	Continuous single lamp strip of 40 w fluorescent lamps is installed in eastern lateral corner of tunnel section. The output of lamps can be regulated. Every 10th lamp is fed permanently through a static inverter [1]	Continuous fluorescent strip lighting to adjust for Day/Night traffic installed in one upper corner of each traffic bore. It is augmented by high press sodium fixtures [1]	Fluorescent and Low pressure sodium lighting fixtures on both sides of road way in corner between wall & ceiling. Fixtures are not continuous. The design illumination is 50 lux though it can be varied [1]	Fluorescent &low pressure sodium fixtures of 40 W each placed alternately 4 m apart. No of low pressure sodium lights are more in transition zone near each portal [1]	The illumination levels of tunnel roadways, walkways, and walking surfaces shall be more than 3 lux at the walking surface	Safe feeding of high voltage and low voltage cables need to be provided for
15	(i)Emergency Lighting	Not provided [1]	Provided on tunnel east wall about 50cm above side walk at 50m intervals [1]	Provided as in St. Gotthard Tunnel [1]	----------	--------	It shall be installed in accordance with relevant codes of NFPA. There shall be no interruption of light for more than 0.5 second	

For [4], ..., pl. see end of table

16	(i)Ventilation system	<1500 m No Ventilation System, Vent. by piston effect >1500 m Ventilation controlled by CO analyzers & opacimeters [1]	Fully transverse, separate fresh air & exhaust ducts placed in arch above ceiling. Fresh air ducts larger than exhaust ducts. Safety tunnel equipped with separate ventilation system where air is supplied at a pressure higher than traffic tunnel Small ducts behind side walls serve outlets above sidewalk on one side of tunnel [1]	Fully transverse Separate fresh /exhaust air ducts. Two normal cross-section used at portals horse shoe section , in middle circular section. The ducts are above ceiling in horse shoe cross section . In circular section exhaust duct above ceiling and fresh air duct below roadway[1]	Fully transverse Separate fresh air/ exhaust air duct on Italian side. On French side a single shaft is portioned into two [1]	Semi transverse Because of high mountain range above the tunnel all ventilation done through portals. The ducts were made under the roadway. Total length divided into four ventilation section [1]	For tunnels > 240 m ventilation is required	Single tunnels with bi directional traffic shall have transverse and/or semi transverse ventilation with exhaust possibilities. Longitudinal ventilation shall be used in these tunnels only when traffic conditions allow uncongested vehicles to drive out of tunnel
	(ii) Fan Location	Vane axial, installed in ceiling arch, exact position not specified [1]	Six fan rooms one at each portal and one under each of vertical	Two fan room at portal and two inside	On each side of gallery there are two exhaust fans & two supply fans [1]	Each duct has two centrifugal fans.		

			shafts. They are placed above roadway ceiling [1]					
	(iii) Maintenance of air quality & Fan speed	Fan speed Low if co conc.>100ppm or opacity >15%, High if co conc.>150ppm or opacity > 30%, Alarm if co conc.>200ppm [1]	Air quality maintained through CO analyzer and visibility level detectors. Exhaust ports at every 25m [1]	Air quality maintained through CO analyzer and visibility level detectors. Exhaust ports are installed in ceiling at 16 m centers. Fresh air flues blowing air into roadways are installed every 8m [1]	Air quality maintained through CO analyzer and visibility level detectors. Maintains 100 ppm of CO with max limit of 150 ppm, Atmospheric clarity maintained between 15 – 30 %. Exhaust ports are installed in ceiling. Fresh air is introduced just above side walk [1,3]	Air quality maintained through CO analyzer and opacity sensing systems. A total of 10 sensors are placed. Exhaust through portals, fresh air through ducts Fresh air is introduced just above side walk [1,4]	Air quality : NFPA states that after first 15 min of exposure, average CO conc. Shall be les than 50 ppm	
	(iv) capacity of fans	----	2150 m^3 [1]	3540 m^3 [1]	1580 m^{3}of fresh air & 1300 m^{3}of exhaust air [1,3]	900 m^{3}of fresh air & 300 m^{3}of exhaust air [5,6]		
	(v) vehicle load(used for design)	----	3600 cars per hour [1]	3600 cars per hour [1]	3600 cars per hour [1]	600 vehicles per hour which has exceeded many times [1]		
17	Fire Detection	Not in tunnels					At least 2 detection	

	systems	<1500m length [1]					systems of which one would be manual	
	(i) Detectors (heat, fire, smoke)	No detectors installed [1]	Two stage sensitivity detectors installed at every 25 m intervals [1]	Fire detectors with single level sensitivity every 30m. In addition it has CO & visibility detectors [1]	No	Italian side – detection by pressure variations in 70-80 m long tubes at ceiling, On French side temperature sensors are placed every 8 m.116 smoke detectors placed every 100m Heat sensors placed at both ends of portals [1,4]	(i) Manual Double action fire alarm should be installed at < 90m and at all cross passages and means of egress from tunnel Spot detectors shall have a light that remains on until device is set	
	(ii) SOS cabinets provided (emergency system) consisting of phone, two dry type fire extinguishers, warning	Only call boxes at hose valves every 100m, push button to request mechanical assistance and a	SOS box every 125 m on west side and 250 m on east side. It contains two dry type fire extinguisher and a	Same as in St. Gotthard Tunnel	The SOS boxes are provided on both sides. The SOS box contains a fire extinguisher, push button for fire or emergency and a telephone [1]	The SOS boxes are placed inside air conditioned glass enclosure every 600 m. It contains a dry powder type fire extinguisher push button and a emergency	(ii) Automatic fire detection system shall be capable of identifying location of fire within	

	lamps etc)	ambulance [1]	intercom. Lifting telephone receiver or removing extinguisher prompts traffic lights to interrupt traffic flow and activated TV system [1]			telephone. At every 100m [5] a fire pull box and 2 fire extinguishers are placed. Total of 72 telephones installed [1,4]	15 m	
	(iii) Television cameras	Nil [1]	83 cameras installed at every 250 m [1]	Every 30 m [1]	Total 83 TV cameras [3]	120 TV cameras are spaced at every 300m interval [4]	(iii) 24 hr supervision is necessary	Video monitoring system to be installed in tunnels longer than 1000 m
	(iv) Traffic control loops to track traffic flow	Nil [1]	Traffic loops are embedded in road way and tied to computer [1]	Same as in St. Gotthard Tunnel	Nil	Nil		
18	Fire Protection Systems	Tunnels <1500m have no fire protection measures	Emergency fire station at each portal	Same as in St. Gotthard Tunnel				
	(i) fire hydrants	Fire mains with hose valves every 100m[1]	Provided at regular intervals	Provided at regular intervals	Provided on one side only	Provided at every 150 m fire niches provide water supply to fire fighters [5]	Hose pipe connection spacing < 85 m, stand pipe required	Water supply to be available at intervals of at-least 150 m

							flow rate 1920 l/min	
	(ii)fire extinguishers	Nil	Inside SOS boxes	Inside SOS boxes	Inside SOS boxes	Inside SOS boxes	Portable fire extinguisher at less than 90 m interval	Fire extinguishers at less than 150 m interval
	(iii) fire engines	With local fire dept., not exclusive [1]	At fire stations, exclusive, have two way radio communication system [1]	Yes , exclusive [1]	Yes , exclusive [1]	Foam equipped jeeps, not exclusive [1]	For long tunnels should be placed at tunnel portals	
19	Control room	yes	Sophisticated control room at each portal [1]	Control room at each portal [1]	Sophisticated Control room at each portal [1]	Control room at each portal and one at center [4]		
20	Transportation of hazardous goods allowed	Yes [1]	No [1]		No, All authorized hazardous cargo to have double escort [3]	No, allowed in limited quantities[1]	Operating agency shall adopt rules and regulations	
21	Escape, Refuge	Nil	Have pressurized refuges [1]		Have pressurized refuges [1]	Have pressurized refuges [1,4]	For single bore tunnel emergency exit every 300 m	Emergency exit at intervals of less than 500 m
	(i) Safety tunnels		Yes, 7.5 m^2 cross section [1]	No	No	No		
	(ii) cross		yes	yes	No	No	For double	Cross

	passages						bore, cross passages at less than 200 m	passages at maximum intervals of 500 m
	(iii) Room for refuge		Sheltered pressurized room at every 250 m between main &safety tunnel [1]	In cross passage between two tunnels	Opposite emergency car park every 2.1 km [1,3]	Every 600 m, 2 AC glass enclosures for refuge [4]		
	(iv) Emergency car parking		Every 750 m [1]		The tunnel is enlarged every 2.1 km	The tunnel is enlarged every 600 m [5]		Distance between lay bys shall not exceed 1000 m

References

[1] (Egilsrud, 1984)
[2] www.gruxa.ch/gotthard_road.htm
[3] www.tunneldufrejus.com/english/presentation.htm
[4] http://news.bbc.co.uk/2/hi/europe/1858436.stm
[5] www.mrtunnel.com/page3.htm
[6] (Bettelini et al., 2001).
[7] http://europa.eu.int/comm/transport/road/roadsafety/roadinfra/tunnels/index _en.htm.
[8] http://europa.eu.int/scadplus/leg/en/lvb/l24146.htm.

From the table following observations are made:

3.1 OBSERVATIONS

(a) These are some of longest road tunnels in Europe, their length varying from 9 – 17 km except Autostrada Tunnels which are 0.1–3 km long. The tunnels are chosen such that one is twin bore, both for traffic, one has safety tunnel, two have single bore.

(b) **Tunnel Gradient:**

The roadway gradient is very gentle, less than 5%.

(c) **Safety Tunnel**

Only St. Gotthard Tunnel has safety tunnel. Seelisberg Tunnel has two bores, both for traffic.

(d) **Emergency Walkway**

All the tunnels have emergency walkway. The walk way width varies from 0.6 m to 0.9 m.

(e) **Traffic Signal System**

There is no traffic signal system in Autostrada tunnels. St. Gotthard tunnel which is 16.9 km has Traffic Signal System every 250 m while Frejus road tunnel which is 13.5 km have Traffic Signal System every 600 m. Mount Blanc Tunnel have Traffic Signal System every 1200 m.

(f) **Lighting System**

(i) All tunnels have adequate lighting provisions. Mostly fluorescent and low pressure sodium fixtures are used. In Mt. Blanc Tunnel the bulbs are placed every 4 m. The illumination can be varied. In Seelisberg tunnel every 10th lamp is fed permanently through a static inverter.

(ii) Emergency Lighting: Autostrada tunnels do not have provisions of emergency lighting system because of short length of tunnels. In St. Gotthard and Seelisberg tunnel emergency lighting is placed every 50 m.

(g) Ventilation System

(i) All tunnels except Autostrada tunnels which are less than 1500 m have ventilation systems incorporated in them.

(ii) Air quality: is maintained through carbon mono oxide analyzers and visibility detectors or opacity sensing systems. The concentration of CO is maintained at 100 ppm with a maximum limit of 150 ppm and atmospheric clarity is maintained between 15-30% in Autostrada and Frejus tunnels.

(iii) In cases where transverse ventilation systems are incorporated i.e Seelisberg Tunnel, St. Gotthard Tunnel, Frejus Tunnel, exhaust ports are placed in the ceilng. In St Gotthard Tunnel they are placed every 25 m while in Seelisberg tunnel they are placed every 16 m.

(iv) Fresh air ducts are placed just above side-walk. In Seelisberg tunnel they are placed at every 8 m.

(h) Fire Detection System

(i) Detectors: St. Gotthard Tunnel uses two stage sensitivity detectors. In such detectors the first stage is more sensitive than the second. Upon sensing a fire the following occurs. The supervising personnel in the control room are alerted by optical and acoustical signals; the position of detected fire is shown on a special panel display in the control room, the TV system is switched on; the tunnel illumination increases to a maximum level, the emergency lighting is turned on and ventilation system is switched over to fire program when less sensitive second stage of a fire detector is activated, the fire department is summoned.

Mont Blanc Tunnel uses a combination of detectors, temperature sensors, smoke detectors, heat detectors etc.

(ii) SOS boxes / cabinets have been used in all the tunnels. These allow a person calling the control room to hear and speak despite noise in the tunnel. These cabinets also have two dry powder fire extinguishers. If a fire extinguisher is removed from the cabinet, an alarm sounds in the control room & tunnel systems respond as they do when first stage of a fire detector is triggered. If a control room decides that help is needed a message appears on the screen in the box indicating in several languages that help is on the way.

In Mt. Blanc Tunnel they are placed in air conditioned glass enclosures every 600 m where as in all other tunnels they are placed at every 100 - 250 m. However, in Mt. Blanc Tunnel a fire pull box and 2 fire extinguishers are placed every 100 m.

(iii) Television cameras: All tunnels except Autostrada tunnels have television cameras installed in them.

(iv) Traffic loops: St. Gotthard & Seelisberg tunnels have traffic loops. These are embedded in the roadway and tied into the computer, which keeps track of traffic flow. The system is sensitive enough to detect a single vehicle stopping, an alarm is sounded and the monitor automatically shows the view from nearest TV camera. Traffic signals turn red or flash yellow behind the vehicle involved on both roadways to warn motorist that there are problem up a head.

(i) Fire Protection System

(i) Fire hydrants: All tunnels are provided with fire mains with hose valves at regular intervals.

(ii) Fire extinguishers: There are no Fire Extinguishers in Autostrada Tunnels. All other tunnels have fire extinguishers placed inside SOS boxes. Only Mt. Blanc Tunnel have fire extinguishers along the side of roads every 100 m apart from SOS cabinets.

(j) Escape and Refuge

(i) Pressurized refuges: Only Mont Blanc Tunnel, Gotthard Tunnel and Frejus Tunnel have pressurized refuges. All tunnels except Autostrada tunnels have refuges. This can be attributed to short length of Autostrada Tunnels.

(ii) Safety Tunnel & cross passages: Only St Gotthard Tunnel has a safety tunnel used exclusively for rescue operations. Seelisberg Tunnel is a twin bore tunnel where both tubes are used for traffic. Hence in times of emergency the other tube can be used as a safety tunnel. Both these tunnels are connected by cross passages every 250 m. All other tunnels do not have cross passages or emergency exits.

(iii) Emergency car parking: St Gotthard tunnel have emergency car parking every 750 m while Mt. Blanc Tunnel have emergency car parking every 600 m. The emergency car parking of Frejus Tunnel at every 2.1 km seems to be a very large distance between two emergency car parkings.

(k) Fire safety arrangements in Mt. Blanc Tunnel till 1999, when tunnel was closed for renovation after a major fire were not as sophisticated and extensive as those of Frejus, St Gotthard and Seelisberg Tunnels. Several fire detection systems have been incorporated after that. The glass enclosed safety rooms, even though they have a ventilation connection from the supply air duct, appear to offer poor refuge for personnel because of glass walls and their size: too small for the number of people that might need them in traffic.

(l) The Swiss have incorporated unique traffic control, safety, fire and other emergency features in St Gotthard tunnel. They have spared little to make their tunnels as safe as possible.

These tunnels have also been compared with NFPA 502 (2001) guidelines and proposed directive of European Commission on 'Safety in European Road Tunnels',2002(http://europa.eu.int/scadplus/leg/en/lvb/l24146.htm; http://europa.eu.int/comm/transport/road/roadsafety/roadinfra/tunnels/index_en.htm.) in Table 3.1 itself. A critical assessment is presented below:

3.2 COMPARISON with NFPA 502 & PROPOSED EUROPEAN COMMISSION DIRECTIVE

(a) Emergency Walkways

NFPA does not mention the minimum width of walkway. However it mentions that on each side of the cross passageways an emergency egress walkway with a minimum clear width of 1 m (3.6 ft) shall be provided. In the tunnels mentioned above the walkway width used is between 0.6-0.9 m. Hence walkway width inside main tunnel should also be taken as 1 m.

(b) Traffic Signal System

NFPA states that tunnels longer than 240 m (800 ft) shall have traffic control system. However, NFPA does not indicate the position of the traffic signal system. In the tunnels studied, traffic signal systems are place at intervals of 250-1200 m. The interval of 1200 m seems to be large. Hence, traffic signal system should be placed at about 600 m distance as in Frejus Road tunnel.

(c) Ventilation System

(i) All tunnels greater than 240 m shall have ventilation systems. Hence in some Autostrada tunnels ventilation system should be incorporated.

(ii) NFPA states that after first 15 minutes of the exposure, average CO conc. shall be 50 ppm or less. In the tunnels mentioned above the average CO concentration is maintained at 100 ppm. Since CO is very toxic hence average CO conc. should be strictly maintained at less than 50 ppm.

(iii) The position of exhaust and fresh air supply ducts is not mentioned in the NFPA code and European Commission directive. In tunnels, exhaust ports are placed in the ceiling at every 16 m. Fresh air ducts are placed just above side walk at every 8 m. Since the size and number and subsequently location of exhaust and fresh air ducts would depend on ventilation requirements of the tunnel which would further depend on the traffic volume hence these would be different for different tunnels. The tunnels studied are having high traffic density hence for such type of tunnels these may be used as a norm.

(d) Fire Detection Systems

(i) NFPA and European Commission directive does not specify the type and location of detectors to be installed. NFPA only states that there shall be at-least two systems to detect a fire of which one should be a manual system. The manual double alarm fire boxes shall be installed at intervals of not less than 90 m and at all cross passages and means of egress. In the tunnels these are placed at intervals ranging between 100 – 600 m. In order to comply with the provisions of the code, fire alarm button or emergency telephone should be placed at intervals of not more than 90 m.

(ii) NFPA and European Commission directive does not mention anything about SOS cabinets. Since it is equivalent to a manual fire detection system and fire extinguisher placed inside it is also to be kept at intervals not exceeding 90 m as per NFPA and at intervals of 150 m as per European Commission directive., hence these should be placed at intervals of 90 m if fire alarm buttons and fire extinguishers are not kept at intervals of less than 90 m.

(iii) Nothing is mentioned about traffic loops in the NFPA and European Commission directive.

(e) Fire Protection Systems

(i) Fire hydrants: A Hose connection spacing shall not exceed 85 m as per NFPA. European Commission directive proposes this to be 150 m. Therefore, for better safety these shall be installed at intervals not exceeding 85 m. They should also be so spaced that no location on road way is more than 45 m. This is as per NFPA code.

(ii) Fire extinguishers: NFPA states they shall be placed in intervals of not more than 90 m while European Commission directive recommendations are of 150 m. The tunnels have fire extinguishers placed inside SOS cabinets whose distance varies from 100–600 m. Hence Fire extinguishers should be placed at intervals of less than 90 m.

(f) Escape and Refuge

(i) Emergency exits (for single bore tunnels): As per NFPA these shall be provided throughout the tunnel and spaced so that the travel distance to an emergency exit shall not be greater than 300 m (1000 ft). However European Commission directive proposes this to be 500 m interval. These should thus be provided at intervals of less than 300 m.

(ii) Cross passageways (for twin bore tunnel): NFPA states that these shall not be farther than 200 m (656 ft) apart. European Commission directive states that Pedestrian cross connections shall link the tubes at maximum intervals of 500 m. Every third cross connections shall allow the passage of emergency service vehicles. Thus as a stringent measure pedestrian cross-connections shall be made at distances less than 200m while every third cross connection shall allow the passage of emergency vehicles.

(iii) Lay bys: Nothing is mentioned of lay bys or emergency car parking in NFPA. As per European Commission directive, distance between lay-bys shall not exceed 1000 meters. In existing tunnels they are placed at intervals as long as 2.1 km and as short as 600 m. Hence European Commission directive should be followed. However for better traffic management, tunnels should have a enlarged section at as less distances as possible.

(g) European Commission directive does not mention anything about width of walkway, air quality standards, and details of fire detection systems.

3.3 CONCLUDING REMARKS

The above assessment shows that the designs, provisions etc. of the existing tunnels should not form a basis for design of new tunnels. There are differences in some provisions as given in NFPA 502 and proposed directive of European Commission. In order to provide better safety features inside tunnels, the most stringent measures are proposed. In view of the provisions of existing tunnels, recommendation of NFPA code and proposed directive of European Commission, following safety guidelines are proposed:

1) The roadway gradient should be very gentle, less than 5%.
2) Single tube tunnels should only be built if long term forecasts show that traffic volume within the tunnel will remain moderate.
3) The walkway width inside main tunnel should be 1 m.
4) The traffic signal system should be placed at about 600 m interval.
5) There shall be provision for emergency lighting as per with NFPA 70 (NFPA, 2002a), National Electrical code; NFPA 110 (NFPA, 2002b), Standard for Emergency and Standby Power; and NFPA 70B (NFPA, 2002c), Recommended Practice for Electrical Equipment Maintenance. There should be no interruption of lighting levels for greater than 0.5 second.
6) Air quality: The concentration of CO should be maintained at 50 ppm and atmospheric clarity should be maintained between 15-30 %.
7) In cases where transverse ventilation systems are incorporated and traffic density is high exhaust ports should be placed in the ceiling at every 16 m. Fresh air ducts should be placed just above side walk at every 8 m.
8) There shall be both manual and automatic detection system installed to detect a fire. The manual double alarm fire boxes, (push button alarm, emergency telephone, SOS cabinets) should be installed at intervals of not less than 90 m and at all cross passages and means of egress.
9) For automatic detection of fire, a combination of detectors should be used. In addition, it is better to use two stage sensitivity detectors. Temperature

sensors should be placed at every 8 m, smoke detectors should be placed at every 100m. Fire detectors should be placed at every 30 m intervals.

10) Spot detectors shall have a light that remains on until device is set.
11) Video monitoring system should be installed in tunnels longer than 1000 m. Their location and number should be such that supervision of entire tunnel is possible.
12) Traffic loops should be embedded in the roadway and tied to computer.
13) Fire hydrants should be placed at intervals of 85 m.
14) Fire extinguishers: needs to be systematically installed in the tunnels at intervals of at least 90 m and at entrances, cross passages and means of egress.
15) Emergency exits should be provided throughout the tunnel and spaced at intervals of less than 300 meters.
16) Distance between lay-bys should not exceed 1000 meters.
17) Escape routes: should be indicated by lighting at least every 100 m and by signs every 25 m.
18) In twin bore tunnels cross passageways should be used for escape and rescue.
19) Pedestrian cross passageways should link the tube at intervals of less than 200 m (656 ft) apart.
20) Every third cross connection should also be designed for passage of emergency service vehicles.
21) Openings in cross passageways should be protected with self-closing fire door assemblies having a minimum of a 1-hour rating and shall be installed in accordance with NFPA 80 (NFPA, 1999), Standard for Fire Doors and

Fire Windows.

22) Cross passages, safety tunnel and refuge rooms should be pressurized to prevent propagation of smoke or gases.

23) The walking surfaces of the emergency exits cross passageways and walkways should be slip resistant.

24) Specific signs should be used to designate escape routes and safety facilities in tunnels.

It is suggested that in future tunnels these observations / recommendation should be complied. The higher safety standards will ensure less fatalities and less damage to tunnel structure in case of fire. The emphasis should be to accept safety features/expenditures as an intrinsic part of transport investment and not as an afterthought to address any undesirable externality. This will also bring more confidence among tunnel users.

CHAPTER - IV

NUMERICAL SIMULATION OF FIRE IN A TUNNEL: COMPARATIVE STUDY OF CFAST AND CFX PREDICTIONS

4.0 INTRODUCTION

This chapter presents a comparative study of two different modeling techniques – zone models and CFD model for simulating a fire inside a naturally ventilated tunnel. The objective is to ascertain the scope and utility of simpler and time efficient zone models in predicting fire environment inside tunnels. The zone model used is CFAST "Consolidation model of fire growth And Smoke Transport" (Jones & Forney, 1990) while field model used is General purpose CFD code "CFX" (AEA Technology, 2002). The vertical variation of temperature at different distances and at different times is used to compare the results.

Since tunnels are long, therefore a stable smoke layer might not be formed. However, when used carefully, a zone model may be used for estimating the fire environment in a tunnel by dividing it into several smaller segments. This concept is used to simulate a tunnel fire using zone model. Therefore, multi-cell approach is an extension of zone models to analysis of long tunnels. At the same time it is an approximation to more complex CFD approach or field models.

The following section describes briefly the fire models used, i.e. CFAST and CFX after which there is a section highlighting the numerical simulations, main results and its analysis.

4.1 FIRE MODELS

4.1.1 CFAST

The model "Consolidation model of fire growth And Smoke Transport" (CFAST) developed at NIST laboratory is a multi-room zone model, with the capability to model multiple fires and targets. This model can be used to calculate the evolving distribution of smoke and fire gases and the temperature throughout a building during a fire. CFAST is the result of a merger of ideas that came out of the FAST (Jones, 1985) and the CCFM.VENTS (Forney & Cooper, 1990) development projects.

In CFAST, each room is assumed to be divided into two layers; an upper hot layer and the lower cool layer. Since these layers represent the upper and lower parts of the room, conditions within a room can only vary from floor to ceiling and not horizontally. The set of equations that predict state variables (pressure, temperature, etc) are solved based on the enthalpy and mass flux over small increments of time. These equations are derived from the conservation equations for mass, momentum, energy and the ideal gas law together with plume models, vent flow equations, radiation and combustion models. Forney and Moss (1992) reviewed that there are 11 variables to be solved; the mass, internal energy, density, temperature and volume for the upper and lower layers (M_u, E_u, ρ_u, T_u, V_u and M_L, E_L, ρ_L, T_L. V_L), and the pressure P. Because there are seven constraints, any four of those variables have to be chosen as solution variables. The four variables solved are the pressure above the reference value, volume of the upper layer, and temperature of upper and lower layer. The fire environment in the tunnel was simulated by CFAST version 3.01.

4.1.2 CFX

CFX, a general purpose commercially available CFD code, can simulate fire and smoke movement in three-dimensions, for steady state and time-dependent applications. The set of equations, which describe the processes of momentum, heat and mass transfer, are known as the Navier-Stokes equations and are given in chapter 2, section 4 (2.4.1). These are partial differential equations, which have no known general analytical solution but have to be solved numerically. CFX uses finite volume technique to solve these equations.

4.2 NUMERICAL SIMULATION

For simulation, a tunnel 150 m long, 10 m wide, 8 m high with opening at both ends is considered. The section of the tunnel is shown in Figure 4.1. A burning wood cribs fire is taken to study fire environment inside a tunnel. The fire is taken to be 9 m^2 (3 m x 3 m) in area and 1 m high. The fire source is assumed to be located 20 m from one end of the tunnel i.e. left portal. The heat release rate curve, as shown in Figure 4.2 is taken from experimental results measured by Ingasson et al. (1994) on a wood crib fire under natural ventilation condition as mentioned by Chow (1996).

4.2.1 Simulation with CFAST

The simulation with CFAST has been done by dividing the tunnel into small compartments where these compartments are joined by openings (or vents) having same cross sectional area as the tunnel. This is the same methodology as has been adopted by Chow (1996); however difference lies in

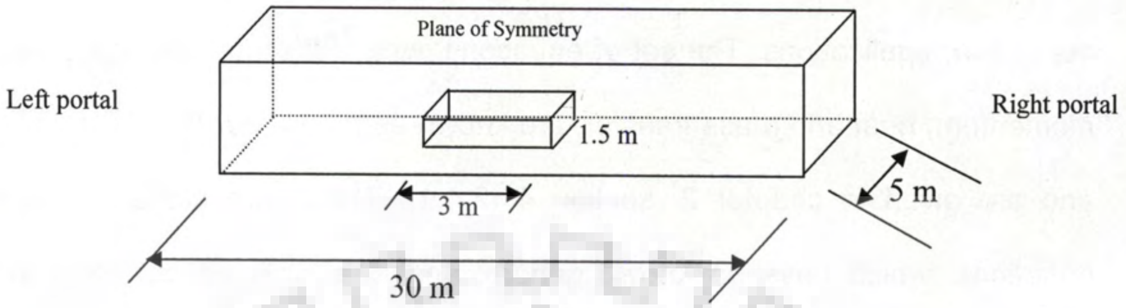


Figure 4.1 A Tunnel Section

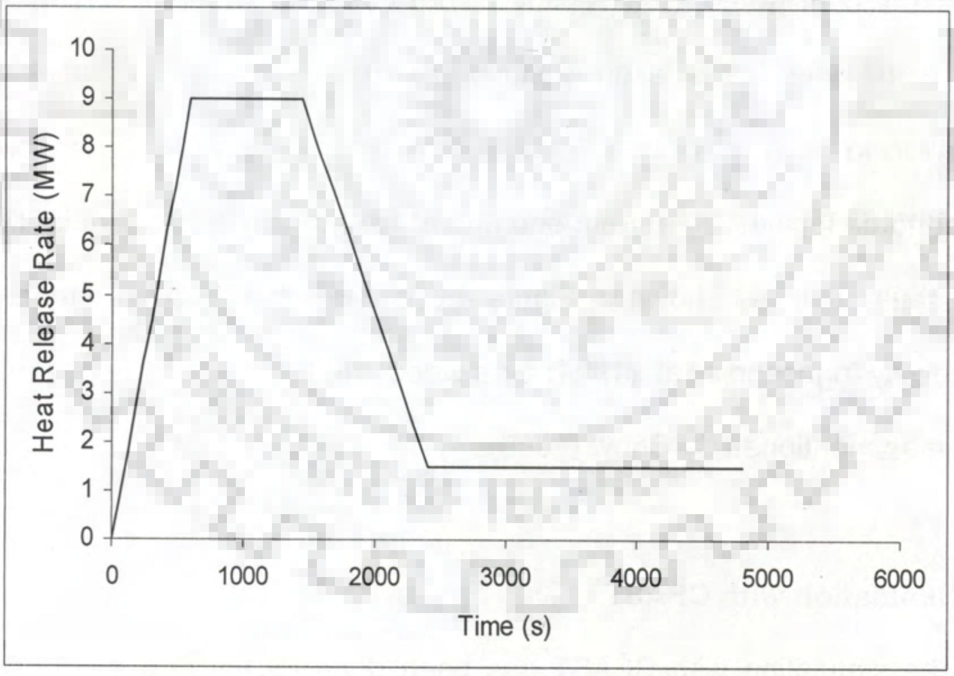


Figure 4.2 Heat Release Rate curve of Wood Crib Fire

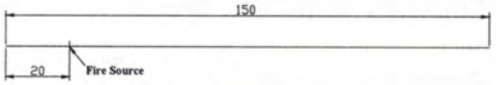
the number of segments the tunnel has been divided into. Chow has carried out simulation by dividing the tunnel into ten segments while in the present work the maximum number of segments in which the tunnel has been divided for simulation is fifteen. Also, this study is more detailed as it studies the effect of location of fire source on predicted environment inside the tunnel.

Initially the tunnel is assumed to be a single compartment of length 150 m (Figure 4.3(a)). The transient variation of smoke temperature and smoke layer interface height is predicted by CFAST. The fraction of heat radiation from fire is assumed to be 0.35. The tunnel was then divided into two equal compartments of 75 m for fire simulation (Figure 4.3(b)). The room containing fire source is called fire room. In the same way, the tunnel was further divided into equal smaller segments of five rooms, eight rooms, ten rooms, twelve rooms, fifteen rooms (Figure 4.3(c)-(g)) and transient variation of smoke layer interface height and temperature for each room is computed. Figure 4.4 and Figure 4.5 predicts fire environment in a tunnel divided into five and eight compartments respectively. Figure 4.6 compares the temperatures of fire room of different cases illustrated in Figures 4.3(a-g).

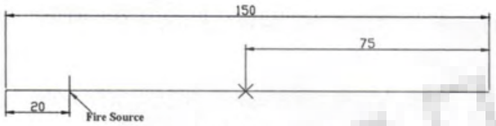
4.2.1.1 Observations

(i) According to literature this version of CFAST can simulate a maximum of thirty compartments; however computational difficulty was encountered when tunnel was divided into more than fifteen compartments.

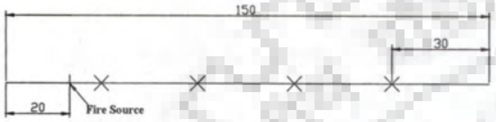
(ii) Results of CFAST in case of a tunnel divided into one and two compartments were compared with those given by Chow (1996) and were found to be in close agreement.



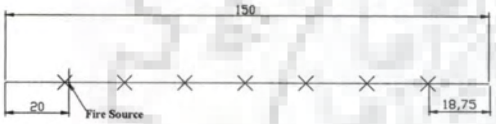
(a) Single Compartment



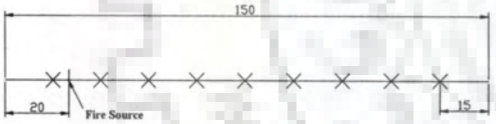
(b) Two Equal Compartments



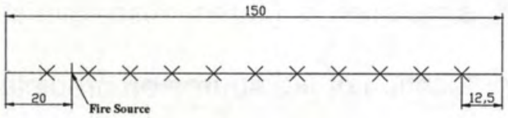
(c) Five Equal Compartments



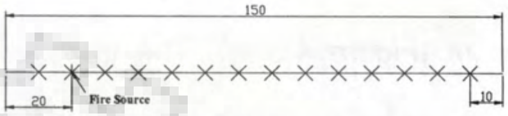
(d) Eight Equal Compartments



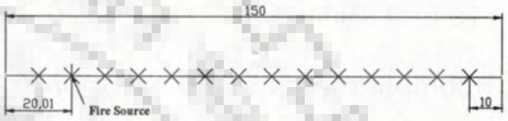
(e) Ten Equal Compartments



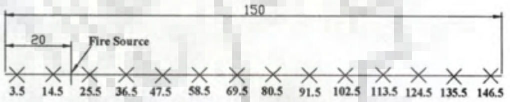
(f) Twelve Equal Compartments



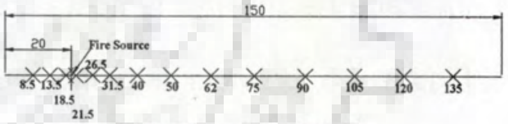
(g) Fifteen Equal Compartments (Case A)



(h) Fifteen Equal Compartments (Case B)

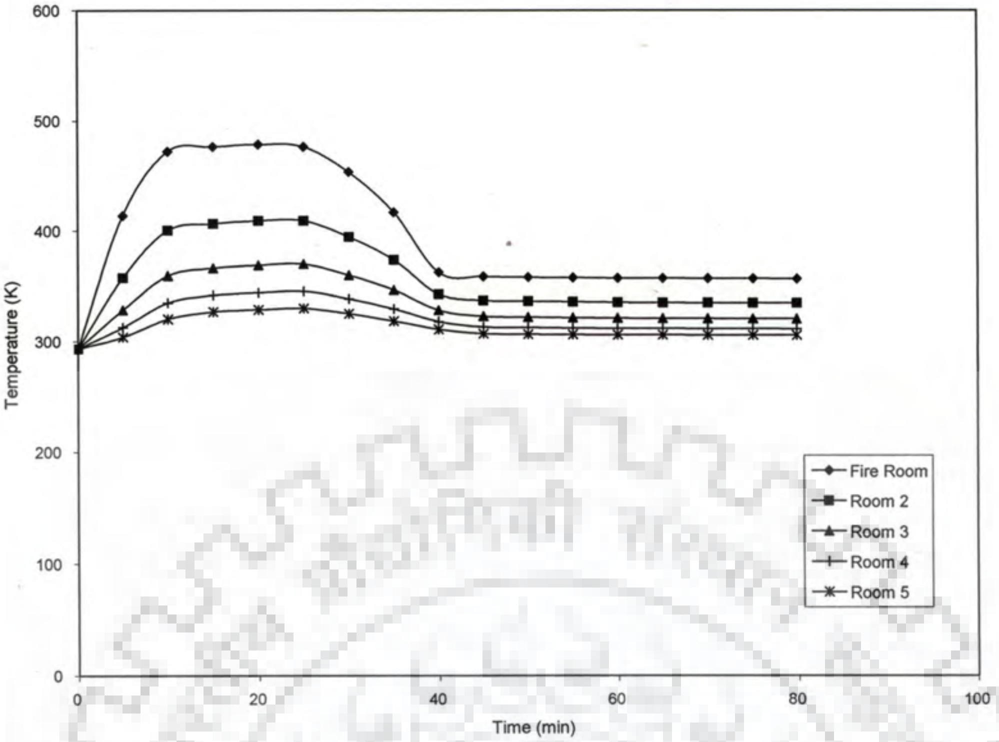


(i) Fifteen Compartments(Case C)

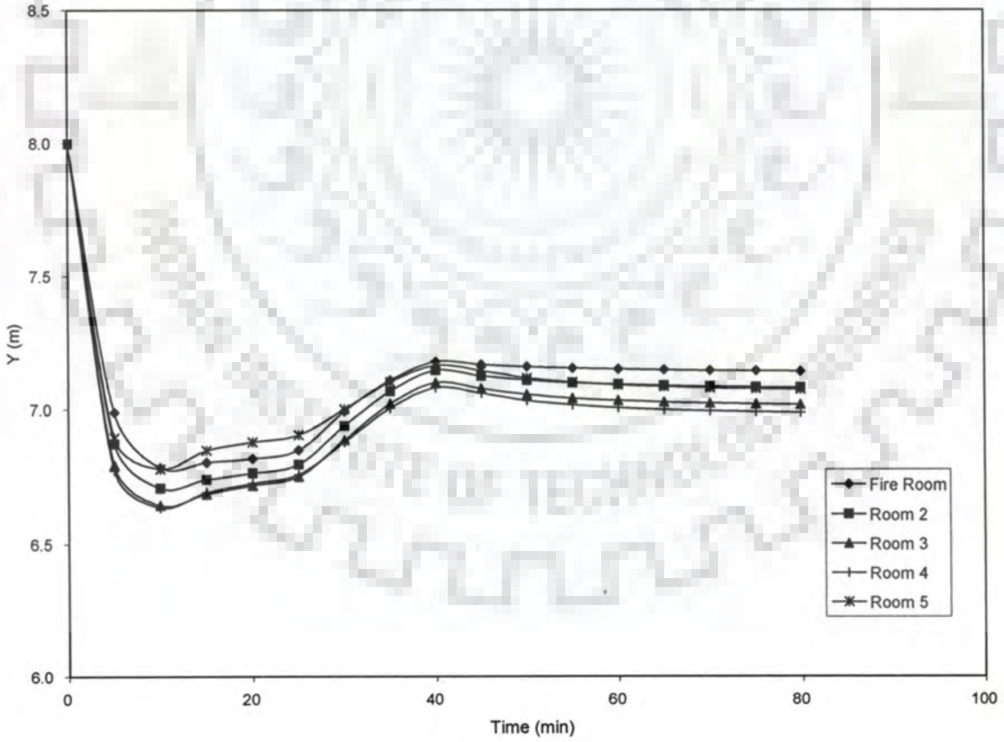


(j) Fifteen Compartments(Case D)

Fig 4.3(a-j) Tunnel divided into compartments

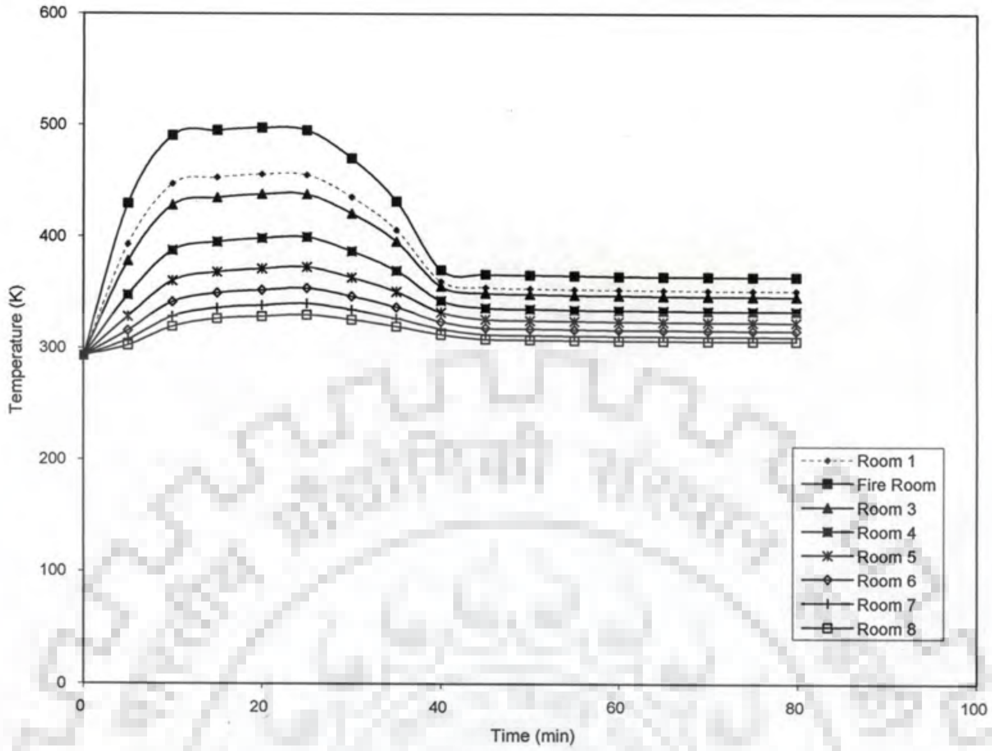


(a) Smoke Temperature

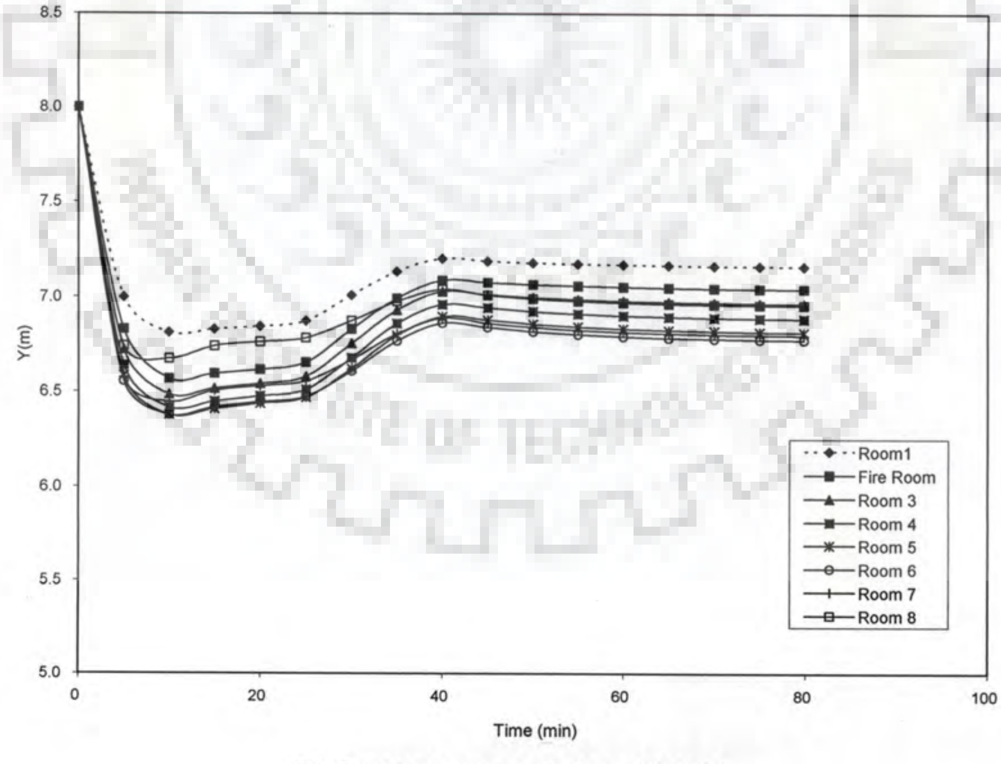


(b) Smoke Layer Interface Height

Figure 4.4 Predicted environment for a five-compartment tunnel



(a) Smoke Temperature



(b) Smoke Layer Interface Height

Figure 4.5 Predicted environment for an eight compartment Tunnel

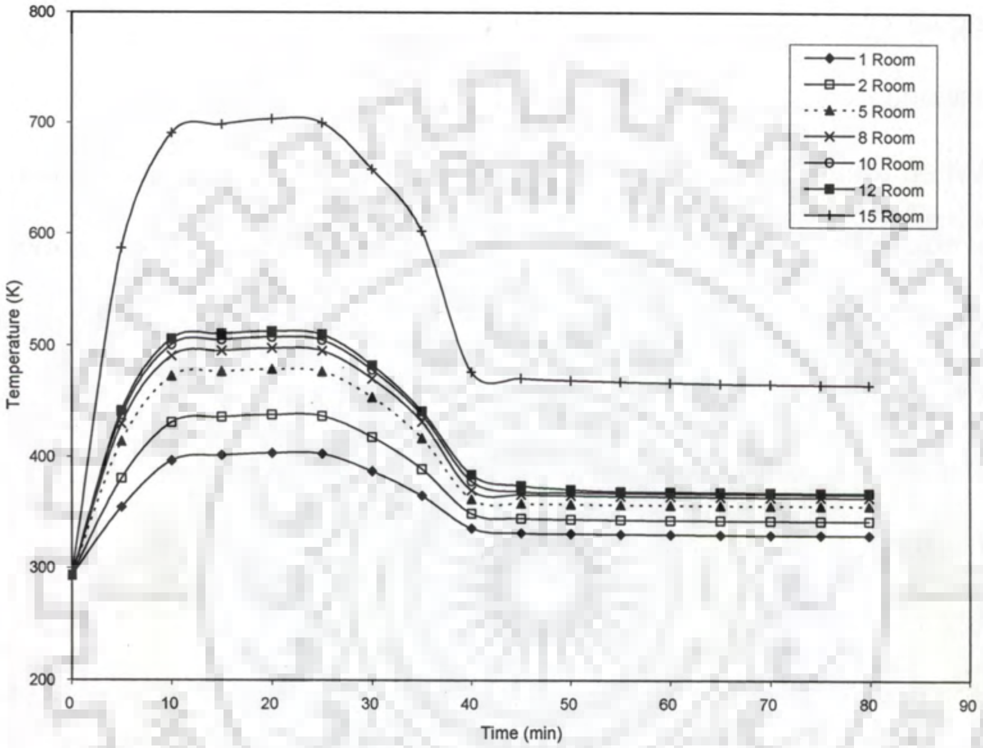


Figure 4.6 Predicted Smoke Temperature of Fire Room of cases as illustrated in Fig 3(a-g)

(iii) The smoke temperature changed with a pattern roughly similar to those of heat release rate. The smoke layer interface height descended in a pattern according to the increase in heat release rate. This is evident from Figures 4.4 and 4.5.

(iv) The fire environment in a tunnel divided into ten, twelve and fifteen equal compartments have been found to be similar to that of tunnel divided into eight compartments.

(v) When tunnel is assumed to be of a single compartment, the maximum smoke temperature reached is 404.58 K in 24 minute. This is expected as heat release rate is maximum i.e. 9 MW from 10 – 24 minutes, after which the heat release rate starts decreasing and so do the temperature of the compartment. The same behavior is observed in the fire room when tunnel is divided into smaller compartments.

(vi) Figures 4.4 and 4.5 shows that the smoke temperature inside fire room is higher than corresponding rooms. As the distance of compartments from the fire room increases, the corresponding temperature decreases. This is due to heat loss to surroundings and walls. The same behavior is observed in case of tunnel divided into ten, twelve and fifteen compartments.

(vii) In case of tunnel divided into eight equal compartments, the fire room is the second compartment, which lies at a distance of 18.75 m from nearest portal (left). The smoke temperature in the first compartment is lower than that of fire room but higher than that of third, fourth etc compartments. This is because of cold air entraining from left portal resulting in lower temperature.

(viii) The minimum value of smoke layer interface height for fire room increases as the number of compartments in which tunnel is divided decreases. This is because as number of compartments in which tunnel is divided decreases, the room sizes increases, resulting in increase in area of room and therefore the given volume of smoke will occupy less height from the ceiling implying that the minimum smoke layer interface height increases.

(ix) The value of smoke temperature of fire room is highest in case of tunnel divided into maximum number of compartments i.e. fifteen. This temperature decreases as the number of compartments in which the tunnel is divided decreases. Figure 4.6 shows this behavior.

(x) The temperature of fire room in case of tunnel divided into fifteen equal compartments (say case A) shows remarkably high temperature relative to temperature of fire room in case of tunnel divided into twelve compartments. This can be because of fire source being placed at the end of second room which lies at the position of vent. To check the accuracy of these results another simulation is carried out in which tunnel is divided into fifteen equal compartments and fire source is assumed to be in third room (Figure 4.3 (h)) instead of second room and position of fire source is taken to be 0.01 m away from the vent (vent connecting second and third room), all other conditions remaining the same (case B). The transient behavior of smoke temperature inside fire room is shown in Figure 4.7. The results show that the temperature of fire room is little higher than the temperature of fire room in case of tunnel divided into twelve compartments.

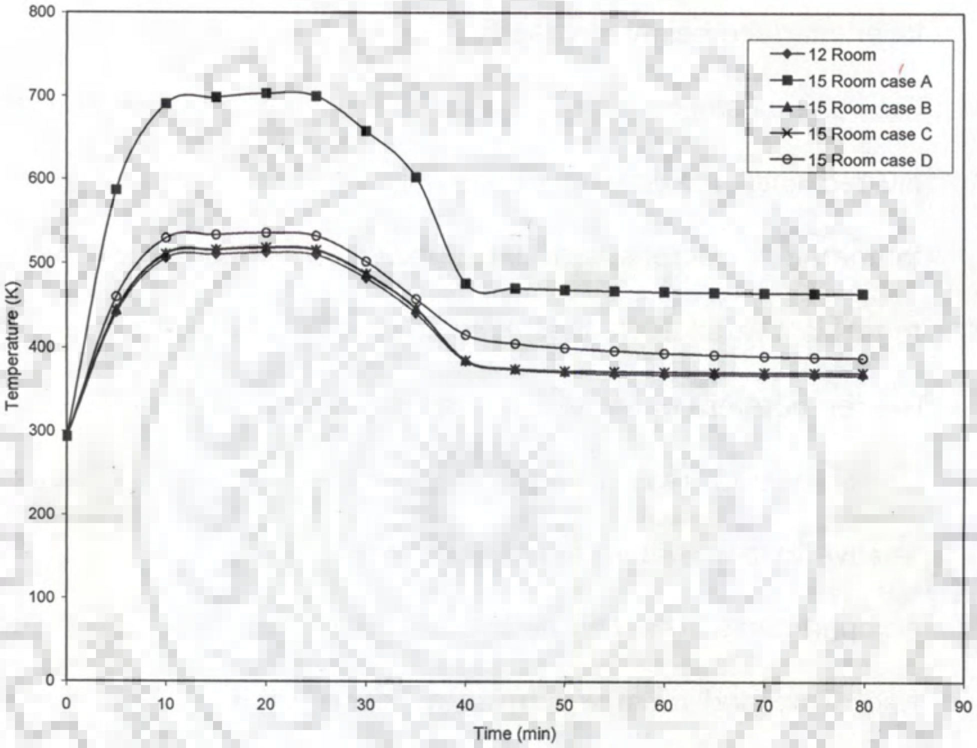


Figure 4.7 Predicted Smoke Temperature of Fire Room of different cases of 15 compartment tunnel.

(xi) To further consolidate the results of temperature of fire room in case of tunnel divided into fifteen segments, another simulation was carried out (case C). In this simulation, the room sizes are taken to be unequal (Figure 4.3(i)). The fire source is positioned at the center of room three, assumed to be of size 11 m. The first and fifteenth rooms are assumed to be 3.5 m in length and all the other rooms are assumed to be 11 m in length. The results are shown in Figure 4.7. The temperature is close to temperature of previous simulation i.e. case B. This implies that the temperature predicted by CFAST in case A when fire room is second room and position of fire is at vent appears to be wrong. It may be concluded that the fire source should not be positioned at vent position for computation by CFAST.

(xii) In order to check whether higher temperature is achieved when a tunnel is divided into fifteen segments and size of fire room is reduced, another simulation (case D) is carried out in which fire room is the fourth room of length 3 m and fire source is kept in center of room (Figure 4.3(j)). The length of rooms is arbitrarily assumed. The 1st, 7th room is 8.5 m long, 2nd, 3rd, 5th, and 6th room are 5 m long each, 8th room is 10 m long, 9th room is 12 m long, 10th room is 13 m long and 11th to 15th rooms are each 15 m long. The transient behavior of temperature of fire room in this case is higher than those of previous two cases as shown in Figure 4.7. This is as expected. The fire starts in a room and releases energy and products of combustion. Since volume of room is less therefore energy is confined to smaller volume resulting in increased temperature of that volume.

For simulation by CFAST, the tunnel was divided into equal compartments. In case of tunnel divided into fifteen compartments the fire source lies at position of vent, CFAST predicted relatively higher temperatures. The tunnel was then divided into unequal compartments in such a way that location of fire is at center of compartment and size of fire room is less than when the tunnel was divided into equal compartments. This simulation predicted higher temperature in fire room. This also confirmed that as size of control volume decreases better predictions could be achieved. Since the size of compartments taken has no basis when the tunnel was divided into unequal compartments, hence three more trials of uneven sized rooms were carried out. It was found that the maximum variation of smoke temperature is of the order of 3°C, hence these trials are not reported. It was concluded that for better results the size of rooms should be so chosen that fire source is placed as far as possible in the center of room and not at position of vents.

4.2.2 Simulation with CFX

The same tunnel geometry has been simulated using CFX solver 5.7 in a PIV 1.83GHz, 1GB RAM Compaq machine. The computational domain is symmetric and so the simulation is carried out only on one half of the tunnel i.e. tunnel of length 150 m, height 8 m and width 5 m. Since the symmetry plane divides the fire into two equal halves, the heat release rate of fuel used in calculation is half of that mentioned above. The computational domain is extended longitudinally, on both sides, outside the tunnel exit to a free boundary condition at a vertical plane parallel to tunnel exit plane. There are two types of boundaries, solid boundary where no slip condition is assumed and free boundary where fixed pressure is assumed. The free boundary is also used at ceiling height outside the tunnel. The physical models used in simulation are as follows:

(i) Combustion: Fire is modeled as a volumetric heat source (VHS) model which does not take chemical reaction into account. Instead it sets heat release rate equivalent to that of the assumed fire.

(ii) Turbulence: Turbulence is accounted for by standard k - ε model.

(iii) Buoyancy: Buoyancy effects have been incorporated in the simulations.

(iv) Wall Function: Wall Function Approach has been used to model the flow in the near wall region.

(v) Radiation: Radiation is modeled using six flux method as suggested by Sinai (2003) in Steckler experiments.

The simulation is carried out for 80 min with a time step of 1 second. The velocity vectors of a portion of tunnel near fire source on the central symmetry plane are shown in Figure 4.8, while temperature contours at 20 minutes from start of simulation are shown in Figure 4.9. The smoke temperatures at different distances from the portal have been measured on central symmetry plane i.e. z=0.001 m and height of 7 m and 6.5 m. These are shown in Figure 4.10 (a,b).

4.2.2 Observations

(i) Figure 4.10 shows that smoke temperature changes with a pattern roughly similar to that of heat release rate.

(ii) The maximum temperature of smoke lies at a distance between 25 m and 35 m and not above center of fire source. This indicates that there is a flame tilt towards right. This has also been observed from the

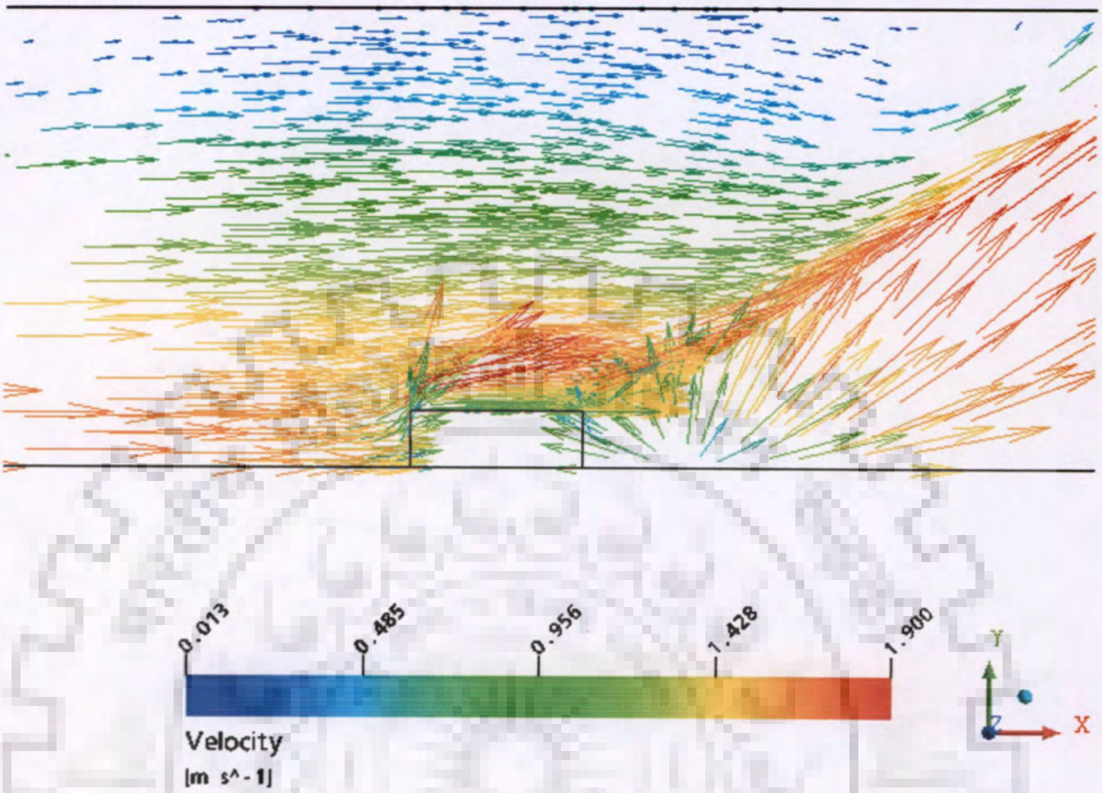


Figure 4.8 Velocity vectors on central symmetry plane.

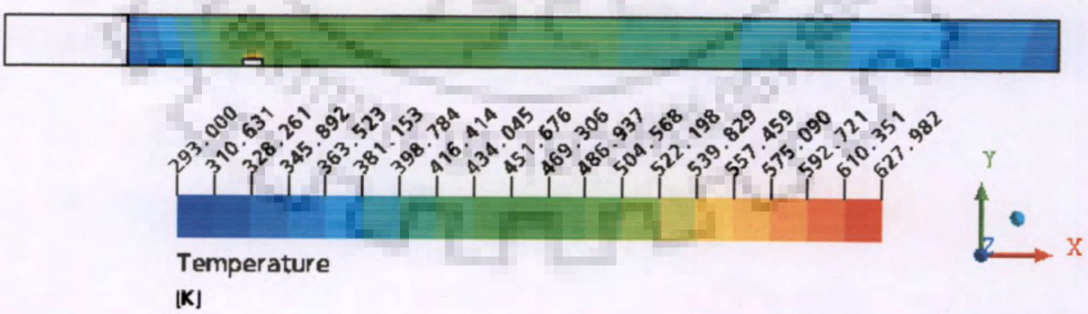


Figure 4.9 Temperature contours at symmetry plane at time 20 min

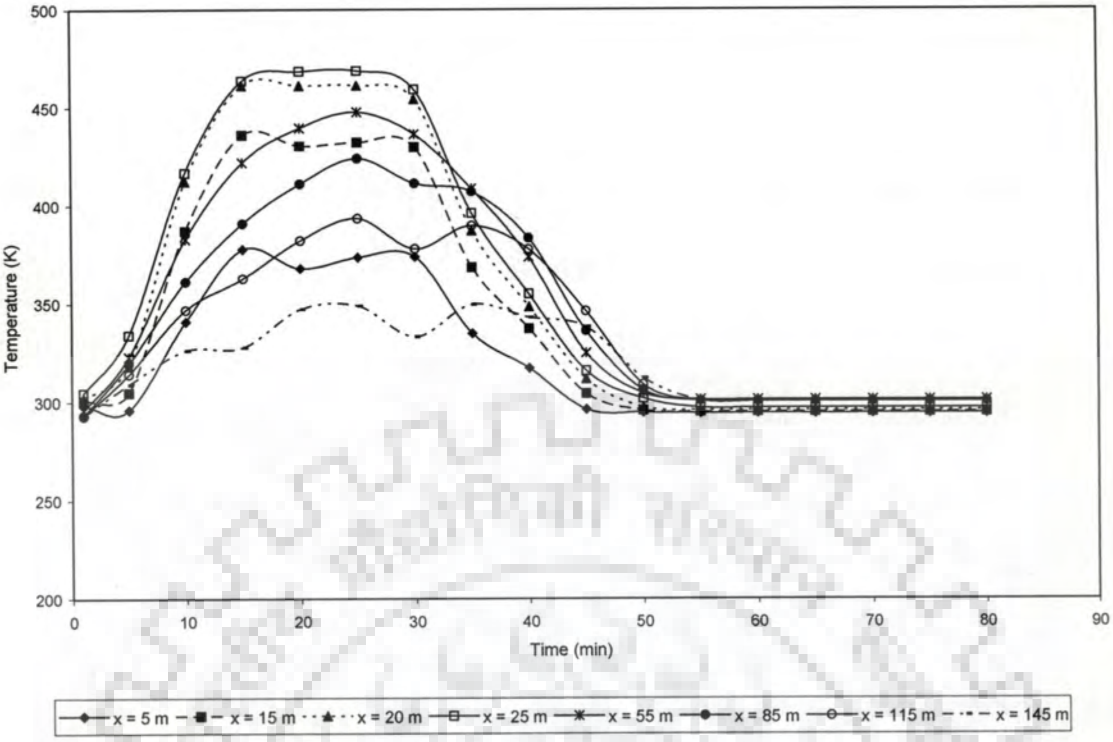


Figure 4.10(a) Predicted Smoke Temperature by CFX at Monitoring Points at different x, y = 7 m and z = symmetry plane

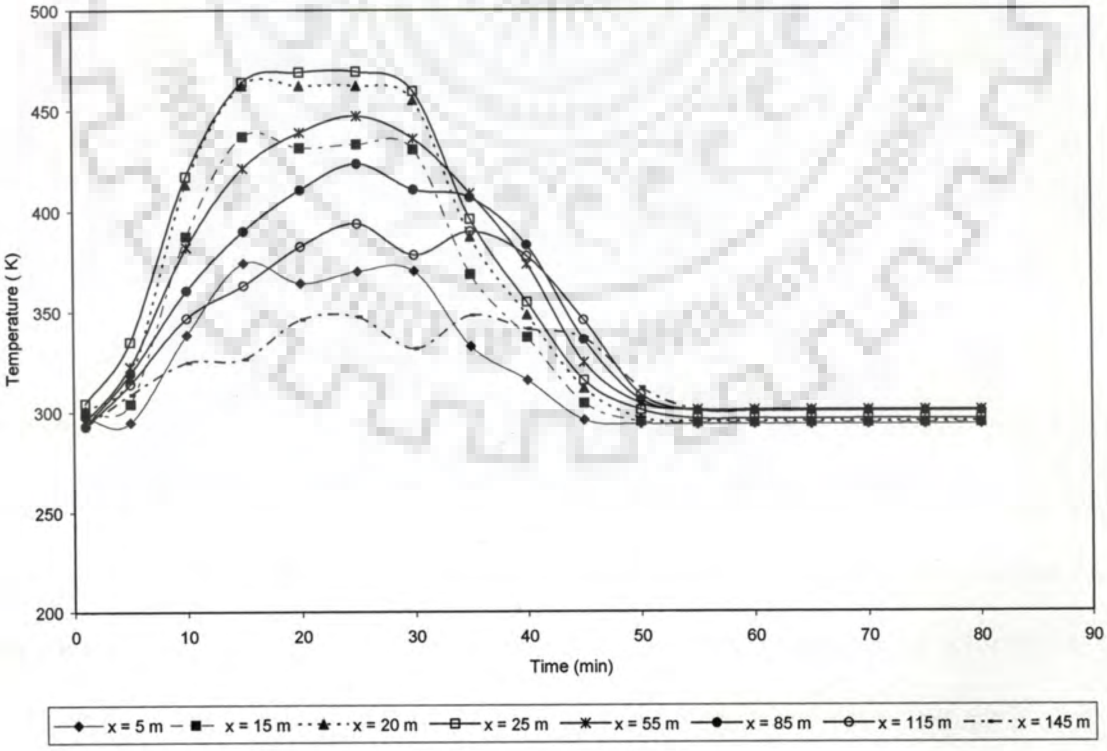


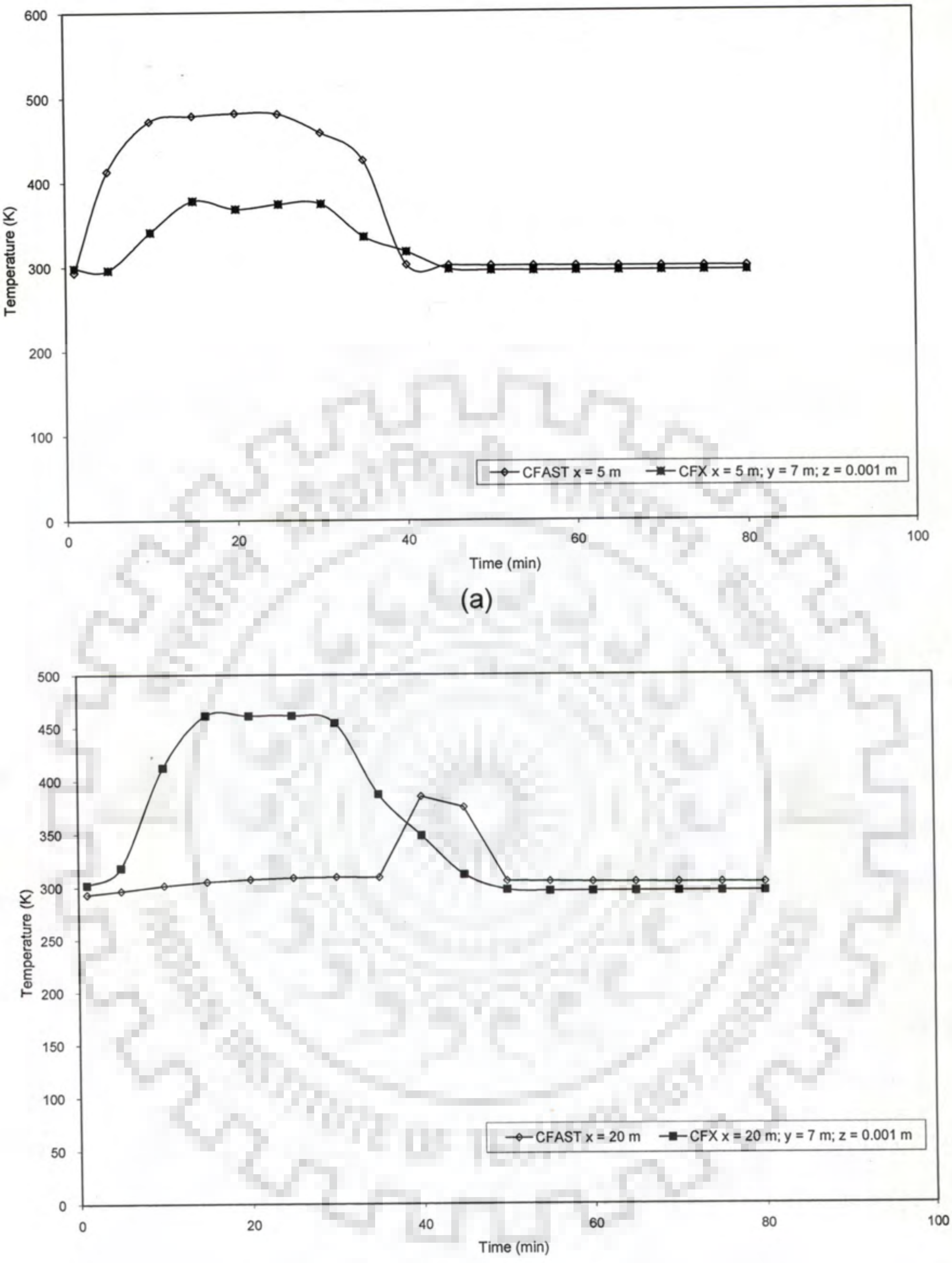
Figure 4.10(b) Predicted Smoke Temperature by CFX at Monitoring Points at different x, y = 6.5 m and z = symmetry plane

temperature contours. This happens because fresh air is blown towards the fire source more from the left side, which is closer to exit, forcing flame to tilt towards right.

(iii) The velocity vectors shown in Figure 4.8 depict the effect of natural ventilation on flow field. The smoke from the fire source moves upward and then side ways in both directions. The fresh air is entrained from both sides towards the fire source. It lies beneath the hot smoke. Since the left side is closer to exit hence airflow from left side is more dominant and it forces the flame to tilt towards right.

4.3 COMPARISON OF CFX RESULTS WITH THOSE OF CFAST

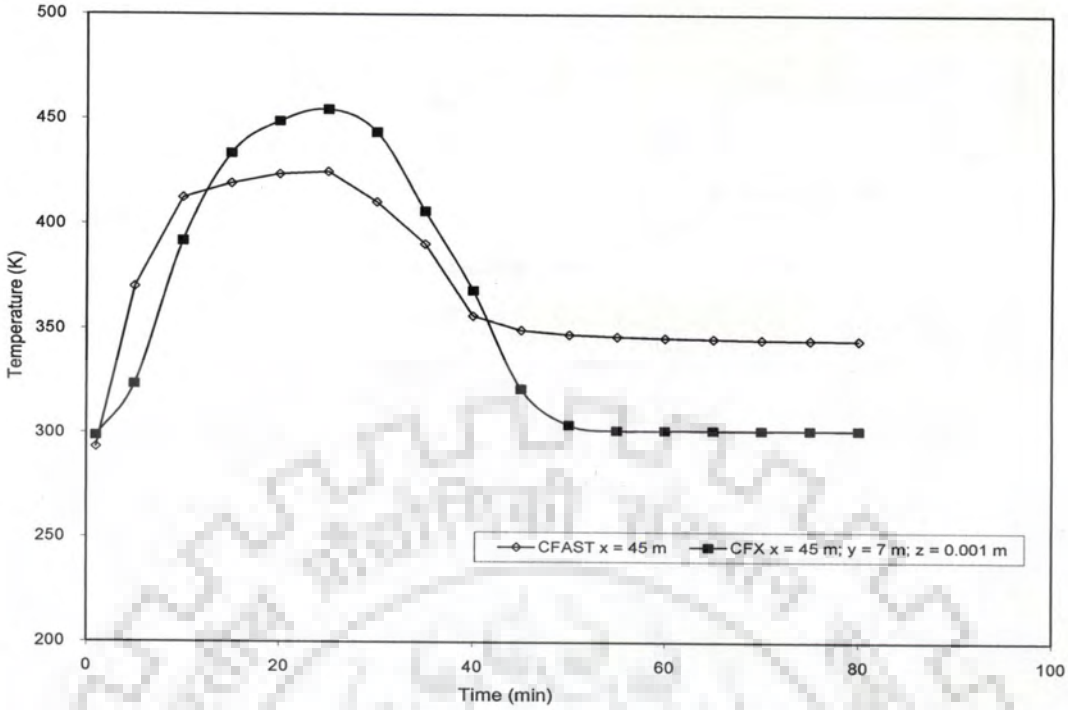
The predictions of smoke temperatures from CFAST are compared with that of temperatures predicted from CFX and are shown in Figure 4.11. The temperatures are measured in CFX at central symmetry plane at a height of 7 m and distance of 5 m, 20 m, 45 m and 135 m, the distances being measured from the exit portal closer to fire source i.e. left portal. In CFAST, the tunnel is divided into small segments. For comparison the tunnel with fifteen segments (case C) is taken. To compare temperatures, say at 45 m length, temperatures of that room are taken in which the distance of 45 m from left portal lies, in this case it is the fifth room. CFAST assumes that there are two layers inside a room, a upper hot layer and a lower cold layer and temperatures of each layers are uniform throughout. For comparison, the temperature of upper layer is taken if the smoke layer interface height is below 7 m and lower layer temperature is taken if smoke layer interface height is above 7 m. The height of monitoring



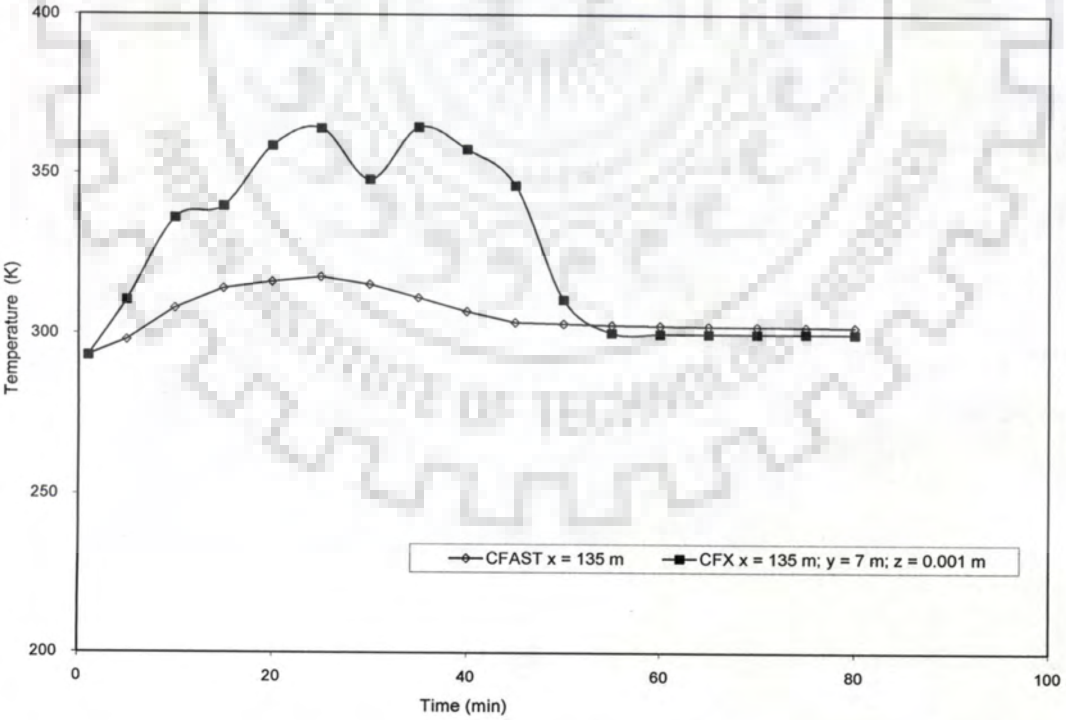
(a)

(b)

Figure 4.11(a,b) Comparison of Predicted Temperatures of CFX with CFAST at Monitoring Points



(c)



(d)

Figure 4.11(c,d) Comparison of Predicted Temperatures of CFX with CFAST at Monitoring Points

points is chosen as 7 m because the minimum smoke layer interface height as predicted by CFAST lies above 6.5 m in most of the cases, so if monitoring points height have been chosen to be less then 6.5 m than temperature of lower layer would be mostly used for comparison. The temperatures as predicted by CFAST and CFX are shown in Figure 11(a-d).

4.3.1 Observations

(i) Figure 4.11(a) shows that at a distance of 5 m, CFAST temperatures over predicted the temperatures of CFX. After about 40 min the temperatures predicted by CFAST are close to those by CFX.

(ii) Figure 4.11(b) shows that at a distance of 20 m, the temperatures predicted by CFAST are less than that predicted by CFX. However, CFAST temperatures shoot up at about 40 min and then decreases. This is because at that point the smoke layer interface height becomes more than 7 m and than falls. Thus upper layer temperatures are taken for those points resulting in large overshoot.

(iii) At a distance of 45 m, CFAST under predicted the temperatures for first 40 min by about 25 K for position of maximum heat release rate. After 40 min, the CFAST temperatures are more than CFX temperatures. This is shown in Figure 11 (c).

(iv) Figure 4.11 (d) shows that at a distance of 135 m, CFAST temperatures under predicted CFX temperatures.

(v) Figure 4.12(a, b) shows the temperature distribution predicted by CFX and CFAST along the tunnel axis, at a height of 7 m at 25 min and 70 min respectively. The CFX results are shown as continuous line while CFAST

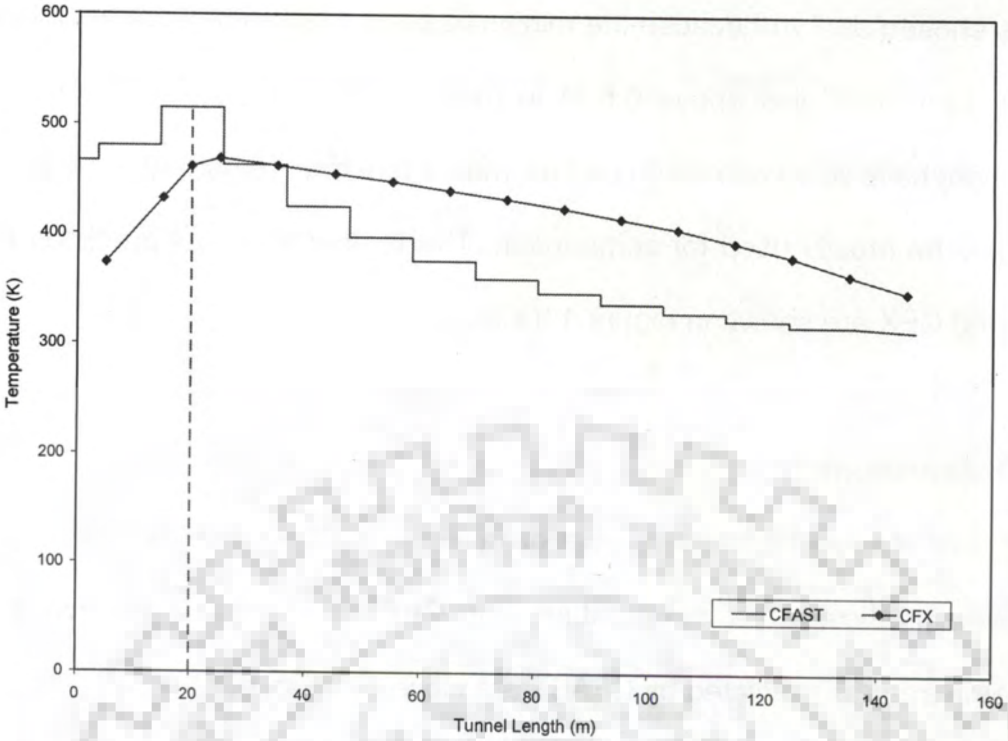


Fig 4.12(a) Temperature Distribution along Tunnel Axis at H = 7 m and t = 25 min

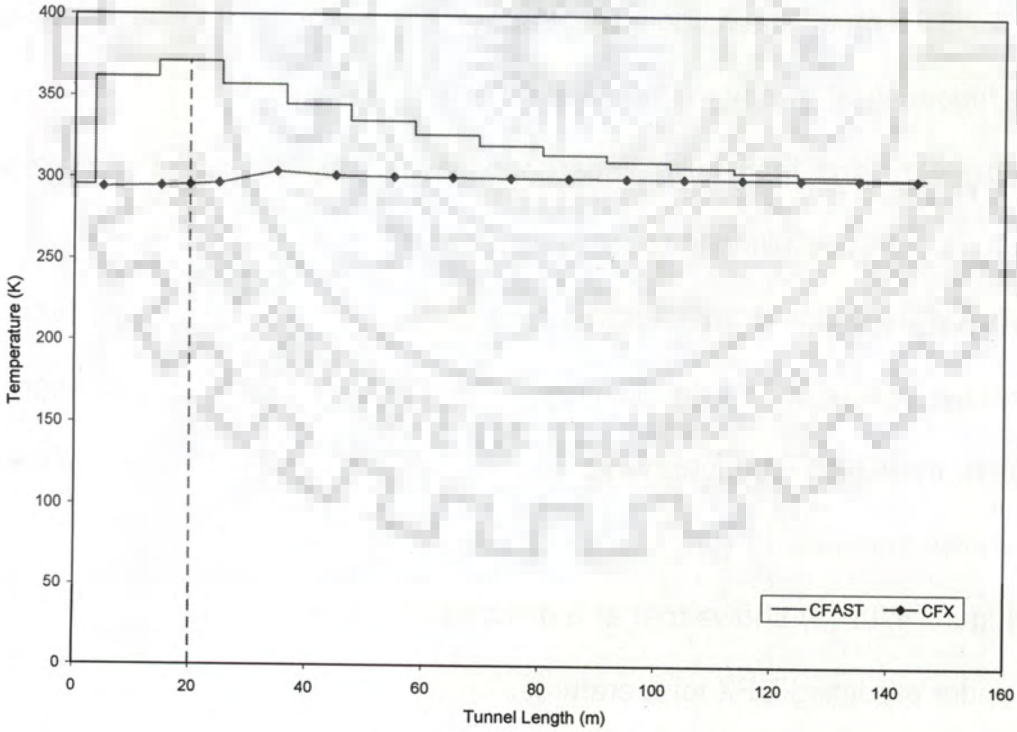


Fig 4.12(b) Temperature Distribution along Tunnel Axis at H = 7 m and t = 70 min

results are shown as stepwise distribution. The fire source is at 20 m, which lies in the third room. CFAST results show that the temperature in fire room is highest at both 25 min and 70 min. However, CFX results predicted highest temperature away from the fire source, towards right. This shows that there is a flame tilt towards right. This can also be seen from movement of velocity vectors shown in Figure 4.8.

4.4 CONCLUDING REMARKS

The fire environment inside a tunnel caused by burning wood crib has been simulated by a zone model CFAST and a field model, CFX. During computation by CFAST, location of Fire source in the compartment is a critical issue. It is observed that for accuracy of results the size of compartment should be so chosen that fire source is placed in the centre of compartment as far as possible; in no case it should be positioned at or near vents.

In CFAST, the temperature of monitoring point is taken to be upper layer temperature if the smoke layer interface height is below the monitoring point and lower layer temperature is considered if smoke layer interface height is above the monitoring point. This results in sharp increase or sharp downfall of temperatures. It can thus be stated that CFAST can only provide global view in terms of two layers; it may not provide temperature profiles at selected locations.

In CFAST the maximum number of compartments that can be simulated is thirty (Jones & Forney, 1990), but computational difficulty was encountered when tunnel was divided into more than fifteen compartments. Thus, the maximum number of compartments in which tunnel can be divided for simulation purpose should be less than or equal to fifteen.

CFX, on the other hand provided detailed information regarding smoke movement inside a tunnel. It not only provided temperatures at various locations but it also predicted the movement of smoke. It can be seen that CFD analysis offers detailed insight into fire dynamics. It can thus be helpful in analyzing problems involving far field smoke flow, impact of fixed ventilation flows. It can thus prove to be an essential tool for systematic design of fire protection measures. However, these require high computational power and are time consuming. On the other hand, modeling techniques used by CFAST require very less computational power and are quick to perform, hence can be used to get overview of the results.

CHAPTER - V

NUMERICAL STUDIES ON EVALUATION OF VENTILATION STRATEGIES OF UNDERGROUND METRO RAIL TRANSPORT SYSTEM WITH JET FANS

5.0 INTRODUCTION

The last chapter showed that CFD is a better tool and can be used for analyzing problems involving far field smoke flow, impact of fixed ventilation flows etc. Therefore, the fire dynamics inside the tunnels are studied using Computational Fluid Dynamic (CFD) technique. In the present chapter, ventilation strategy of a tunnel section, similar to Delhi Metro tunnel section and fitted with jet fans are evaluated. The study is carried out in two parts.

In the first part of study, the numerical model to be used is first verified using experimental results from Steckler experiment. The model is then used to simulate the fire environment inside the tunnel. The results of CFD simulations are compared with empirical correlations available in the literature.

In the second part, effectiveness of ventilation system to control smoke spread in the event of fire inside the tunnel, is studied. For this, the effect of ventilation flow rate on thermal environment inside the tunnel is studied. Two scenarios of ventilation are studied. In scenario 1, the induced airflow is assumed to be uniform; this corresponds to a case, where fans are located at sufficient distance from the fire source. In second scenario, the fans are assumed to be located near the fire source resulting in non-uniform airflow. The effect of operating exhaust fans in addition to supply fans is also studied for this scenario. The critical velocity necessary to prevent back layering is determined.

5.1 STECKLER ROOM FIRE EXPERIMENT

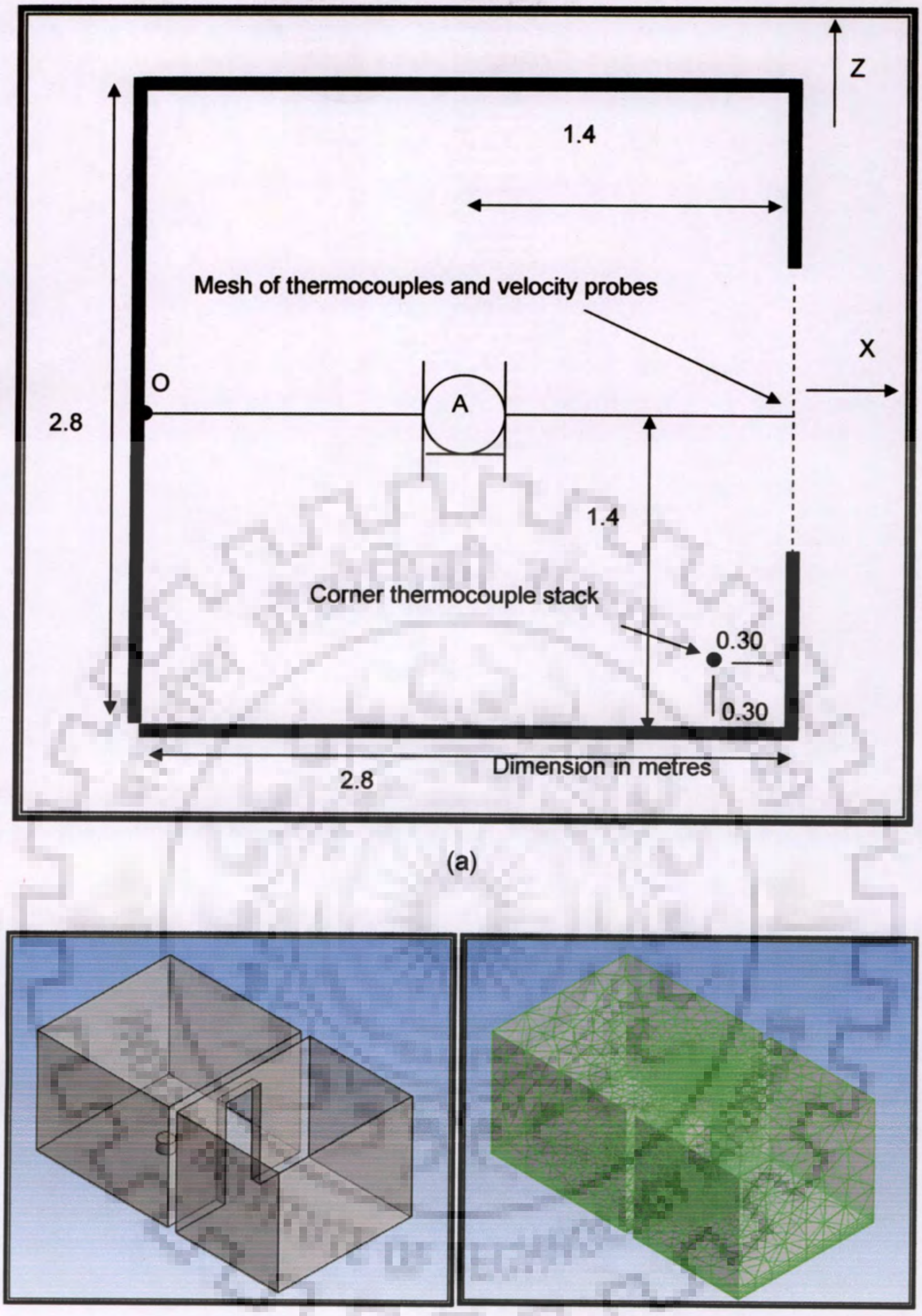
The experiment being considered is one of many performed by Steckler et al. (Steckler et al., 1982; Sinai, 2003). It involved a single, cubical compartment, 2.8 m x 2.8 m in the horizontal plane, and 1.83 m high. A door, 0.74 m wide and 1.83 m high, is in the middle of a sidewall. The walls are made of concrete, 0.1 m thick, and insulated on the interior by low conductivity lining. A circular burner is located in the center, with its top flush with the floor. The diameter of the burner is 0.3 m. The burner supplied commercial grade methane at a fixed rate, which provided a theoretical heat output of 62.9 kW. At the centre of the doorway, a tree of instruments measured speed and temperature. Another thermocouple rake measured temperature in a vertical line, which is 0.305 m from the two walls meeting one of the two corners. The plan view of Steckler experiment is shown in Figure 5.1(a).

5.1.1 Model Verification:

The CFD model consists of the enclosure and an extension of connected space, outside the door. The extended domain ensures realistic simulation of the plume outside the door opening. The three dimensional computational domain along with sample control volume is shown in Figure 5.1(b). The computations are done using CFX 10 on a Compaq PIV 1.83 GHz, 1GB RAM machine. The physical models used in CFD model are described below:

(i) Turbulence: The flow is assumed to be steady and in fully turbulent regime. The standard two-equation k-ε model of turbulence with buoyancy modification is used.

(ii) Wall Function: A standard wall function is used to estimate the turbulence kinetic energy and dissipation at the cell near the wall.



(a)

(b)

Figure 5.1(a) Plan view (b) computational domain, sample control volume of Steckler experiment (shown with lesser number of nodes for clarity)

(iii) Radiation: Thermal radiation is represented by P1 or six flux model.

(iv) Boundary condition: The free pressure boundary conditions are applied on the extended domain boundary, where conditions are assumed to be ambient. All walls are assumed to be smooth, adiabatic and gray, with an emmisivity of 0.5. At solid boundaries no slip condition is assumed. The supply duct is treated as inlet and exhaust duct as outlet. In case when fans are not working, the boundaries of these ducts are treated as walls.

(v) Combustion: Combustion is not modeled. Instead the fire source is represented as a volumetric heat source (VHS) where the volumetric fire source is modeled as a cylinder of diameter 0.3 m, which is based on the diameter of burner. The spatial extent of the heat source has been assumed for the VHS model. Initially the height of volumetric heat source is taken to be 1 m. It is subsequently reduced to 0.2 m and 0.05 m.

5.1.2 Observations

(i) The spatial distribution of the heat released within the fire plume where the height of the volumetric heat source has been reduced from 1.0 m to 0.05 m, but keeping the same fire base area is shown in Figure 5.2. It shows the temperature distribution on the vertical central plane through the fire source and door opening. It can be seen that the increased height of the volumetric heat source prevents the tilting of the fire plume towards the wall, which is caused by the cool air jet entrained from the bottom of the door opening. The plume shape appears more realistic for short heat source height i.e. 0.2 m. This is because the fire tilts towards the back wall, forced by the makeup air through the door.

(ii) The maximum temperature inside the compartment for different height of heat source is shown in Table 5.1.

Table 5.1 Maximum temperature in compartment using VHS of various source heights

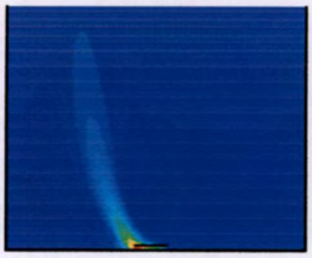
Fire source volume height (m)	1.0	0.2	0.05
Temperature (K)	660	1160	2400

This table shows that the peak temperature inside the fire plume increases with decreasing height over which heat is released in the VHS model. From Table 5.1, it is also observed that for low heights of the volumetric heat source, peak temperatures inside the fire plume are unrealistically high because the heat is distributed over very low source volume. It has been observed that the large differences in the predicted maximum fire source temperature in the VHS model (shown in Table 5.1) do not have significant influence on the predicted temperature distribution away from the fire source i.e. at the door centerline and at corner.

(iii) Empirically, the height of flame given by Heskestad (1995) is defined as

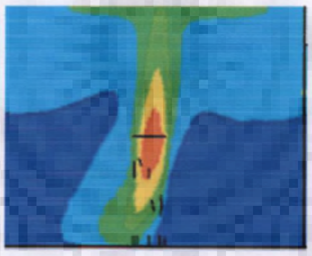
$$Y_f = 0.235 * Q^{0.4} - 1.02 * D_f \qquad (5.1)$$

For a fire of 62.9 kW and diameter of 0.3 m, this correlation estimates the flame length to be 0.93 m which is close to assumed length of 1 m. This



(a) Heat source height = 0.05 m

(b) Heat source height = 0.2 m



(c) Heat source height = 1.0 m

Figure 5.2 Predicted temperature contours on the vertical plane across center of the room of Steckler experiment for various heights of the volumetric heat source in VHS model length also gives more realistic temperatures. Thus, a reasonable assumption in case of VHS model is that the height of the source should be close to that of flame length.

The results presented above shows that it is very important to assume suitable height and volume of the volumetric heat source, appropriate to the heat release rate of the fire, in order to have reasonable prediction of the flow pattern and the flame temperatures (S. Kumar, Personal Communication, 14th March 2007; Wang et al., 2006).

5.1.3 Comparison with Experimental Results

The velocity and temperature profiles at the door centerline and at corner predicted by CFX are compared with experimental results and shown in Figure 5.3 (a-c)). In the Figure 5.3(b), the positive sign for horizontal velocity means flow out of the enclosure. In general, the numerical results are found to be in good agreement with the experimental results.

The airflow and heat transfer through the enclosure door are more relevant for fire within a tunnel. Therefore, the agreement with the experimental results provides a verification of the numerical model to evaluate fire environment inside a tunnel.

5.2 FIRE INSIDE TUNNEL (NO VENTILATION)

The tunnel section considered is similar to that of Delhi Metro Rail section. The section considered is 100 m long, 6 m wide and 9 m high. The dimensions taken are approximate to actual dimensions. At each end of tunnel portal within

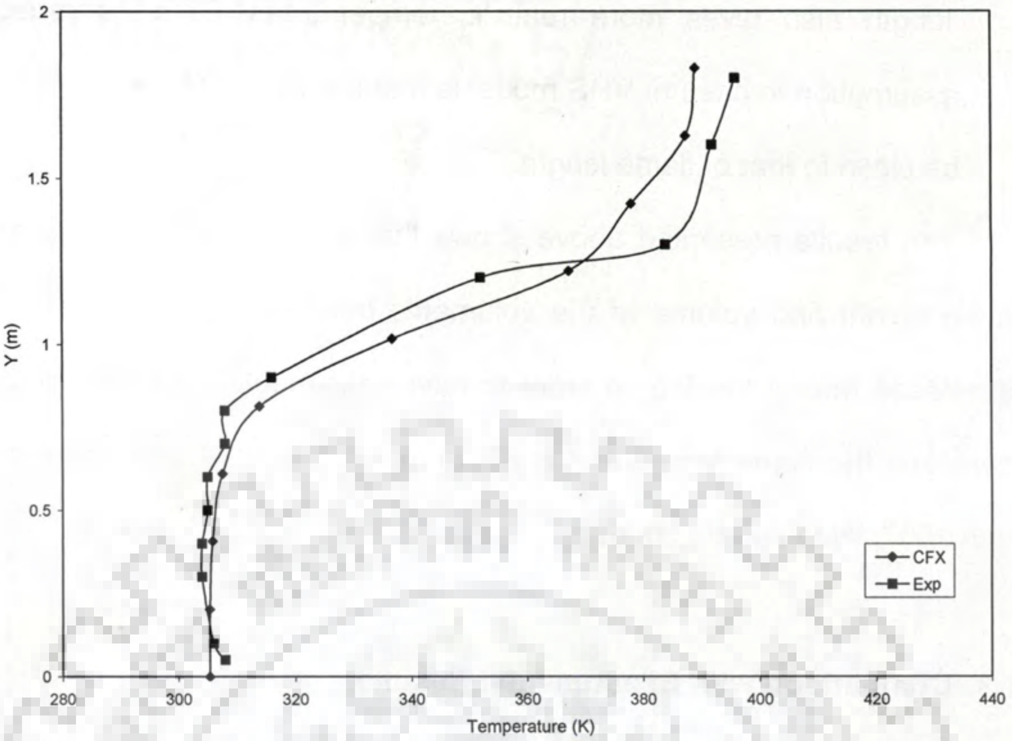


Figure 5.3(a) Comparison of Predicted and Experimental Temperatures at door centerline of Steckler experiment

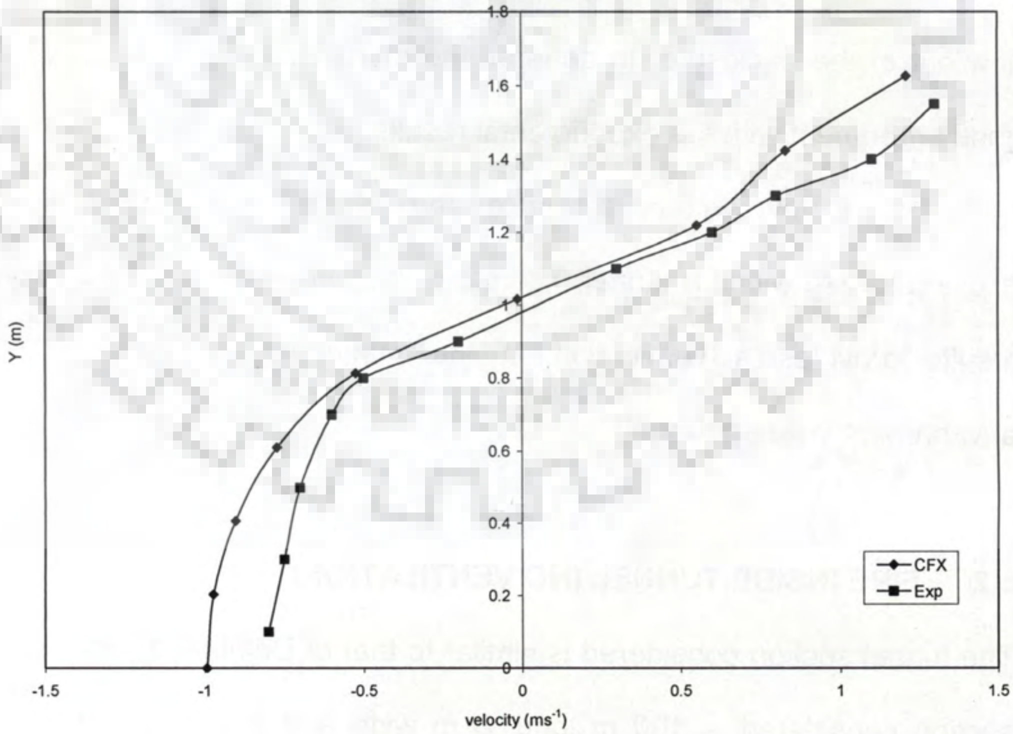


Figure 5.3(b) Comparison of Predicted and Experimental velocity at door centerline of Steckler experiment.

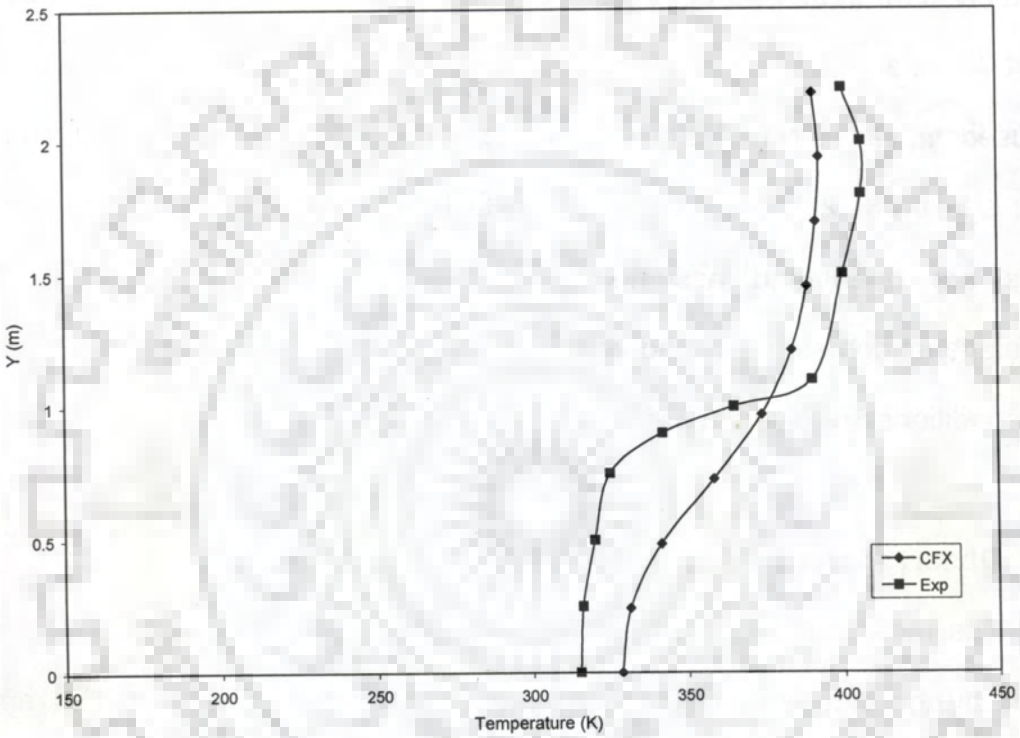


Figure 5.3(c) Comparison of Predicted and Experimental Temperatures at Corner of Steckler experiment

the track way, a jet fan 3 m x 2 m is located below the ceiling to assist in controlling air movement within the track ways. All walls are assumed to be smooth, adiabatic and gray. A view of the computational domain of modeled section is shown in Figure 5.4. For all simulations, a fire with a constant heat release rate of 4000 kW is used to represent vehicle fire. The fire source is represented by a solid block whose base area is assumed to be square of side 3.53 m (12.5 m^2 in area) and located at the middle of the tunnel. The fire source is kept 0.5 m above the floor level. The physical models used are the same as that used in validation of Steckler experiment. The Volumetric heat source height is taken as 2.5 m (calculated from Heskestad's formula). Both the tunnel portals are open and assumed to be at atmospheric conditions. The computational domain is divided into 1,25,693 control volumes. The CFD model input conditions are given in Table 5.2.

5.2.1 Observations

The most significant effect of fire inside a tunnel is the buoyant effect caused by the difference of density between smoke and fresh air. This effect tends to create a layer of hot smoke and gases flowing away from the fire near the crown of the tunnel, while air supporting combustion moves towards the fire beneath the smoke layer. This can be observed from Figure 5.5 which shows predicted temperature distribution on the vertical central plane through the fire source and tunnel portals at various times. It can be seen from the Figure 5.5 that the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals move towards the fire source. This can also be seen from Figure 5.6, which shows the velocity distribution at the same plane.

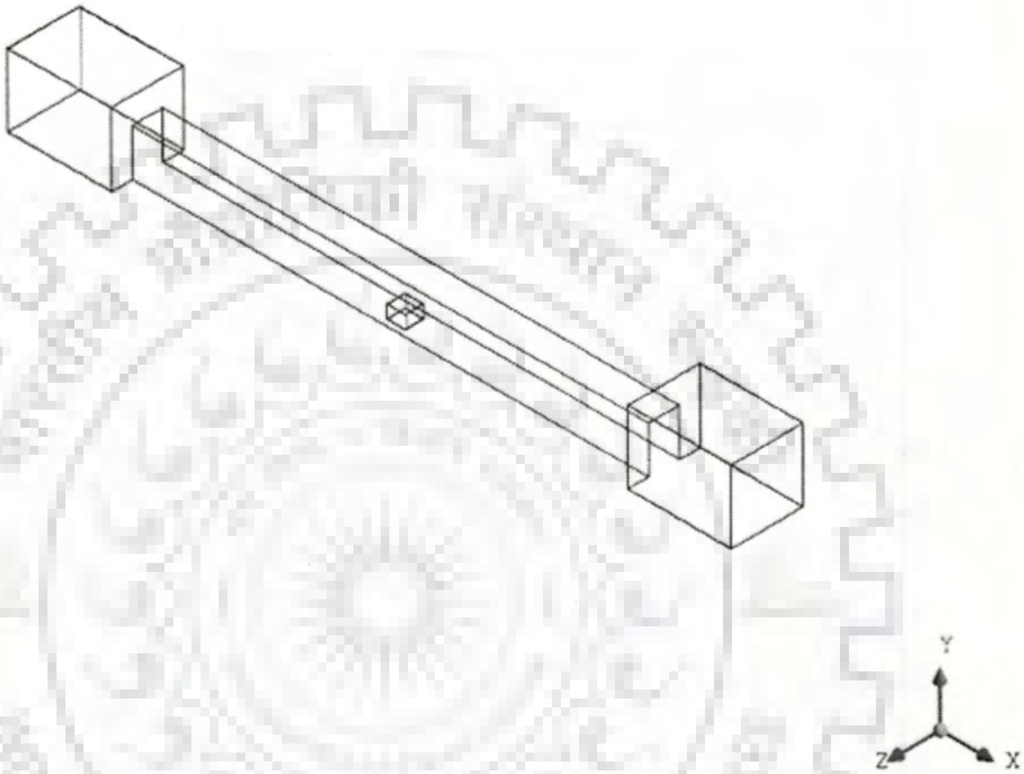
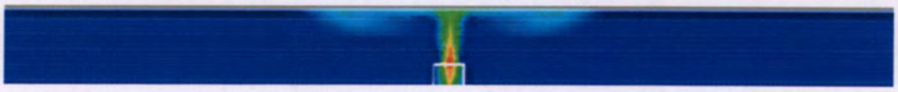


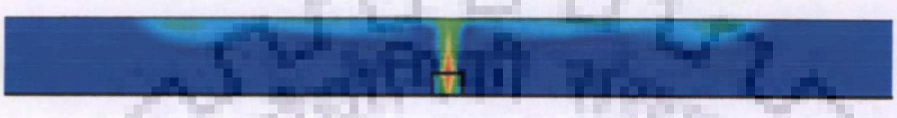
Figure 5.4 Computational Domain of Tunnel section with extended domain

Table 5.2 CFD Model Input Conditions

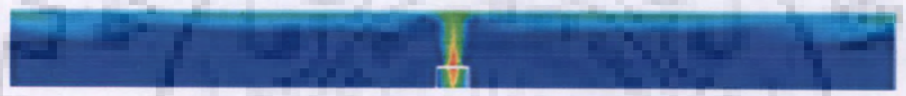
Domain	Tunnel and an extension of connected space outside the door. Extended domain - 20 m x 14 m x 12 m
Nodes Tetrahedron Elements	1,25,321 5,93,566
Turbulence	standard k-ε model with buoyancy modification
Radiation	P1 model / Six flux model
Heat Source	VHMS model
Ref. Temp.	300 K
B.C	Walls-smooth, adiabatic and gray, emmisivity of walls – 0.9, emmisivity of ceiling and floor – 0.5 extended domain boundary – Free Pressure Boundary
Wall function	Standard wall function
Wall influence on flow	No slip
Advection Scheme	High Resolution
Domain Material	Air
Density	1.185 [kg m^{-3}]
Viscosity	1.831E-05 [kg m^{-1} s^{-1}]
Specific heat	1.0044E+03 [J kg^{-1} K^{-1}]
Conductivity	2.61E-02 [W m^{-1} K^{-1}]



(a) Time = 10 s



(b) Time = 20 s



(c) Time = 30 s

Figure 5.5 Temperature Distribution along central plane in the tunnel at various times (a) 10 s, (b) 20 s and (c) 30 s

Figure 5.6 Velocity vector along central plane in the tunnel

5.2.2 Comparison with Empirical Formula

The temperature above the fire source is compared with the plume centerline temperature correlation given by Heskestad (1995). The correlation given by him is:

$$T_{cp} = T_0 + 9.1\left(\frac{T_0}{gC_p^2\rho^2}\right)^{1/3}\frac{Q_c^{2/3}}{(Y-Y_0)^{5/3}} \tag{5.2}$$

Where, z_0 is the height of virtual origin (equal to flame height) and is defined as

$$Y_0 = 0.235 * Q^{0.4} - 1.02 * D_f \tag{5.3}$$

A comparison of predicted temperature by CFX and Heskestad's correlation is given in Figure 5.7. It can be seen from the figure that temperatures are in close agreement away from the fire source, while at points near fire the temperature differs significantly. This can be because of assumed height of volumetric heat source. A lesser height will result in increased temperatures near the source. Also, in Heskestad's relationship, radiation loss is assumed to be 30% while in simulations radiation is being modeled. It also corroborates the fact, as done in Steckler experiment, that better modeling of fire source is necessary to predict the temperatures near the fire source.

5.3 SMOKE CONTROL IN TUNNELS - TUNNEL VENTILATION

As discussed in chapter 2 section 1 (2.1), the design principle of the ventilation system in emergency mode is its ability to prevent back-layering of smoke in the upstream area. The minimum air velocity required to prevent back layering and to force the hot air and smoke in the desired direction, called critical velocity is the design velocity and is to be ascertained.

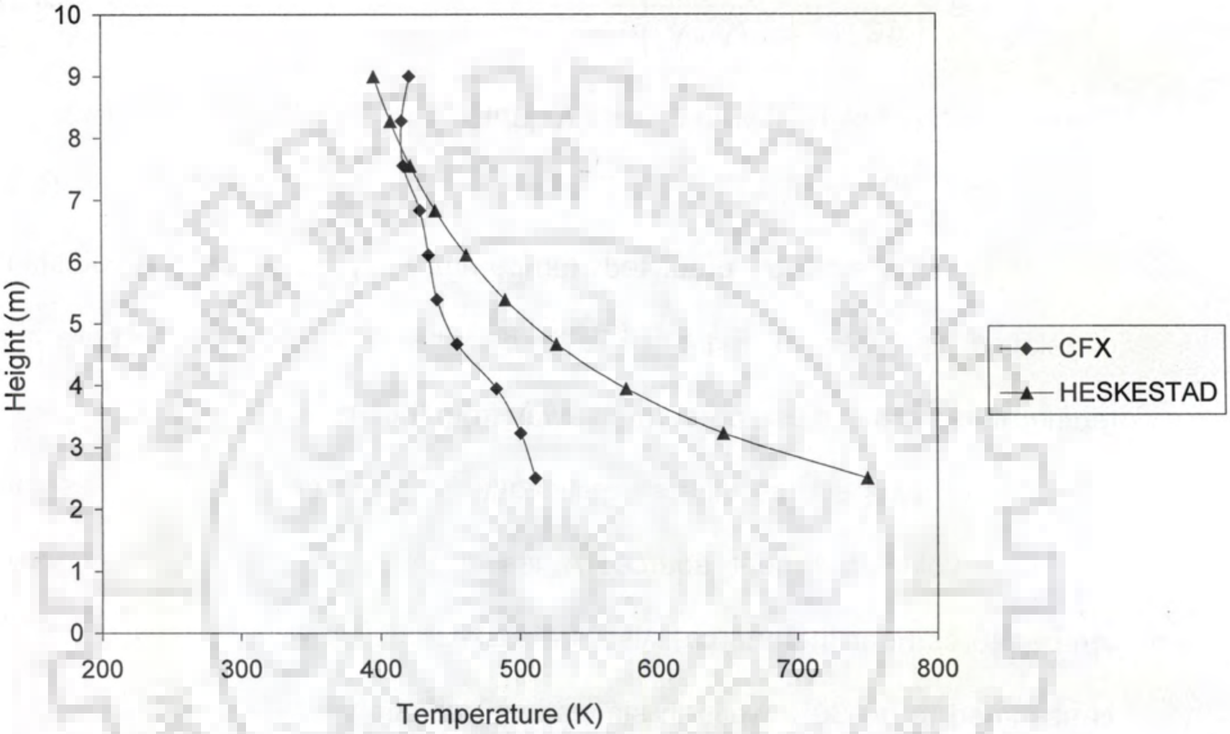


Figure 5.7 Comparison of predicted centerline temperature with Hekskestad's correlation

A review of the correlation of critical velocity is given chapter 2 section 3 (2.3.1). The critical velocity predicted by CFX is to be compared with the empirical formulae of Wu and Bakar (2000). For brevity, the formula proposed by Wu and Bakar is again mentioned below:

$$V_c^* = 0.40 * \left(\frac{Q^*}{0.20}\right)^{1/3} \quad \text{for } Q^* \le 0.20 \tag{5.4}$$

and

$$V_c^* = 0.4 \text{ for } Q^* \ge 0.20 \tag{5.5}$$

where the dimensionless heat release rate, Q*, and the dimensionless critical velocity, V_c^*, are given by

$$Q^* = \frac{Q_c}{\rho_0 C_p T_0 g^{1/2} H_D^{5/2}} \tag{5.6}$$

$$V_c^* = \frac{V_c}{\sqrt{gH_D}} \tag{5.7}$$

5.3.1 Determination of Critical Ventilation Velocity for Fire inside Tunnel

The effect of ventilation flow rate on thermal environment inside the tunnel is studied for two scenarios. In scenario 1, the induced airflow is assumed to be uniform; this corresponds to a case, where fans are located at sufficient distance from the fire source. In second scenario, the fans are assumed to be located near the fire source resulting in non-uniform induced airflow.

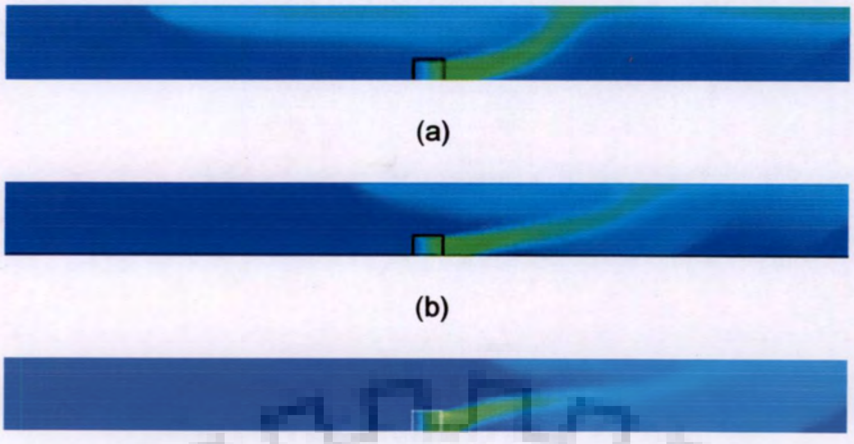
5.3.1.1 <u>Uniform Induced Air Flow</u>

Based on the empirical formulae by Wu and Bakar, the critical velocity is 1.52 ms^{-1}. CFD simulations were carried out for different volumetric flow rates to study the influence of the volumetric flow rate on the thermal distribution. Three different volumetric flow rates of 80, 106, 132 m^3s^{-1} corresponding to velocities of 1.5, 2, 2.5 ms^{-1} respectively are induced inside the tunnel. The above airflow rates are assumed to be uniform and fully developed inside tunnel.

5.3.1.1.1 Observation

Figure 5.8(a-c) shows temperature distribution along central plane for the case of different velocities. The results indicate that smoke back layering occurred for velocities of 1.5 $ms^{-1}$and ms^{-1} and that the smoke layer covers a longer area when the flow rate in the tunnel is low. At flow rate of 2.5 ms^{-1} all the smoke moves upstream indicating that this flow rate is more than the critical velocity. This implies that critical velocity lies between 2 $ms^{-1}$and 2.5 ms^{-1}. To determine the critical velocity, CFD simulations were carried out with flow rates of 2.1 ms^{-1}, 2.18 ms^{-1} and 2.25 ms^{-1}. Their temperature distributions are given in Figure 5.9(a-c). It can be seen from the figure that the critical velocity corresponds to velocity of 2.18 ms^{-1}. This velocity is about 43% higher than calculated from formulae.

The increased critical velocity may be because of the manner in which the fire source is defined. The fire source is defined as solid block of height 2.5 m, thereby reducing the tunnel cross section at fire source. This situation can be correlated with a stalled train in a tunnel, thereby reducing the tunnel cross section. Also in the formulae, heat release rate taken is convective which is assumed to be 70% of total heat release rate i.e. 30% heat is assumed to be lost through radiation whereas in CFD computations modeling of radiation are done.

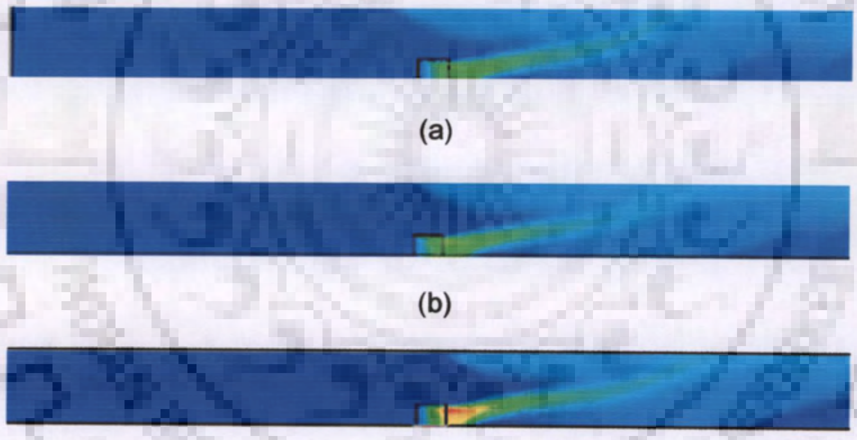


(a)

(b)

(c)

Fig 5.8 Temperature Distribution along central plane in the tunnel when velocity at inlet is (a) 1.5 ms^{-1} (b) 2 ms^{-1} (c) 2.5 ms^{-1}



(a)

(b)

(c)

Fig 5.9 Temperature Distribution along central plane in the tunnel when velocity at inlet is (a) 2.1 ms^{-1} (b) 2.18 ms^{-1} (c) 2.25 ms^{-1}

It is found that on upstream side temperatures remained around 300 K up to 3 m height from the bottom of tunnel and at a distance up to 45 m from the inlet to the tunnel.

5.3.1.2 Non Uniform Induced Air Flow

Two cases of non uniform induced air flow are studied. The first case corresponds to a situation where only supply fan is activated. In the second case, the effect of operating exhaust fans in addition to supply fans is also studied.

(i) Only Supply Fan Activated

In this set of CFD simulations, the airflow forced inside tunnel is not assumed to be uniform. This corresponds to a case where jet fans are located near fire source. The cross section area of the jet fan is taken to be 3 m x 2 m. To maintain airflow of 2 ms^{-1} in the tunnel, the velocity of jet fan using equation of continuity should be 18 ms^{-1}. However, this velocity comes out to be very high and forces smoke downstream rapidly. This becomes a case of strong ventilation. Different simulations are done in which the flow velocity of fan is taken as 6 ms^{-1}, 8 ms^{-1}, 10 ms^{-1}. The temperature distribution along central plane for these cases is shown in Fig 5.10(a-c). It can be seen from Figure 10(b) that smoke back layering exists for velocity of 8 ms^{-1}. When velocity lies between 8 ms^{-1} and 10 ms^{-1} there is no back layering.

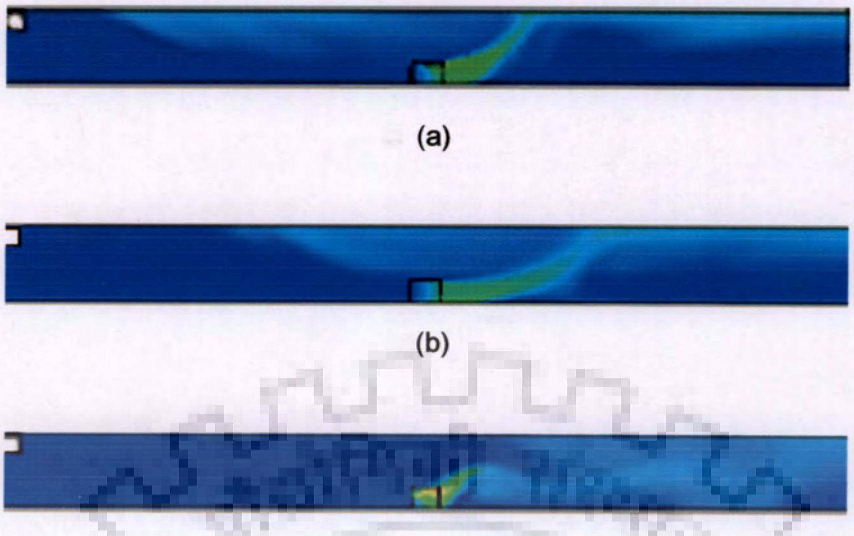
(ii) Both Supply and Exhaust Fans Activated

In this set of simulations, both inlet and exhaust fans are activated. The simulations are carried out for flow velocities of 10 ms^{-1}, 12 ms^{-1}, 14 ms^{-1} both at inlet and exhaust. Their temperature distributions at the center plane are shown in Fig 5.11(a-c). It can be seen from Figure 5.11 that the stratified layer of smoke in downstream region is not formed. For inlet and exhaust velocity of 10 ms^{-1} the air flow tilts the plume downstream and its trajectory comes to floor in the downstream area from where the exhaust fan sucks this air to exhaust. The smoke thus remains at the bottom of tunnel. As the flow velocity is increased, inlet air does not touch floor downstream, rather it gets exhausted to atmosphere, while smoke layer remains at bottom.

It is found that temperatures remained below 320 K up to a height of 3 m from the bottom of tunnel and the velocities were below 5 m/s. However, higher velocities (6.3 m/s) were predicted close to fire source at certain times.

5.4 CONCLUDING REMARKS

The numerical model to be used for predicting environment inside tunnel is validated using results of one of Steckler experiments. It is found that physical models used in CFX predicted well the results of Steckler experiments at distances away from the fire source. Then using same model parameters thermal environment for a 4 MW fire for a section of tunnel is predicted. It is observed that the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals move towards the fire source. The centre line plume temperatures are compared with empirical relation. The critical velocity for this tunnel configuration assuming uniform induced air flow is then determined.

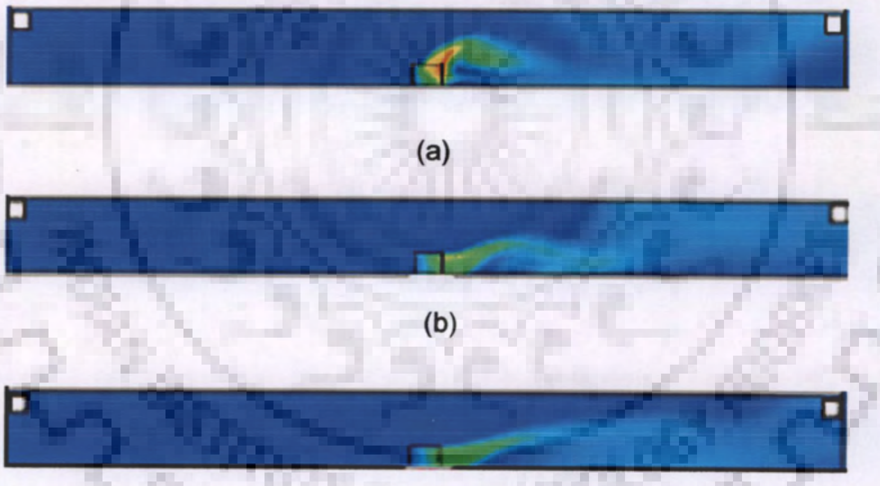


(a)

(b)

(c)

Fig 5.10 Temperature Distribution along central plane in the tunnel when velocity of inlet fan is (a) 6 ms^{-1} (b) 8 ms^{-1} (c) 10 ms^{-1}



(a)

(b)

(c)

Fig 5.11 Temperature Distribution along central plane in the tunnel when velocity of both inlet and exhaust fan is (a) 10 ms^{-1} (b) 12 ms^{-1} (c) 14 ms^{-1}

It is found that CFX predicted higher critical velocity than the empirical relations. This may be because of reduced cross-section of tunnel at fire site due to height of fire source or because thermal heat transfer through radiation is modeled. In absence of any data on heat loss due to radiation, the empirical formula assumes that heat lost due to radiation is 30% of total heat release rate. It is also found that on upstream side temperatures remained around 300 K up to 3 m height from the bottom of tunnel and at a distance up to 45 m from the inlet to the tunnel.

In case of non uniform induced air flow, it is found that still higher critical axial fan velocities are required. On upstream side the temperatures remained below 320 K up to a height of 3 m from the bottom of tunnel and the velocities were below 5 m/s. However, higher velocities (6.3 m/s) were predicted close to fire source at certain times. It is observed that in region downstream of fire, there is no well-stratified smoke layer. A well-stratified smoke layer in downstream region is not observed even when exhaust fans are activated. The air flow tilts the plume downstream and its trajectory comes to floor in the downstream area from where the exhaust fan sucks this air to exhaust. At larger flow rates the induced air moves along the ceiling and smoke layer remains at bottom. It implies that the only through upstream areas escape can take place.

CHAPTER - VI

NUMERICAL STUDIES ON EVALUATION OF VENTILATION STRATEGIES OF UNDERGROUND METRO RAIL TRANSPORT SYSTEM WITH JET INJECTION SYSTEM (INCLINED FANS)

6.0 INTRODUCTION

In the last chapter, the effectiveness of jet fans in controlling smoke in a section of underground tunnel was investigated. The fire source produced constant HRR. This chapter studies the influence of HRR curve slope on the smoke flow dynamics in a realistic tunnel model fitted with jet injection system. The tunnel section considered is similar to that of Delhi Metro where the supply and exhaust fans provide / extract air through ventilation shafts located in the ceiling. These shafts are inclined at an angle of 10 degrees from the plane of the ceiling. This study simulates the smoke movement resulting from the prescribed heat release rate for the above tunnel configuration and correspondingly predicts the temperature field. The velocity of supply and exhaust fans necessary to produce desired critical velocity for this type of tunnel configuration is ascertained.

The present study differs from the previous study mainly in three different ways. Firstly, the tunnel considered in this study is about four times long and its volume is about 144% more than that of tunnel earlier studied. Secondly, in this study the fire source produces a variable heat release rate (HRR) starting from zero to a maximum of 16 MW while in the earlier study the fire source considered produces a constant HRR of 4 MW. Thirdly the ventilation fans in this study are located in a shaft in the ceiling and this shaft is inclined at 10^{o} to the plane of ceiling while in the earlier study the fans are located axially below the ceiling. Thus this study involves a more complex tunnel / fan / fire source configuration.

The fire dynamics inside tunnel for the case of natural ventilation is studied for 17 minutes using three different mesh sizes. In case of fire inside tunnel, two scenarios of ventilation are studied: (i) fans activated immediately and achieve its full speed as soon as the fire is detected (ii) fans activated at delayed times to take into account the response time for the fans to achieve its maximum speed. The velocity of supply and exhaust fans necessary to remove smoke in 30 s from the upstream direction is determined.

6.1 NUMERICAL EXPERIMENTS

6.1.1 Tunnel Specifications

The tunnel considered is similar to that of a Delhi Metro Rail System. Three dimension CAD drawings of station and part of tunnel section are prepared from available two dimension CAD drawings. The tunnel has two track ways. The full and sectional views are shown in Figure 6.1(a, b). Only one track way is analyzed. The section considered is 400 m long, 5.5 m wide and 6 m high. A view of the computational domain of modeled section is shown in Figure 6.2. The supply and exhaust fans are located in ceiling at a distance of 5 m from both ends of portal. Their dimensions are 5 m x 4m. The fans can operate in reverse mode also. The ventilation ducts are inclined at an angle of 10 degrees from the plane of ceiling. However in the figure the duct is assumed to be lying on the plane of the ceiling while supply and exhaust velocity will be inclined at an angle of 10 degrees. The fire source is assumed to be located in the middle of the tunnel. The numerical model and physical

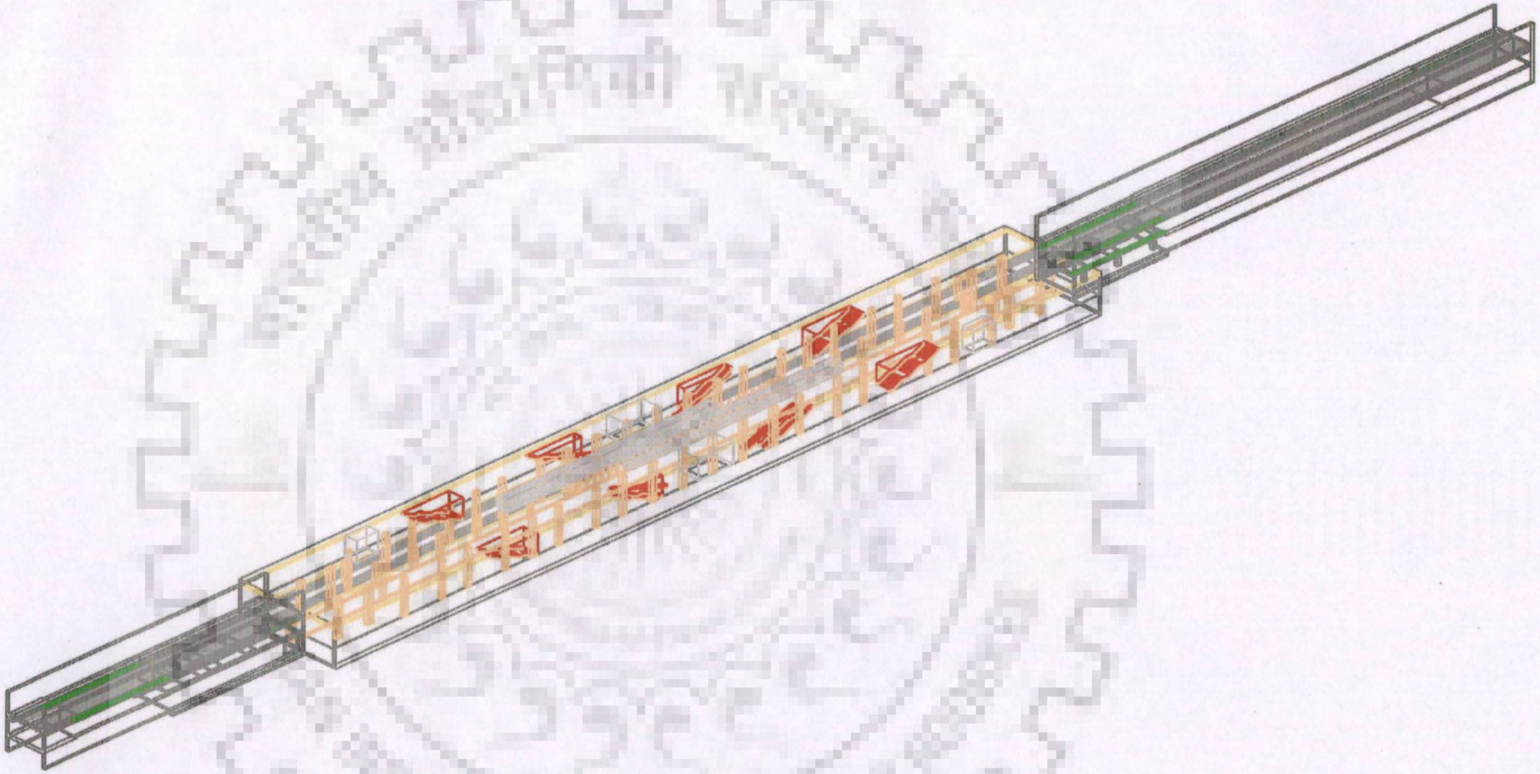


Figure 6.1(a) Full view of station and part of trackways

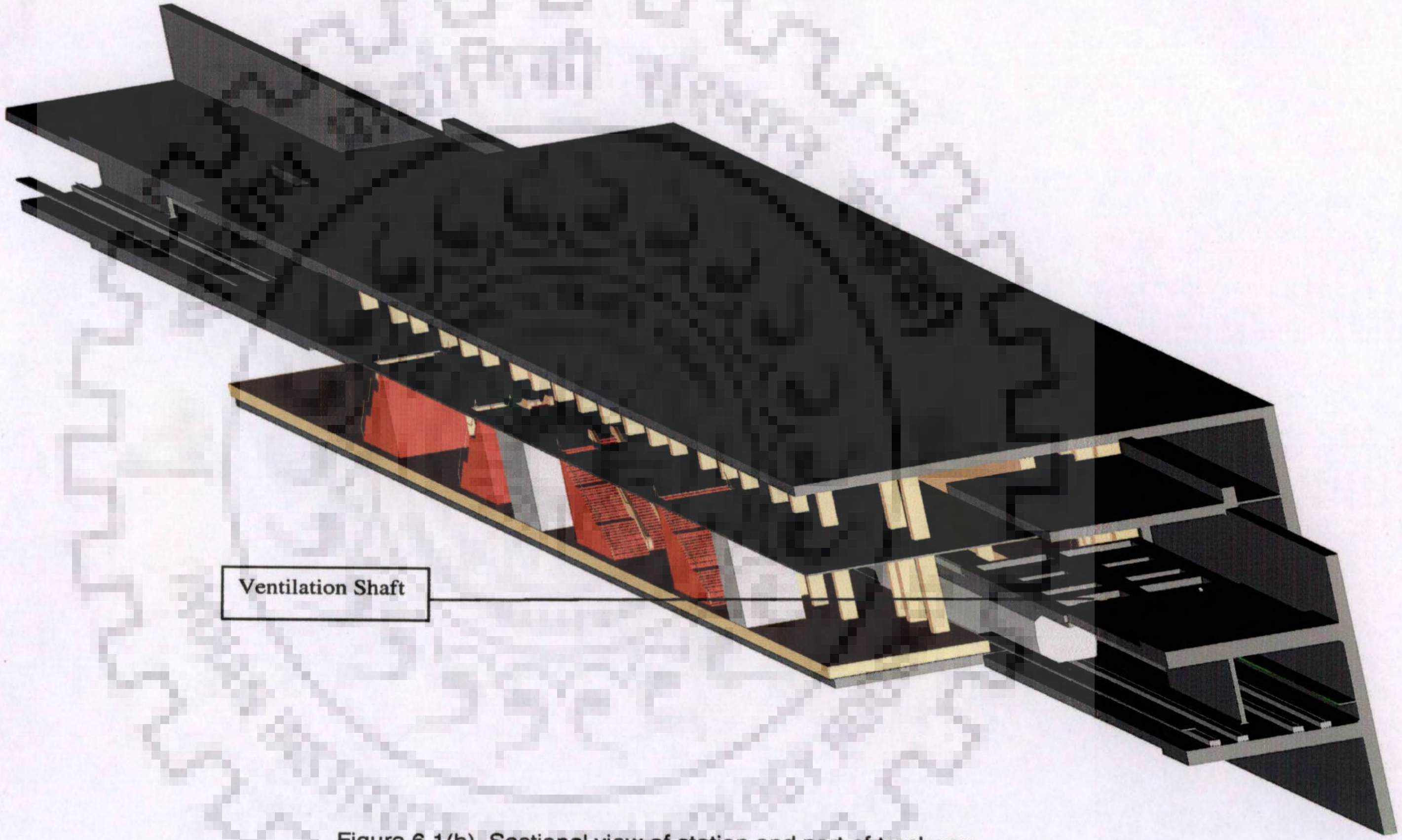


Figure 6.1(b) Sectional view of station and part of trackway

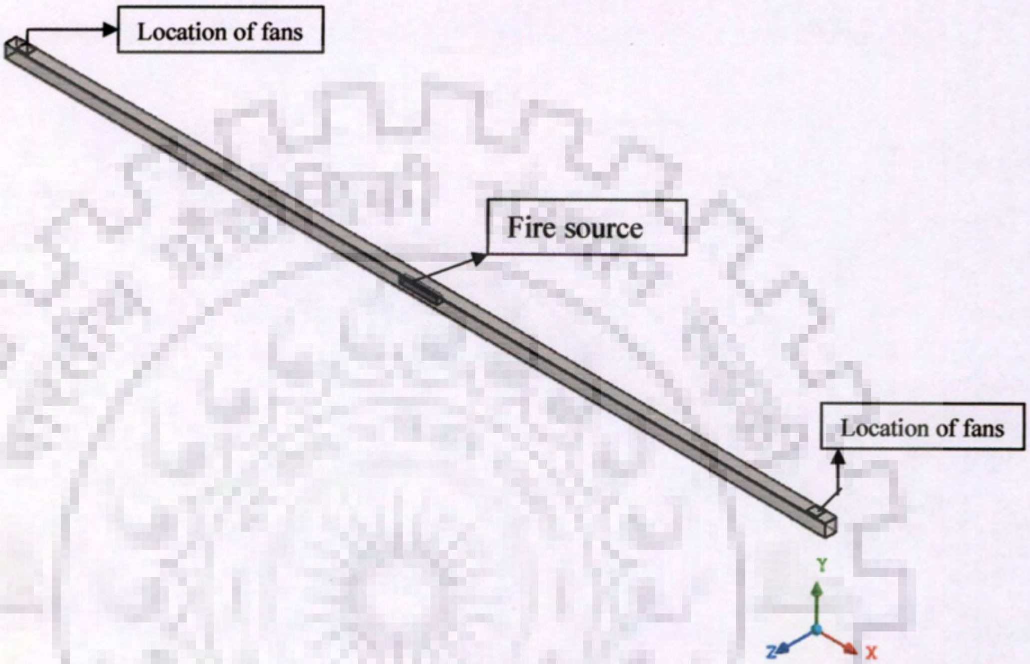


Figure 6.2. Tunnel section

models employed are the same as used in previous chapter. However, a brief description of the various sub models used is given below:

6.1.2 Physical Models and Boundary Conditions

(i) Fire Source Model

The volumetric heat source model (VHS) is used to define the distributed heat release from a fire. The heat release rate per unit volume from the VHS source represents the heat generated from the fire. The total heat is assumed to be released uniformly throughout the whole source. For all simulations, a fire with a variable heat release rate is used to represent a carriage fire. The fire source is taken as a volumetric heat source with dimensions – 18 m x 3 m x 2.8 m. The base area of fire source corresponds to the area of subway car as mentioned in Ingasson (2006). The heat source is located 0.5 m above floor and starts from left portal at x = 191 m. The maximum heat release rate (HRR) taken is 16 MW. The variation of heat release rate of fire source (HRR) with time is shown in Figure 6.3. It can be seen from Figure 6.3, that there are three stages of fire development, namely the growth phase followed by a constant phase and then by a decay phase. The coefficients of growth and decay phase, assuming quadratic growth are taken from the work of Ingason (2006). The variation of HRR with time is represented by the following formula

$$Q = \alpha t^2 \qquad 0 < t \le t_{max} \tag{6.1}$$

$$Q = \alpha t_{max}^2 = Q_{max} \qquad t_{max} < t < t_D \tag{6.2}$$

$$Q = Q_{max}\, e^{-\beta(t - t_D)} \qquad t_D < t \tag{6.3}$$

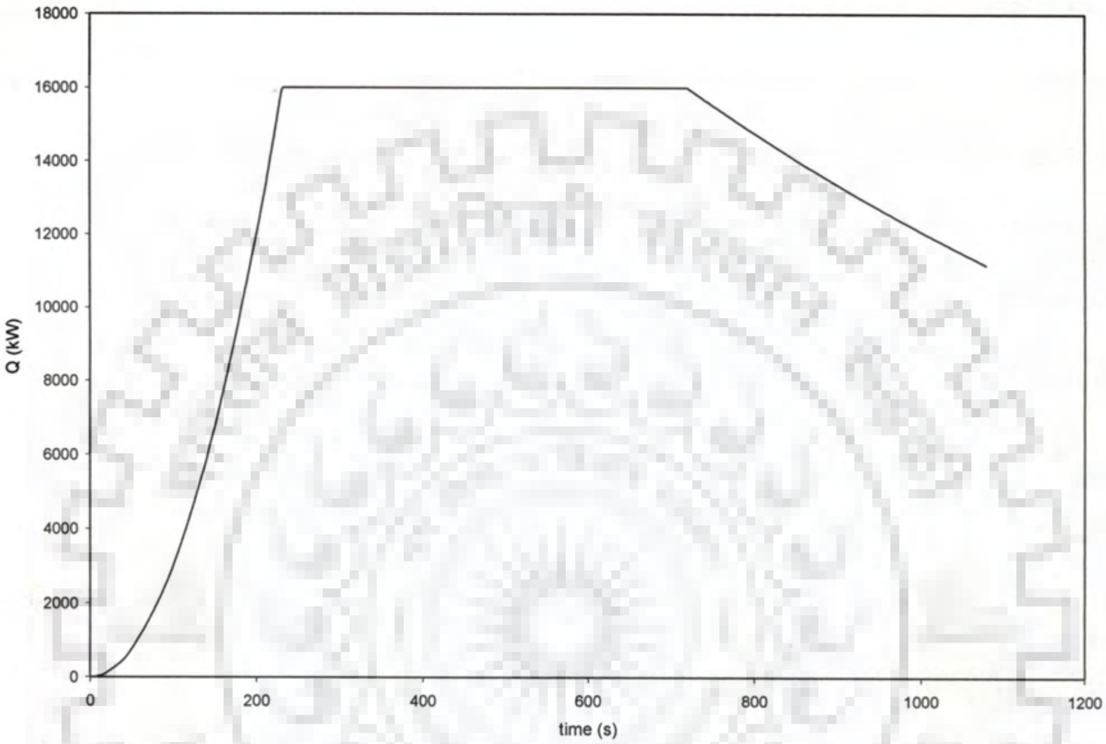


Figure 6.3. Heat release rate vs Time

where

α=0.3 kWs^{-2}, β=0.001 s^{-1}, t_{max} = 3.85 min = 231 s, t_D = 12 min = 720 s, Q_{max} = 16000 kW

(ii) Height of Volumetric Heat Source

The base of fire source taken is 18 m x 3 m. For the HRR of 16 MW (max), the height of volumetric heat source is taken to be equal to height of flame from Heskestad correlation and is calculated as follows.

Empirically, the height of flame given by Heskestad (SFPE Handbook, 1995) is defined as

$$Y_f = 0.235 * Q^{0.4} - 1.02 * D_f \tag{6.4}$$

For a base area of 18 m x 3 m the equivalent diameter is D_f = 8.29 m

Therefore from equation 6.4, the flame height is 2.8 m.

Thus fire source is taken as a volumetric heat source with height of 2.8 m.

(iii) Turbulence Model

The buoyancy modified standard two-equation k-ε turbulence model is used.

(iv) Wall Function

Standard wall functions have been used to estimate the turbulence kinetic energy and dissipation at the cell near the wall.

(v) Radiation Model

The thermal radiation is modeled using the six-flux radiation model or P1 model.

(vi) Boundary Conditions

Both the tunnel portals are open to stations and assumed to be at atmospheric conditions. All walls are assumed to be smooth, adiabatic and gray. At solid boundaries no slip condition is assumed. The supply duct is treated as inlet and exhaust duct as outlet. In case when fans are not working, the boundaries of these ducts are treated as walls. The tunnel portals are modeled as open boundary conditions. The set of parameters used in the present study are C_p = 1.004 $kJKg^{-1}K^{-1}$ ambient temperature T_o = 310 K, ambient air density ρ_o = 1.225 kgm^{-3}, acceleration due to gravity g = 9.81 m s^{-2}.

6.2 NUMERICAL SIMULATION

In order to simulate the system, two cases are considered and analyzed.

Case 1: Fire inside tunnel, Ventilation Fan non-operational

Case 2: Fire inside tunnel, Ventilation Fans operational

For case 2, two scenarios are further studied. Fans activated immediately after detection of fire and fans activated at delayed time to account for response time of activation of fans.

The effect of ventilation considering constant peak HHR of 16 MW is then studied to design the maximum capacity of fans required. All the simulations are carried out on Compaq PIV 3.0GHz, 2GB RAM machine.

6.2.1 Fire inside Tunnel, No Ventilation Fan

The simulations, in order to study environment inside tunnel, are carried out using a coarse mesh of 44618 nodes for approximately 17 minutes using a time step of 0.1 s. This coarse grid is chosen so as to check whether the physics is properly defined and results are appropriate. The temperature and vector profiles are recorded at intervals of 15 s.

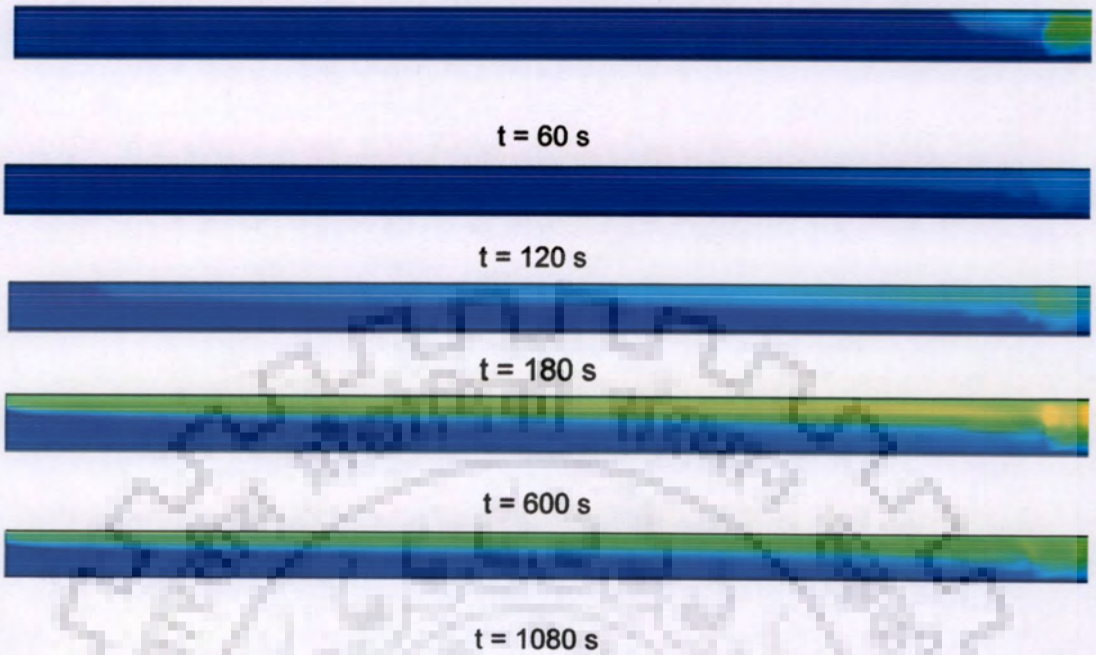
As observed in last chapter, the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals moves towards the fire source. This can be observed from Figure 6.4(a) which shows predicted temperature distribution on the vertical central plane through the fire source and tunnel portals at various times and Figure 6.4(b) which shows velocity distribution at t= 90 s. It can also be seen from Figure 6.4(a) that the plume reaches close to tunnel portals in about 180 s.

The above simulation is then carried out with fine mesh having 66850 nodes, and 101916 nodes. This helps in studying the sensitivity of mesh on simulation results. Table 6.1 shows back layering distance and temperature of plume for different mesh sizes at t= 180 s.

Table 6.1 Back layering distance and temperature of plume for different mesh sizes at t = 120 s

Mesh Size (No of Nodes)	Back layering distance (m) from Left portal	Temperature (K) of plume at back layering distance
44618	108	318
66850	110.9	324.3
101916	109	322

Figure 6.5 (a-d) shows the temperatures predicted by three different mesh sizes at time = 120 s along the length of tunnel at a height (y) of 5 m, 4 m, 3 m, 2 m and z = 2.75 m. In the figures T1, T2, T3 denote the temperature predicted by mesh having 44168, 66850, 101916 nodes respectively. Figure 6.6 (a-d) shows the temperatures predicted by three different mesh sizes at time = 120 s in a vertical plane at various horizontal distance (x) of 150 m, 170 m, 210 m, 225 m and z = 2.75 m.



t = 60 s

t = 120 s

t = 180 s

t = 600 s

t = 1080 s

Figure 6.4(a) Temperature distribution on the vertical central plane through the fire source and tunnel portals at different times (only one half of tunnel portion is shown in figure)

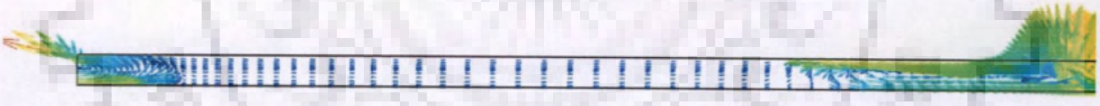


Figure 6.4(b) Velocity distribution on the vertical central plane through the fire source and tunnel portals at time = 90 s (only one half of tunnel portion is shown in figure)

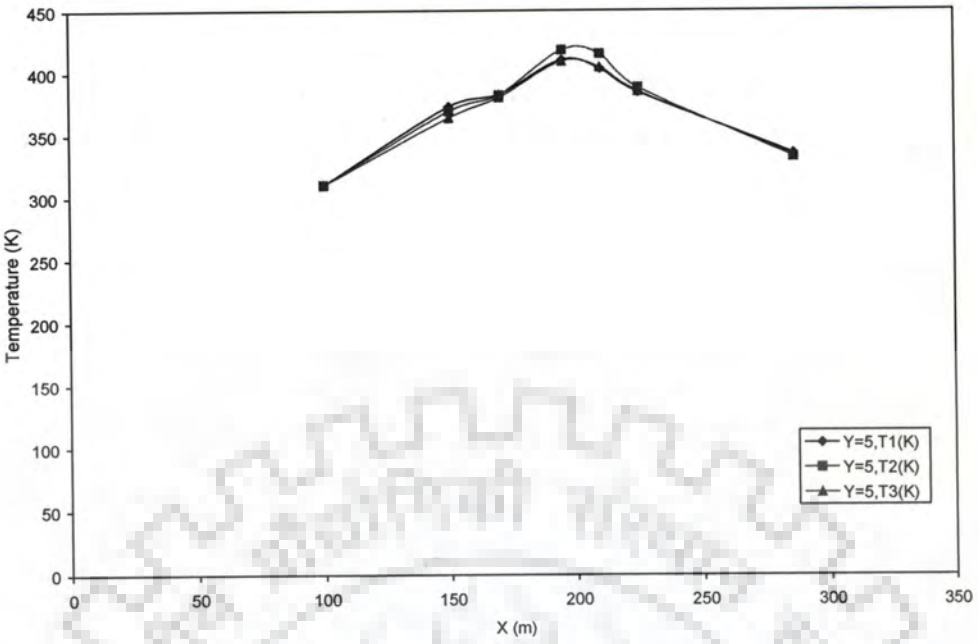


Figure 6.5(a) Comparison of temperature predicted from three different meshes with distance along the length of the tunnel at Y = 5 m, Z = 2.75 m, t = 120 s

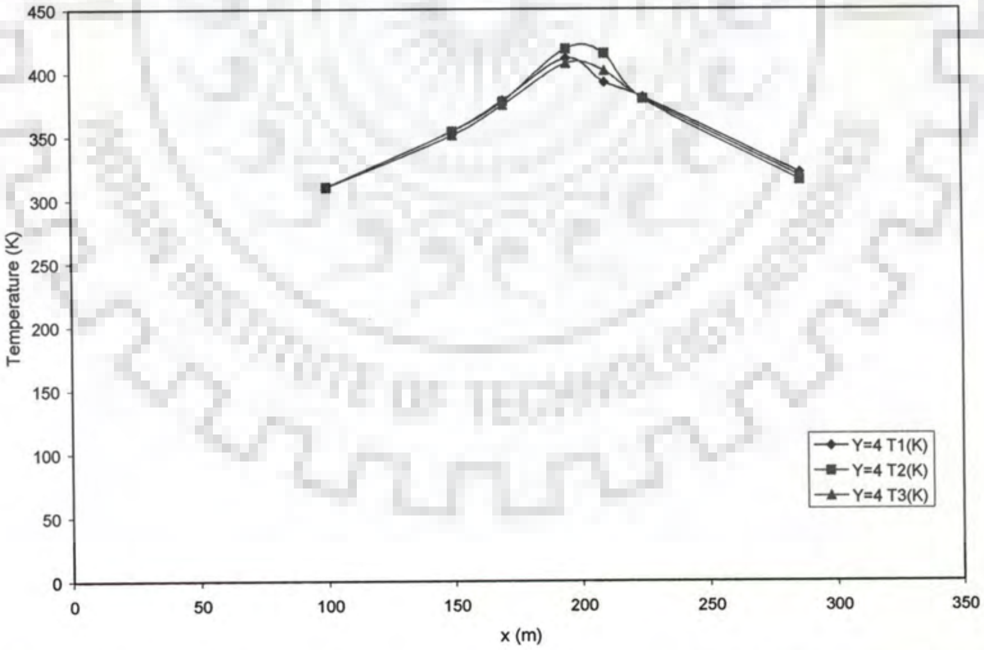


Figure 6.5(b) Comparison of temperature predicted from three different meshes with distance along the length of the tunnel at Y = 4 m, Z = 2.75 m, t = 120 s

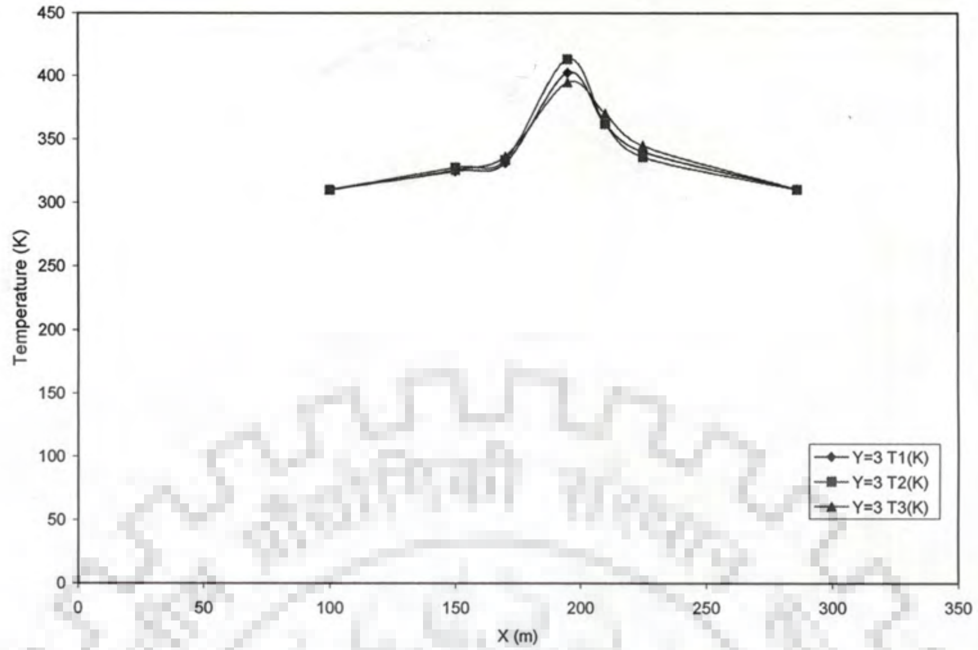


Figure 6.5(c) Comparison of temperature predicted from three different meshes with distance along the length of the tunnel at Y = 3 m, Z = 2.75 m, t = 120 s

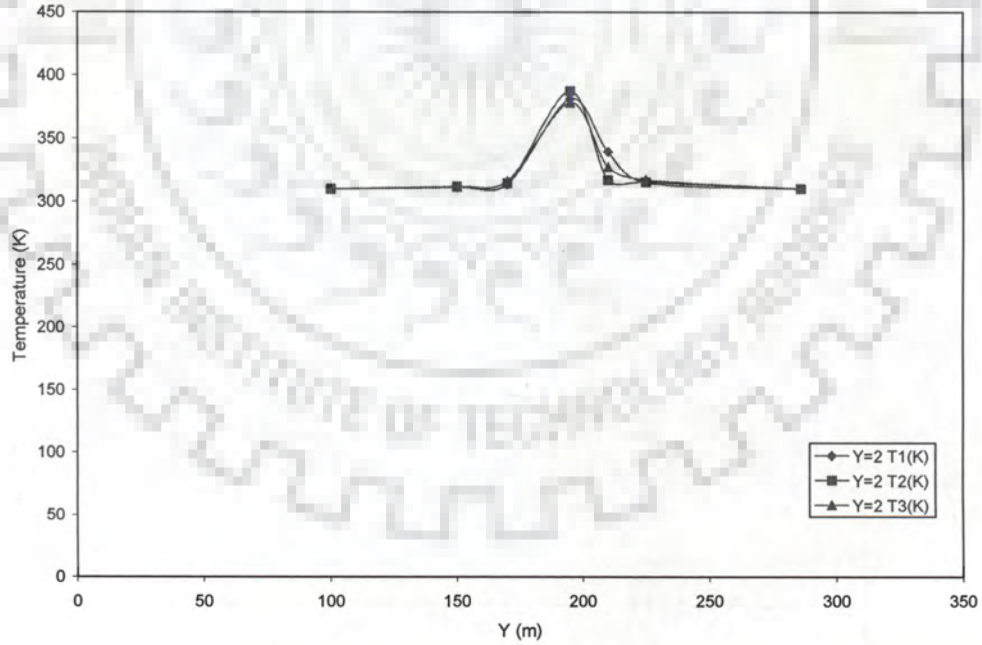


Figure 6.5(d) Comparison of temperature predicted from three different meshes with distance along the length of the tunnel at Y = 2 m, Z = 2.75 m, t = 120 s

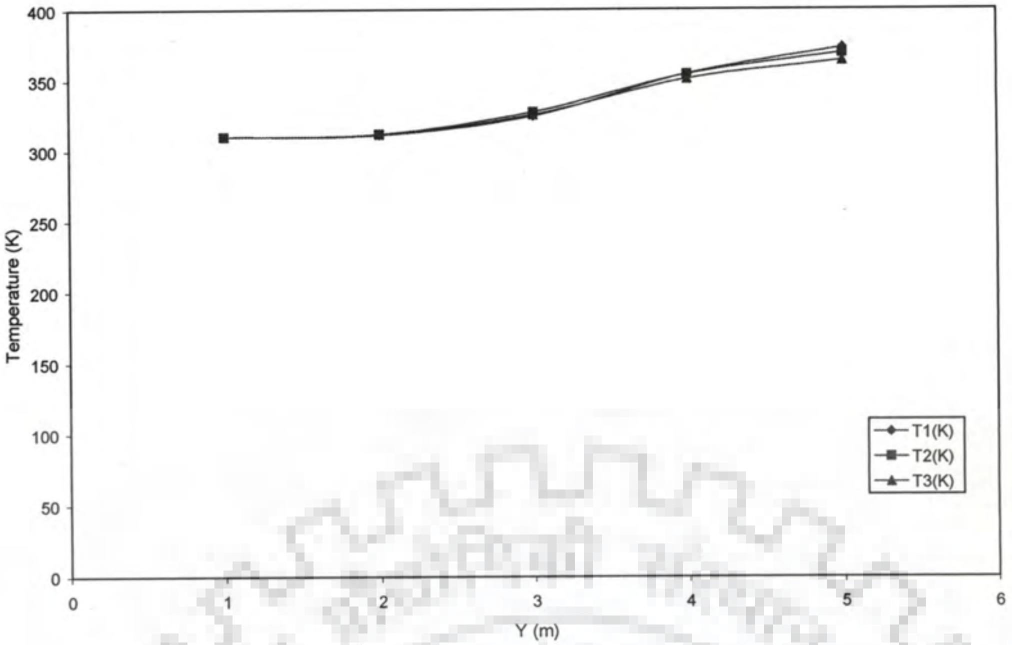


Figure 6.6(a) Comparison of temperature predicted by three different meshes along vertical plane at X = 150 m, Z = 2.75 m, t = 120 s

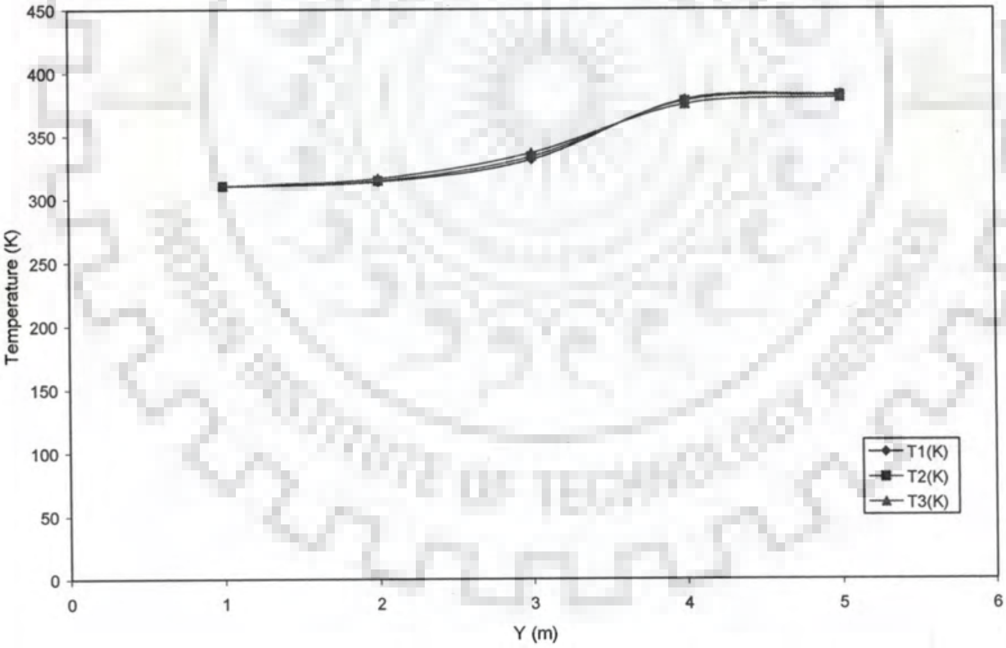


Figure 6.6(b) Comparison of temperature predicted by three different meshes along vertical plane at X = 170 m, Z = 2.75 m, t = 120 s

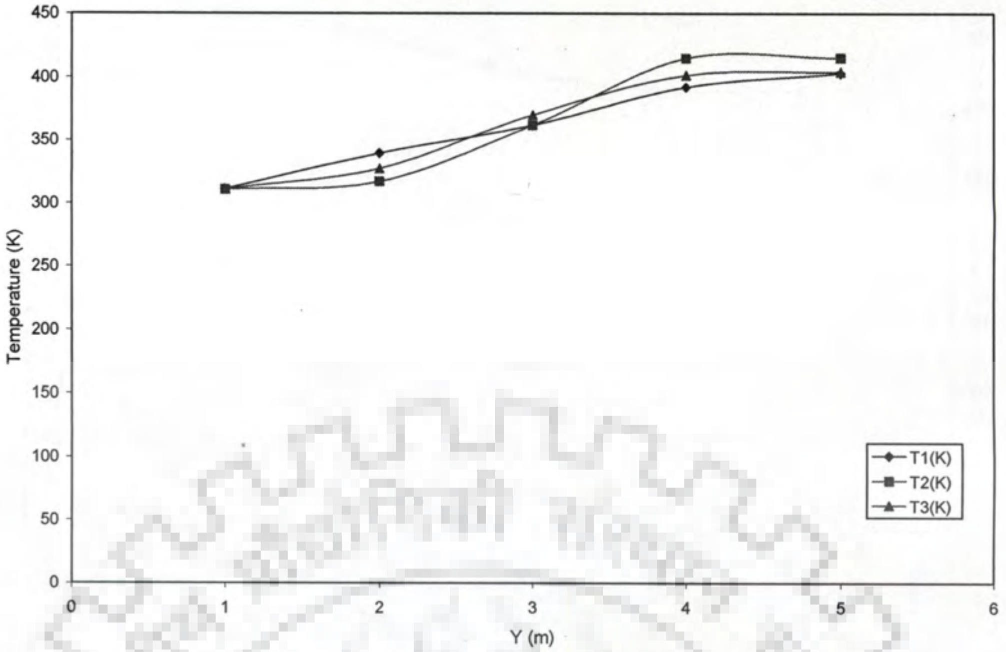


Figure 6.6(c) Comparison of temperature predicted by three different meshes along vertical plane at X = 210 m, Z = 2.75 m, t = 120 s

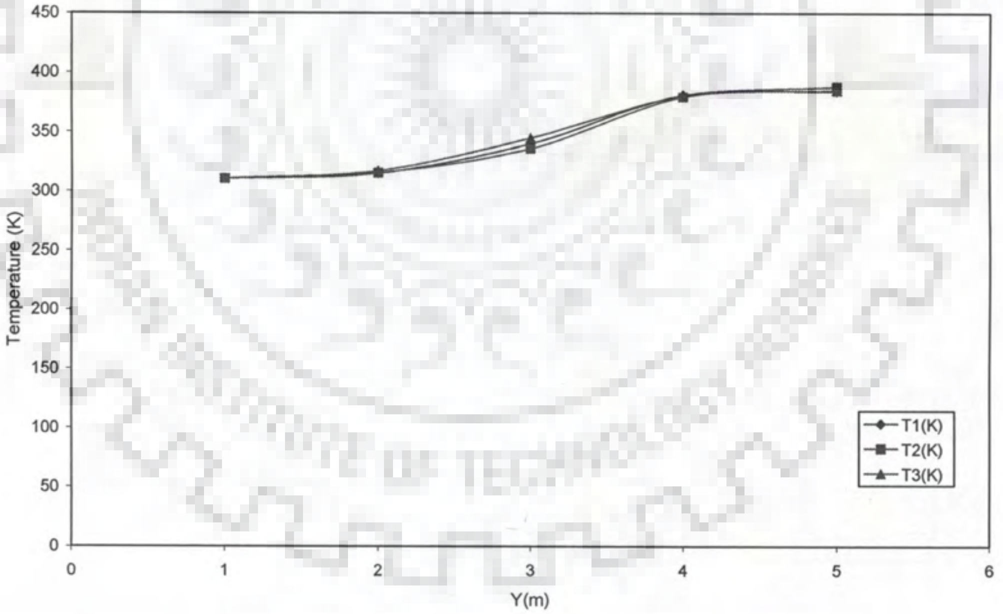


Figure 6.6(d) Comparison of temperature predicted by three different meshes along vertical plane at X = 225 m, Z = 2.75 m, t = 120 s

It can be seen from these figures that temperatures predicted by three different meshes are approximately same at distances away from the fire source. At distance of 191 m and 210 m from the left portal, which signifies the length of fire source, the difference of temperatures predicted is approximately 10 K. Since in this study the emphasis is more on smoke control in tunnels, hence temperatures away from the fire source are more important. For smoke control studies the fine mesh of 101916 nodes is used. Though for such size of tunnel, even this number of nodes will represent a coarse mesh, however because of hardware limitations the computations are done with this size of mesh. Table 6.2 shows back layering distance at t= 60, 120, 150, 180 s for simulations done with this mesh.

Table 6.2 Back layering distance at various times

Time (s)	60	120	150	180
Back layering distance (m) From Left portal	174.4	109	70.7	21

It can be seen from above table that the plume reaches the tunnel portal in about 180 - 200 seconds when location of fire is at the centre of the tunnel. This implies that plume reaches portal before fire source produces maximum HRR.

6.2.2 Fire inside Tunnel, Ventilation Fans Operational at 60s.

From Table 6.2, it can be inferred that the tunnel ventilation should start well in advance of 180 s to be effective. It is also observed from Figure 6.4(a) that at 60 s the position of plume is slightly away from the fire source in both directions. It can thus be safely assumed that the detectors will detect the fire somewhere between 30 to 60 seconds and fan will be started. Therefore, as a first case, the longitudinal ventilation system is assumed to be fully activated at

60 s. The maximum time as per NFPA (NFPA 502, 2001) code for fan to achieve its maximum speed is 60 s, however it is assumed in this study that fans are activated to their maximum speed instantaneously, after detection of smoke and speed variation of fan with time is not modeled. In the later part of study, fans fully activated at 120 s, 150 s and 180 s will also be studied to include response time of fans.

Based on the formulae by Wu and Bakar (2000), the critical velocity required is approximately 2.51 ms^{-1}. This velocity is the required uniform velocity in the axial direction. It is also found from simulations done in last chapter that higher critical velocity is required when predicted from CFD using these models.

First, CFD simulations are carried out using inclined velocity of 3 ms^{-1} (vertical component of velocity - 2.95 ms^{-1}), both at supply side and exhaust side. It is found that this velocity is insufficient to prevent back layering. Since the jet fans are located in the ceiling hence by forcing the air in the tunnel with this rate results in very low axial velocity inside the tunnel. To find appropriate critical velocity for this type of system, CFD simulations are carried out using different volumetric flow rates. As an initial guess, flow rate corresponding to supply and exhaust velocity of 17 ms^{-1} in direction perpendicular to ducts is taken. This flow rate produces high axial velocity, approximately 6.22 ms^{-1} which results in all smoke pushed in downstream direction in just 15 s. This velocity is also high enough for passengers to safely evacuate from the upstream direction. Further simulations are carried out with flow rates corresponding to vertical velocities of 14 ms^{-1} and10 ms^{-1}. It is found that with 10 ms^{-1} velocity there is no back layering at t = 90 s. A simulation using vertical velocity of 8 ms^{-1} shows slight back layering at t = 90 s. Another simulation is carried out using velocity of 9 ms^{-1}. It is found that with this velocity there is no back layering at t = 90 s and average velocity is found to be nearly 2.53 ms^{-1}. This velocity thus corresponds to critical velocity. Figure 6.7 shows thermal environment inside tunnel for different flow rates of supply and exhaust fans. Table 6.3 shows the average velocity inside tunnel corresponding to various flow rates.

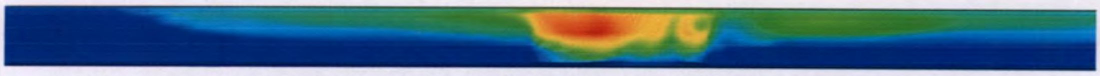
Table 6.3. Average velocity inside tunnel for different ventilation velocities

Supply and exhaust velocity (vertical velocity component) V_{sv} (ms^{-1})	2.95	8	9	10	14	17
Average velocity V_{avg} (ms^{-1})	0.18	2.07	2.53	3.01	5.1	6.53

The variation of Average velocity in axial direction inside tunnel upstream of fire for different ventilation velocities at t= 90 s is shown in Figure 6.8. This figure can be used to determine the vertical velocity required in supply and exhaust ducts when desired critical velocity in axial direction is known.

6.2.2.1 Effect of Exhaust Fans on Axial Flow Rate

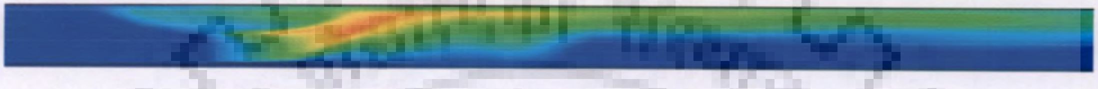
Two sets of simulations are done to see whether the exhaust fans have any significant effect on the axial flow rate upstream of fire. To study this, the exhaust fans are turned off while supply fans forces air at velocities of 17 ms^{-1} and 10 ms^{-1}. Table 6.4 shows the average velocities predicted in upstream area for the two cases when exhaust fan is on and off.



(a) **Supply and Exhaust velocity (vertical component) 2.95 ms^{-1}, t = 90 s**

(b) Supply and Exhaust velocity (vertical component) 17 ms^{-1}, t = 75 s

(c) Supply and Exhaust velocity (vertical component) 10 ms^{-1}, t = 90 s



(d) Supply and Exhaust velocity (vertical component) 8 ms^{-1}, t = 90 s

(e) Supply and Exhaust velocity (vertical component) 9 ms^{-1}, t = 90 s

Figure 6.7 Thermal environments inside tunnel for different flow rates (vertical component) of supply and exhaust fans

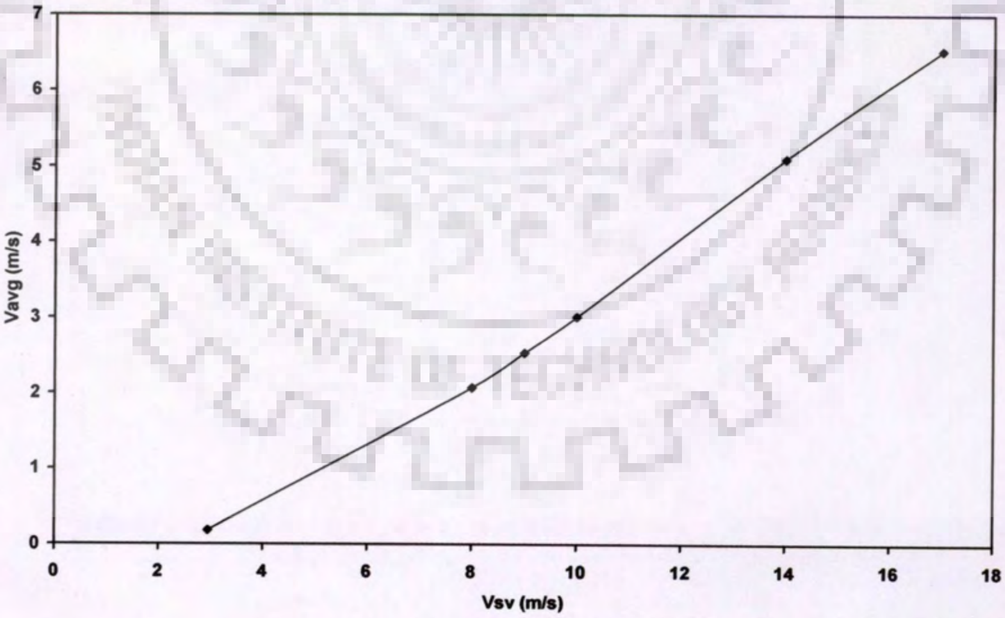


Figure 6.8 Average velocity inside tunnel vs supply and exhaust fan velocity

Table 6.4 Average velocity in upstream area when exhaust fan is on and off

Ventilation velocity (vertical velocity component) (ms^{-1})	10	17
Average velocity (ms^{-1}) when exhaust fan is on	3.01	6.53
Average velocity (ms^{-1}) when exhaust fan is off	2.95	6.22

It is seen that turning exhaust fan off does not significantly reduce the average velocity upstream of fire. This can be because the fire engulfs the whole width and height of tunnel. This does not imply that the exhaust fans should be turned off. The exhaust fans have influence in the downstream direction and helps in removal of smoke.

6.2.3 Fire inside Tunnel, Ventilation Fans Operational using above Determined Velocity at Later Times.

In this set of simulation, the fans are assumed to start at 120 s (assuming that response time for emergency operations is more) using the above determined critical velocity of 9 ms^{-1}. It is found that smoke still moves in downstream direction. However the time taken to prevent back layering is more. It occurs at t = 210 s. This can be seen from Figure 6.9 which shows the temperature distribution at t = 210 s. This implies that either time for smoke removal is increased or the ventilation velocity is increased to remove smoke in the same time. Further simulations are carried out with fans starting at 150 seconds and 180 seconds with fan velocity of 9 ms^{-1}. It is found that back layering is zero when t = 330 s and 435 s for the above two cases respectively. This implies that time taken to achieve zero back layering increases when the fans are activated at delayed times.

This is obvious because as time increases the plume moves farther in both directions and to remove smoke more time is required. Table 6.2 shows that at t = 180 s the plume is very close to tunnel portal (at a distance of 21 m from portal). If fans are activated at times later than these (say at t = 200 s) the plume would have reached the portal and supply fan will not be able to push smoke effectively in downstream direction. Thus for the ventilation system to be effective it is necessary that the fans are activated as early as possible as and possibly not later than 180 s for this type of scenario.

6.2.4 Fire Inside Tunnel, Ventilation Fans Operational, Constant (Peak) Heat Release Rate = 16 MW

The above analysis is carried out with fans starting at 60 s, 120 s, 150 s and 180 s. The heat release rates at these times are 1.08 MW, 4.32 MW, 6.75 MW and 9.72 MW respectively. This implies that during this course of fire detection and smoke control, the maximum heat release rate of fire i.e. 16 MW is not achieved. The fire curve indicates that maximum HRR is achieved at t = 231 seconds. The fans should however be designed to control smoke for this maximum HRR. Figure 6.10 shows the thermal environment inside the tunnel at various times for the case of 16 MW constant heat output. It can be seen that the smoke moves sufficient distance within 15 seconds of start of fire. The fire will be detected well before this time and ventilation fans will be started. The calculated critical velocity from Wu and Bakar formula for 16 MW fire is 2.51 ms^{-1}. The CFD simulations predict the critical velocity to be equal to 3.75 ms^{-1}. This velocity corresponds to the case when flow is induced at a uniform rate from tunnel portal at t = 15 s and whole smoke is pushed in downstream direction in 30 s i.e. at t = 45 s. Figure 6.11 shows the temperature distribution at t = 45 s

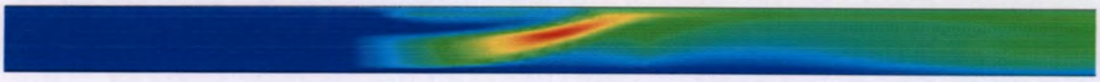
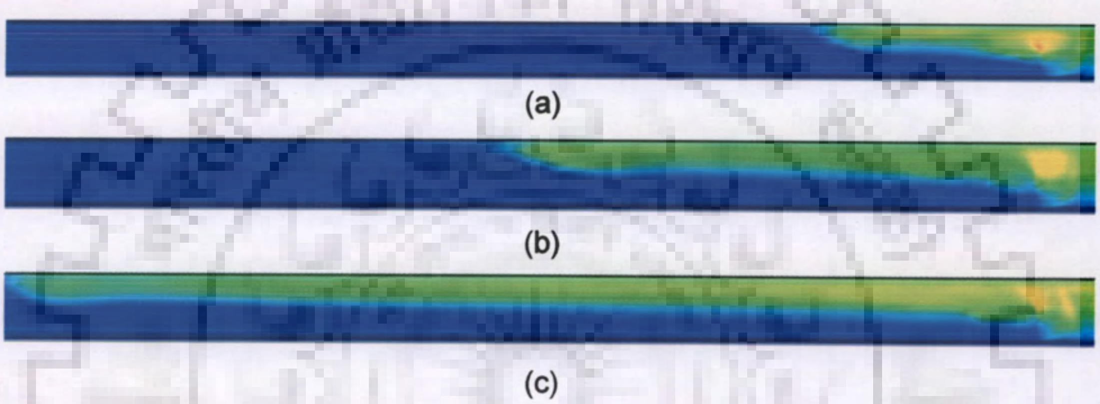


Figure 6.9. Temperature distribution on the vertical central plane through the fire source and tunnel portals at t= 210 s when fans start at t = 120 s with velocity (vertical component) of supply and exhaust fan as 9 ms^{-1}



(a)

(b)

(c)

Figure 6.10. Temperature distribution on the vertical central plane through the fire source and tunnel portals at t = (a) 15 s, (b) 30 s, (c) 60 s for HRR of 16 MW. (only one half of tunnel portion is shown in Figure)

Figure 6.11. Temperature distribution on the vertical central plane through the fire source and tunnel portals at t= 45 s when fans start at t = 15 s with uniform induced velocity of 3.75 ms^{-1}

when fans are started at uniform rate at t= 15 s. From Figure 6.8 the fan (located in ceiling) velocity required for smoke control is 10.6 ms^{-1}. The fans are therefore to be designed for supplying of air at 10.6 ms^{-1}. Table 6.5 shows the predicted and calculated critical velocities for the case of 16 MW heat output with the smoke pushed in downstream direction within 30 s of start of ventilation fans.

Table 6.5. V_c from Empirical Formulae and CFD Simulation for 16 MW fire

HRR	V_c from Wu & Bakar Formula	V_c from CFD simulation assuming uniform induced flow	V_c from CFD simulation with fans located at ceiling
16 MW	2.51 ms^{-1}	3.70 ms^{-1}	10.6 ms^{-1}

6.3 CONCLUDING REMARKS

The fire environment inside a section of tunnel, assuming a variable HRR from the fire source is predicted. It is found that the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals moves towards the fire source. The effect of ventilation flow rate, both from supply fans and exhaust fans on thermal environment inside tunnel is analyzed. It is found that for this type of tunnel configuration where fans are located in the ceiling and inclined to a certain angle with the plane perpendicular to ceiling, higher supply and exhaust velocities are required to produce the desired critical velocity. The velocities of fan required to produce desired velocity in the longitudinal direction of the tunnel can be determined from Figure 6.7. The exhaust fans do not influence the velocity in upstream area but is necessary for smoke removal in the downstream direction. It is also necessary that fans are activated to full speed within three minutes of starting of fire in order for the ventilation system to be effective for smoke removal.

CHAPTER - VII

CONCLUSIONS AND RECOMMENDATIONS

7.1 CONCLUSIONS

Conclusion of the thesis has been described section-wise as given below:

7.1.1 Comparative Study of Long Distance Road Tunnels

A number of long distance road transport tunnels have been studied with respect to fire protection measures installed in them. These are then compared with guidelines of NFPA 502 and proposed directive of European Commission. All the tunnels have different fire safety features depending upon their requirement. The above assessment showed that the designs, provisions etc. of the existing tunnels should not form a basis for design of new tunnels. There are differences in some provisions as given in NFPA 502 and proposed directive of European Commission. In order to provide better safety features inside tunnels, the most stringent measures are proposed and are given in section 3.3. It is suggested that in future tunnels these observations / recommendation should be complied. The higher safety standards will ensure less fatalities and less damage to tunnel structure in case of fire.

7.1.2 Comparative Study of Zone model and Field Model for Simulating a Tunnel Fire

The fire environment inside a tunnel caused by burning wood crib was simulated by a zone model CFAST and a field model, CFX and compared. The results showed that

(i) Both CFAST and CFX results predicted that the smoke temperature changed with a pattern roughly similar to those of heat release rate.

(ii) The CFAST results predicted that the value of smoke temperature of fire room is highest in case of tunnel divided into maximum number of compartments i.e. fifteen. The size of compartment was found to influence the temperature in the fire room.

(iii) In CFAST, the minimum value of smoke layer interface height for fire room increases as the number of compartments in which tunnel is divided decreases.

(iv) During computation by CFAST, location of Fire source in the compartment is found to be a critical issue. It is observed that for accuracy of results the size of compartment should be so chosen that fire source is placed in the centre of compartment as far as possible; in no case it should be positioned at or near vents. There is no such limitation while computing using CFX.

(v) In CFAST the maximum number of compartments that can be simulated is thirty but computational difficulty was encountered when tunnel was divided into more than fifteen compartments.

(vi) In CFAST, the temperature of monitoring point is taken to be upper layer temperature if the smoke layer interface height is below the monitoring point and lower layer temperature is considered if smoke layer interface height is above the monitoring point. This results in sharp increase or sharp downfall of temperatures where as CFX predicts temperatures at any location.

(vii) CFX, on the other hand provided detailed information regarding smoke movement inside a tunnel. It not only predicts temperature at any locations, it also predicts other detailed features like flame tilt etc.

It was seen that CFD analysis offered detailed insight into fire dynamics, however CFD analysis were computationally expensive. On the other hand, CFAST required very less computational power and were quick to perform, hence can be used to get overview of the results. CFAST predicts the interface height between the two layers and gas temperatures well. It can be concluded that CFAST can only provide global view in terms of two layers while CFD should be used as an effective tool for smoke modeling and subsequently evaluating ventilation strategies.

7.1.3 Evaluation of Ventilation Strategies of Underground Metro Rail Transport System with Jet Fans

The effect of longitudinal ventilation through jet fans mounted axially below the ceiling on smoke movement inside the tunnel was studied. The numerical model to be used for predicting environment inside tunnel was validated using results of one of Steckler experiments. It was found that model using CFD well predicted the results of Steckler experiments at distances away from the fire source. Then using same model parameters thermal environment for a 4 MW fire for a section of tunnel was predicted. The effect of uniform and non-uniform induced airflow in tunnel, on thermal environment inside the tunnel was studied. The study revealed that

(i) The representation of fire source as a volumetric heat source and the assumption of height of fire source is a critical issue. The importance of assuming suitable height and volume of the volumetric heat source, appropriate to the heat release rate of the fire in order to have reasonable prediction of the flow pattern and the flame temperature was illustrated. It was deduced that height can be taken to be approximately equal to flame length predicted by Heskestad.

(ii) Under conditions of natural ventilation, the smoke moves symmetrically along the crown in both directions and cool entrained air from bottom of tunnel portals move towards the fire source. This qualitative predicted by CFX confirms to experimental observations reported in the literature.

(iii) The centre line plume temperatures predicted by CFX when compared with empirical relation shows that the temperatures are in close agreement away from the fire source while at points near fire the temperature differs significantly. This can be because of assumed height of volumetric heat source and the assumption that heat is uniformly distributed inside volumetric source. This shows that better modeling of fire source is required through use of combustion models.

(iv) In case of uniform induced air flow, CFX predicted higher critical velocity than the empirical relations by about 43%. This may be because of reduced cross-section of tunnel at fire site due to height of fire source or because thermal heat transfer through radiation is modeled. In absence of any data on heat loss due to radiation, the empirical formula assumes that heat lost due to radiation is 30% of total heat release rate.

(v) In case of uniform induced air flow, on the upstream side temperatures remained around 300 K up to 3 m height from the bottom of tunnel and at a distance up to 45 m from the inlet to the tunnel.

(vi) When induced flow is non-uniform, higher critical axial fan velocities are required. The stratified layer of smoke in downstream region is not formed. A well-stratified smoke layer in downstream region is not observed even when exhaust fans are activated. The air flow tilts the plume downstream and its trajectory comes to floor in the downstream area from where the exhaust fan sucks this air to exhaust. As the fan velocity is increased, inlet air does not touch floor downstream, rather it gets exhausted to atmosphere, while smoke layer remains at bottom. It implies that escape can take place only through upstream areas.

(vii) In case of non uniform induced air flow, on upstream side the temperatures remained below 320 K up to a height of 3 m from the bottom of tunnel and the velocities were below 5 m/s. However, higher velocities (6.3 m/s) were predicted close to fire source at certain times.

7.1.4 Evaluation of Ventilation Strategies of Underground Metro Rail Transport System having Jet Injection Type Ventilation System

The influence of the fire HRR curve slope on the smoke flow dynamics in a realistic tunnel model having jet injection ventilation system is investigated. Ventilation shafts are located in the ceiling near the tunnel portals and inclined at 10° to the plane of the ceiling through which air is discharged. In case of fire two cases are studied: (i) fans activated immediately and achieve its full speed as soon as the fire is detected (ii) fans activated at delayed times to take into account the response time for the fans to achieve its maximum speed. The velocity of supply and exhaust fans necessary to remove smoke in 30 sec from the upstream direction is determined. The study revealed that:

(i) higher supply and exhaust fan velocities are required to produce the desired critical velocity. The velocities of fan required to produce desired axial velocity can be determined from Figure 6.7.

(ii) the exhaust fans do not influence the velocity in upstream area but is necessary for smoke removal in the downstream direction.

(iii) fans should be activated to full speed within three minutes of starting of fire in order for the ventilation system to be effective for smoke removal. However during this time, the maximum heat release rate of fire is not achieved. The fans should therefore be designed to control smoke for this maximum HRR.

7.2 REMARKS

It is hoped that the research work presented here provides an insight into efficient design of the ventilation system of tunnels and helps to evaluate the mode of operation of the facility. The effects of fire and mechanical ventilation on personnel, evacuating and fire fighting, can be predicted and visualized.

The research work will help in designing efficient ventilation system and evaluating the evacuation procedures and integrity of egress way.

7.3 RECOMMENDATIONS FOR FUTURE WORK

(i) The studies conducted above utilizes volumetric heat source model to represent fire source. The region near the fire source cannot be appropriately simulated using this model. The determination of volume of heat source is a very critical issue. The future studies should be carried out using appropriate combustion models to adequately model the area near the fire.

(ii) The walls of the tunnel are assumed to be adiabatic. In this situation, the heat is not lost to the walls resulting in higher temperatures and longer backed up length of smoke. Thus it is a very conservative approach and defines worst case scenario. The other extreme is to assume a constant wall temperature leading to maximum rates of heat transfer. In future studies the heating of the wall can be modeled by solving thermal conduction equation within the wall. However, this would require a much finer grid resolution near the wall and specification of the properties of the wall.

(iii) It is recommended that some experimental studies should be carried out so that CFD results are validated and reliance can be placed on future numerical simulations conducted using the above physical models and boundary conditions.

PUBLICATIONS

1. Jain, S., Sharma, T. P., Shashi, & Kumar, S. "Comparative Study of Fire Safety Measures In Long Distance Road Transport Tunnels". Proceedings of *Fourth International Symposium on Safety Science & Technology*, Shanghai, China, Oct 25-28, 2004.

2. Jain, S., Shashi, Kumar, S., Sharma, & T. P. "Numerical Simulation of Fire in a Tunnel: Comparative Study of CFAST and CFX Predictions". *Tunneling and Underground Space Technology*, 23(2), March 2008.

3. Jain, S., Shashi, Kumar, S., & Sharma, T. P. "Studies on Evaluation of Ventilation Strategies of Underground Metro Rail Transport system". Proceedings of Int. conf. on *7th Asia-Oceania Symp. on Fire Science and Technology*, Hong Kong, September 20-22, 2007.

4. Jain, S., Shashi, & Kumar, S. "Numerical Studies on Evaluation of Smoke Control of Underground Metro Rail Transport System having Inclined Fans". *Fire Safety Journal*. (communicated).

REFERENCES

Abanto, J., Reggio, M., Barrero, D., & Petro, E. (2007)
Prediction of fire and smoke propagation in an underwater tunnel
Tunnelling and Underground Space Technology, 22, 90-95.

AEA Technology. (2002)
CFX 5.5.1 User Manual.

Altinakar, M. S., Weatherill, A., & Nasch, P. H. (1997)
The use of zone model in predicting fire and smoke propagation in Tunnels
Proceedings of the 9th Aerodynamics and Ventilation of Vehicle Tunnel, Aosta Valley, Italy.

Anderson, J. D. (1995)
Computational Fluid Dynamics: The basics with applications
MacGraw- Hill. USA.

Apte, V. B., Green, A. R., & Kent, J. H. (1991)
Pool fire plume flow in a large-scale wind tunnel
Proc. 3rd Int. Symp. on Fire Safety Science (IAFSS), 425-434, Boston.

Armstrong, J., Bennett, E. C., Matthews, R. D., Smith, T. W., & Tabarra, M. (1993)
The Ventilation of Vehicle Tunnels by Jet Fans -The Axisymmetric Case
Proceedings of the Seminar on Installation Effects in Fan Systems, 59-64, London: Mechanical Engineering Press.

ASHRAE (2003)
Enclosed Vehicular Facilities
ASHRAE Handbook - HVAC Applications, Chapter 13: ATLANTA, GA.

ASTRA - Swiss Federal Road Office. (2001)
Ventilation of Road Tunnels, Selection of System, Design and Operation. Switzerland

Atkinson, G. T., Bettis, R. J., Jagger, S. F., and Lea, C. J. (1996)
Fire Tests and CFD Modeling of Critical Velocity Behavior
ITC Conference, Health and Safety Laboratory, London.

Bettelini, M., Brandt, R., & Riess, I. (2001)
Progress in Tunnel Ventilation – The Mont Blanc Tunnel
Paper presented at AITES – ITA 2001 World Tunnel Congress, June 10-13, Milano.

Bettis, R. J., Jagger, S. F., & Wu, Y. (1993)
Interim validation of tunnel fire consequence models: summary of phase 2 tests
Project Report IR/L/FR/93/11, Health & Safety Laboratory, UK.

Bettis, R. J., Jagger, S. F., Macmillan, A. J. R., & Hambleton, R. T. (1994a)
Interim validation of tunnel fire consequence models: summary of phase 1 tests
Project Report IR/L/FR/94/02, Health & Safety Laboratory, UK.

Bettis, R. J., Jagger, S. F., & Moodie, K. (1994b)
Reduced Scale Simulations of Fires in Partially blocked Tunnels
Proc. Int. Conf. Fires in Tunnels, 162-186, October 10-11, Boras.

Bettis, R. J., Jagger, S. F., Lea, C. J., Jones, I. P., Lennon, S., & Guilbert, P. W. (1994c)
The use of physical and mathematical modelling to assess the hazards of tunnel fires.
Proceedings of 8th International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, 439-470, July 6-8, Liverpool, UK: BHRA, The Fluid Engineering Centre.

Bickel, J. O., Kuesel, T. R., & King, E. H. (1982)
Tunnel Engineering Handbook
New York: Van Nostrand Reinhold.

Bickel, J. O., Kuesel, T. R., & King, E. H. (1996)
Tunnel Engineering Handbook (2nd ed.)
New York: Chapman & Hall.

Borello, D., Giuli, G., & Rispoli, F. (2002)
A CFD methodology for fire spread and radiative effects simulation in longitudinal ventilation tunnels: Application to the memorial tunnel
International PHOENICS User Conference, September 23 – 27, Moscow.

Brandeis, J., & Bergmann, D. J. (1983)
A numerical study of tunnel fires
Combustion Science and Technology, 35, 133-155.

Budnik, E. K., & Walton, W. D. (1990)
Computer fire models
In A. E. Cotte (Ed.), Fire Protection Handbook, 16th ed., Section 21, Chapter 4. Qunicy, MA: NFPA.

Carlsson, J. (1999)
Fire Modelling using CFD-An introduction to Fire Safety Engineers, Part I
Report No.5025, Department of Fire Safety Engineering, Lund University, Lund.

Carvel, R., & Marlair, G. (2005)
A history of fire incidents in tunnels
In A. Beard & R. Carvel (Eds.) The Handbook of Tunnel Fire Safety (Chapter 1, 1-41) London: Thomas Telford.

Carvel, R., Beard, A., & Jowitt, P. (2001)

How much do tunnels enhance the HRR of fires?
Proceedings of the 4th International Conference on Safety in Road and Rail Tunnels, 457- 466, Madrid, Spain.

Chaiken, R. F., Singer, J. M., & Lee, C. K. (1979)
Model coal tunnel fires in ventilation flow
Investigations Report no. 8355, US Bureau of Mines.

Chang, X., Laage, L. W., & Greuer, R. E. (1990)
A user's manual for MFIRE: a computer simulation program for mine ventilation and fire modelling
Report no. IC245. US Bureau of Mines.

Charters, D. (2005)
Control volume modeling of tunnel fires
In A. Beard & R. Carvel (Eds.) The Handbook of Tunnel Fire Safety (Chapter 13, 284-298) London: Thomas Telford.

Charters, D. A., Gray, W. A., & McIntosh, A. C. (1994)
A computer model to assess fire hazards in tunnels (FASIT)
Fire Technology, 30, 134-154.

Chen, F. (2000)
Smoke propagation in road tunnels
Appl Mech Rev, 53 (8), 207 – 218.

Chow, W. K. (1996)
Simulation of tunnel fires using a zone model
Tunneling and Underground Space Technology, 11, 21–236.

Cox, G. (1995)
Compartment fire modeling
In G.Cox (Ed.), Fundamentals of Fire (329–404) London: Academic Press.

Daish, N. C., & Linden, P. F. (1994)
Interim validation of tunnel fire consequence models: comparison between the phase 1 trials data and the CREC/HSE near-fire model
Report no. FM88/92/6, Cambridge Environmental Research Consultants.

D'Albrand, N., & Bessiere, C. (1992)
Fire in a road tunnel: comparison of tunnel sections
Proc. of 1st Int. Conf. on Safety in Road and Rail Tunnels, 439-449, November 23-25, Basel, Switzerland: University of Dundee and Independent Technical Conferences Ltd.

Danziger, N. H., & Kennedy, W. D. (1982)
Longitudinal ventilation analysis for the Glenwood Canyon tunnels
Proc. Of 4th International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, 169-186, March 23-25, York, England: BHRA Fluid Engineering Centre.

Directorate General for Public Works and Water Management. (2002)
Project "Safety Test" – Report on Fire Tests
Civil Engineering Division, Utrecht, The Netherlands. (Available online at www. minvenw.nl)

Directorate of Public Roads. (1992)
Norwegian Design Guide - Road Tunnels
Public Roads Administration, Norway.

DOT. (1997)
Subway environment simulation (SES) computer program
Version 4: User's manual and programmer's manual", Issued as Volume II of Subway Environmental Design Handbook. Pub. No. FTA-MA-26- 7022-97-1. US Department of Transportation, Washington, DC.

Drysdale, D. (1998)
An introduction to Fire Dynamics (2nd ed.)
New york: John Wiley & Sons.

Egilsrud, P. E., (1984)
Prevention and Control of Highway Tunnel Fires
US Department of Transportation / Federal Highway Administration Report No FHWA-RD-83-032. Retreived on 17.12.2003 from www.fhwa.dot.gov/bridge/prevent1.htm; http://www.fhwa.dot.gov/bridge/prevabst.htm#appb

Eisner, H. S., & Smith, P. B. (1954)
Convection effects from underground fires: the backing of smoke against the ventilation
Research Report no. 96, Safety in Mines Research Establishment.

Feizlmayr, A. H. (1976)
Research in Austria on tunnel fire
Proceedings of 2nd International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, Paper J2, March 23-25, Cambridge, UK: BHRA, The Fluid Engineering Centre.

Ferziger, J. H., & Peric, M. (1996)
Computational Methods for Fluid Dynamics
Springer Verlag, Newyork.

Fletcher, D. F., Kent, J. H., Apte, V. B., & Green, A. R. (1994)
Numerical Simulation of Smoke Movement from a Pool Fire in a Ventilated Tunnel
Fire Safety Journal, 23, 305-325.

Flow3D Releases 2.4(1990), 3.1(1991) and 3.2(1992)
User Manual Computational Fluid Dynamics Services
AEA Industrial Technology, Harwell Laboratory, Didcot, Oxon, UK.

Forney, G. P and Moss, W. F. (1992)
Analyzing and Exploiting numerical characteristics of zone models
NISTIR-4763, National Institute of Standards and Technology, Gaithersburg, MD, USA.

Forney, G.P., Cooper, L.Y., 1990
The consolidated compartment fire model (CCFM) computer application
CCFM.VENTS – Part-II: software reference guide, NISTIR 90-4343, National Institute of Standards and Technology, Gaithersburg, MD, USA.

Gao, P. Z., Liu, S. L., Chow, W. K., & Fong, N. K. (2004)
Large eddy simulations for studying tunnel smoke ventilation
Tunnelling and Underground Space Technology, 19, 577-586.

Gao-shang, Y., Yong-lin, A., Li-min, P., & Jin-hua, Z. (2006)
Simulation of smoke flow and longitudinal ventilation in tunnel fire
Transactions of Nonferrous Metals Society of China, 741 – 746.

Gasiorek, J. (1991)
Some Theoretical Considerations of Air Flow in Longitudinal Road Tunnel Ventilation Using Jet Fans
Internal Report, School of Engineering Systems & Design. South Bank University.

Gobeau, N., & Zhou, X. X. (2004)
Evaluation of CFD to predict smoke movement in complex enclosed spaces - Application to three real scenarios: an underground station, an offshore accommodation module and a building under construction
Research Report 255. Available online
http://www.hse.gov.uk/research/rrpdf/rr255.pdf

Gobeau, N., Ledin, H. S., & Lea, C. J. (2002)
Guidance for HSE Inspectors: Smoke movement in complex enclosed spaces- Assessment of Computational Fluid Dynamics
Health & Safety Laboratory Report- HSL/2002/29. (Available online www.hse.gov.uk/research/hsl_pdf/2002/hsl02-29.pdf)

Grant, G. B., Jagger, S. F., & Lea, C. J. (1998)
Fires in tunnels
Philosophical Transactions of the Royal Society of London, 356, 2873 – 2906.

Haack, A. (2007)
Design Fire Scenarios
Technical Report - Part 1. Thematic Network FIT 'Fire in Tunnels'.
(Available online from www.etnfit.net)

Haack, A. (1995)
Introduction to the EUREKA project / BMBF research project 'Fire protection in underground transportation facilities'

Int. Conf. on Fire Protection in Traffic Tunnels, 6-18, September 12 - 13, Dresden, Germany.

Haerter, A. (1994)
Fire tests in the Offenegg tunnel in 1965
Proc. of Int. Conf. on Fires in Tunnels, 95-214, October 10-11, Boras, Sweden.

Heselden, A. J. M. & Hinkley, P.L. (1970)
Smoke travel in shopping malls: Experiments in co-operation with Glasgow Fire Brigade Part 1
Fire Research Note no. 832, Joint Fire Research Organization.

Heselden, A. J. M. (1976)
Studies of fire and smoke behavior relevant to tunnels
Proc. of 2nd International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, March 23-25, Cambridge, UK: BHRA,The Fluid Engineering Centre.

Heskestad, G. (1995)
Fire Plumes
Chapter 2-2. SPE Handbook of Fire Protection Engineering, Society of Fire Protection Engineers, Quincy, MA, USA.

Highway Authority. (1999)
Design Manual for Roads and Bridges
Volume 2, Highway Structure Design Materials, Section 2: Special Structures, Part 9, BD 78/79: Design of Road Tunnels. UK.

Hinckley, P. L. (1970)
The flow of hot gases along an enclosed shopping mall: a tentative theory
Fire Research Note no. 807, Fire Research Station. UK.

Hu, L.H., Huo, R., Peng, W., Chow, W. K., & Yang, R. X. (2006)
On the maximum smoke temperature under the ceiling in tunnel fires
Tunnelling and Underground Space Technology 21, 650-655.

Hu, L. H., Huo, R., Wang, H. B., & Yang, R. X. (2007)
Experimental and Numerical Studies on Longitudinal Smoke Temperature Distribution Upstream and Downstream from the Fire in a Road Tunnel
Journal of Fire Sciences, 25, 23-43.

Hwang, C. C., Chaiken, R. F., Singer, J. M., & Chi, D. N. H. (1977)
Reverse stratified flow in duct fires: a two-dimensional approach
16th Symp. (Int.) on Combustion, 1385-1395. The Combustion Institute.

Hwang, C. C, & Edwards, J. C. (2001)
CFD Modeling of smoke reversal

Proceedings of the international conference on engineering fire protection design, 376–87. Bethesda, Maryland, USA: Society of Fire Protection Engineers.

Ingason, H. (1995a)
Findings concerning the rate of heat release
Int. Conf. on Fire Protection in Traffic Tunnels. 94-103, September 12-13, Dresden, Germany.

Ingason, H. (1995b)
Effects of ventilation on heat release rate of pool fires in a model tunnel
Report 1995:55, SP Technical Research Institute, Boras, Sweden.

Ingason, H. (1995c)
Design Fires in Tunnels
Conference Proceedings of Asiaflam 95, 77-86, March 15-16, Hong Kong.

Ingason, H. (2005)
Fire Development in Large Tunnel Fires
8th International Symposium on Fire Safety Science, September 18-23, Beijing, China.

Ingason, H. (2006)
Design Fires in Tunnels
Second International Symposium on Safe & Reliable Tunnels, Innovative European Achievements, Lausanne.

Ingason, H., Gustavsson, S., & Dahlberg, M. (1994)
Heat release rate measurements in tunnel fires
SP Report 1994:08, Swedish National Testing and Research Institute, Sweden.

Ingason, H., & Lonnermark, A. (2003)
Large scale fire tests in the Runehamar Tunnel: Gas Temperature and Radiation
Proceedings of International Conference on Catastrophic Tunnel Fires, SP Report 2004:05, 81-92, November 20-21, Boras, Sweden.

Ingason, H., & Lonnermark, A. (2005)
Heat Release Rates from Heavy Goods Vehicle Trailers in Tunnels
Fire Safety Journal, 40,646-668.

Innovative Research Inc. / Parsons Brinckerhoff Inc. (2000)
SOLVENT Version 1.0
ITC Conference, London.

IOS-98. (1998)
Manual for the Design, Construction and Operation of Tunnels
Spain.

ISO. (1999)
Fire Safety Engineering - Part 2: Design fire scenarios and design fires
Technical Report ISO/TR 13387-2:1999.

Jojo, Li, S. M., Chow, W. K. (2003)
Numerical studies on performance evaluation of tunnel ventilation safety systems
Tunnelling and Underground Space Technology, 18, 435-452.

Jones, W. W., & Forney, G. P. (1990)
A programmer's reference manual for CFAST, the useful model of fire growth and smoke transport
NISTTN-1283, National Institute of Standards and Technology, Gaithersburg, MD, USA.

Jones, W. W., Matsushita, T., & Baum, H. R. (1992)
Smoke movement in corridors – Adding the horizontal Momentum Equation to a zone model
Proceedings of 12th joint Panel Meeting of the UJNR Panel on Fire Research and Safety, 42-54, Oct. 27 – Nov 2, Tsukuba, Japan: Building Research Inst., Ibaraki, Japan Fire Research Inst., Tokyo. Available online from http://www.fire.nist.gov/bfrlpubs/fire94/art154.html

Jones, W. W. (1985)
A multicompartment model for the spread of fire, smoke and toxic gases
Fire Safety Journal, 9, 55-79.

Kang, K. (2006)
Computational Study of Longitudinal Ventilation Control during an Enclosure Fire within a Tunnel
Journal of Fire Protection Engineering, 16, 159-181.

Kashef, A., Bénichou, N., & Lougheed, G. (2003)
Numerical Modeling of Movement and Behavior of Smoke Produced from Fires in the Ville-Marie and L.-H. – La Fontaine Tunnels: Literature Review
IRC-RR-141.

KIVI - Royal Institute of Engineers. (1993)
Ventilation of Road Tunnels
Netherlands.

Ku, A. C., Doria, M. L., & Lloyd, J. R. (1976)
Numerical modelling of unsteady buoyant flows generated by a fire in a corridor
Proceedings of 16th International Symposium on Combustion, 1373-1384. The Combustion Institute.

Kumar, S., & Cox, G. (1985)
Mathematical modeling of fires in road tunnels

Proceedings of 5th International Symposium on the Aerodynamics and Ventilation of Road Tunnels. Lille, France.

Kumar, S. and Cox, G. (1986)
Mathematical Modeling of Fire in Road Tunnels Validation of JASMINE
Transport and Road Research Laboratory, Contractor Report 28, Department of Transport.

Kumar, S., & Cox, G. (1988)
Radiant heat and surface roughness effects in the numerical modeling of tunnel fires
6th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, September 27 -29, Durham, England: BHRA, The Fluid Engineering Centre.

Kumar, S. & Cox, G. (2001)
Some guidance on "correct" use of CFD models for fire applications with examples
9th International Interflam Conference Proceedings, 2, 823-834, September 17-19, Edinburgh, Scotland.

Kunikane, Y., Kawabatta, N., Takekuni, K. and Shimoda, A. (2002a)
Heat release rate induced by gasoline pool fire in a large-cross-section tunnel
Proceedings of Fourth International Conference on Tunnel Fires, Basel, 387-396, December 2-4, Switzerland.

Kunikane, Y., Kawabatta, N., Ishikawa, T., Takekuni, K. and Shimoda, A. (2002b)
Thermal fumes and smoke induced by bus fire accident in large cross sectional tunnel
Proceedings of the 5th JSMEKSME Fluids Engineering Conference, November 17-21, Nagoya, Japan.

Lacroix, D. (1997)
New French Recommendations for Fire Ventilation in Road Tunnels
9th International Conference on Aerodynamics and Ventilation of Vehicle Tunnels, October 6-8, Aosta Valley, Italy.

Lea, C. J. (1994)
CFD modeling of fire on a Channel Tunnel HGV shuttle
Health & Safety Executive Project Report IR/L/FR/94/13, Health & Safety Laboratory, Buxton.

Lea, C. J. (1995)
CFD modeling of the control of smoke movement from tunnel fires using longitudinal ventilation
Abstracts of the 1st Eur. Symp. on Fire Safety Science, August 21- 23, Zurich: ETH Institute for Structural Engineering.

Lee, C. K., Chaiken, R. F., & Singer, J. M. (1979)
Interaction between duct fires and ventilation flow: an experimental study
Combustion Science & Technology, 20, 59-72.

Lemaire, T. (2003)
Runehamar Tunnel Fire Tests: Radiation, Fire Spread and Back Layering
International Symposium on Catastrophic Tunnel Fires (CTF), SP Report 2004:05, 105-116, Boras, Sweden, November 20-21.

Leschziner, M. A. (1992)
Turbulence Modelling Challenges Posed by Complex Flows
Roomvent '92, Third International Conference, Vol 1, pp 31-58, September 2-4 , Aalborg, Denmark.

Lonnermark, A. & Ingason, H. (2003)
Large scale fire tests in the Runehamar Tunnel: heat release and radiation
Proceedings of International Conference on Catastrophic Tunnel Fires, SP Report 2004:05, 93-104, November 20-21, Boras, Sweden.

Lonnermark, A., & Ingason, H. (2005)
Gas Temperatures in Heavy Goods Vehicle Fires in Tunnels
Fire Safety Journal, 40,506-527.

Lotsberg, G. (2009)
The worlds longest Tunnel Page
http://www.lotsberg.net/

Magnussen, B. F,, Hjertager, B. H., Olsen, J. G., & Bhaduri, D. (1978)
Effects of Turbulent Structure and Local Concentrations on Soot Formation
Seventeenth International Symposium on Combustion, The Combustion Institute.

Malhotra, H. L. (1995)
Goods Vehicle Fire Test in a Tunnel
2nd Int. Conf. on Safety in Road and Rail Tunnels, 237-244, April 3-6, Granada, Spain.

Mankbadi, R. R. (1995)
Turbulence Models for CFD
In W. G. Habashi & M. M. Hafez (Eds.) Computational Fluid Dynamics Techniques (589-619): Gordon and Breach Publishers.

Massachusetts Highway Department / Federal Highway Department. (1996)
Memorial Tunnel Fire Ventilation Test Program
Interactive CD-ROM & Comprehensive Test Report. Parsons Brinckerhoff, New York.

Massachusetts Highway Department and Bechtel/Parsons Brinckerhoff (1999)
Memorial Tunnel Fire Ventilation Test Program – Phase IV
Report. Massachusetts.

McCaffrey, B. J. (1979)
Purely buoyant diffusion flames: some experimental results
Final Report NBSIR 79-1910, Centre for Fire Research, National Bureau of Standards, Gaithersburg, MD, USA.

McGrattan, K. (Ed.) (2005)
Fire Dynamics Simulator (Version 4): Technical Reference Guide
NIST Special Publication, 1018. NIST, Gaithersburg, MD, USA. Available online http://www.fire.nist.gov/bfrlpubs/fire04/art100.html

McGrattan, K. B., & Hamins, A. (2002)
Numerical Simulation of the Howard Street Tunnel Fire
NISTIR 6902, Building and Fire Research Laboratory, National Institute of Standards and Technology (NIST), Gaithersburg, MD, USA.

McGrattan, K., & and Forney, G. (2005)
Fire Dynamics Simulator (Version 4) User's Guide
National Institute of Standards and Technology (NIST), Gaithersburg, MD, USA.

Megret, O., Vauquelin, O., & Chasse, P. (1995)
A reduced scale tunnel for the study of fire induced smoke control
3rd Int. Conf. on Safety in Road and Rail Tunnels, 713-723, 9-11 March, Nice, France.

Migoya, E., Crespo, A., Garcia, J., & Hernandez, J. (2008)
A simplified model of fires in road tunnels. Comparison with three-dimensional models and full-scale measurements
Tunnelling and Underground Space Technology, Available online from www.elsevier.com

Miles, S., Kumar, S., & Andrews, R. (1999)
Validation of a CFD Model for Fires in the Memorial Tunnel
International Conference on Tunnel Fires and Escape from Tunnels, 5-7 May, Lyon, France.

Ministry of Public Works. (1999)
Safety of Traffic in Road Tunnels with Particular Reference to Vehicles Transporting Dangerous Materials. Italy.

Ministry of Public Works. (2000)
Safety in the Tunnels of the National Highways Network
(Inter-Ministerial Circular No. 2000-63) France: Ministry of Public Works.

Mizutani, T., Horiuchi, K. & Akiyama, K. (1982)
Experimental study of tunnel fires
Journal of the Japan Road Association, 24-28.

Moodie, K. & Jagger, S. F. (1992)

The King's Cross fire: results and analysis of scale model tests
Fire Safety Journal, 18, 83-103.

NFPA. (1972)
NFPA 502T (Tentative) Standard for Limited Access Highways, Tunnels, Bridges and Elevated Structures
Quincy, MA: National Fire Protection Association.

NFPA. (1981)
NFPA 502 - Recommended Practice on Fire Protection for Limited Access Highways, Tunnels, Bridges, Elevated Roadways and Air Right Structures
Quincy, MA: National Fire Protection Association.

NFPA. (1983)
NFPA 130 - Standard for Fixed Guideway Transit Systems
Quincy, MA: National Fire Protection Association.

NFPA. (1995)
NFPA 130 - Standard for Fixed Guideway Transit Systems
Quincy, MA: National Fire Protection Association.

NFPA. (1998)
NFPA 502 - Standard for Road Tunnels. Bridges and other Limited Access Highways
Quincy, MA: National Fire Protection Association.

NFPA. (1999)
NFPA 80 - Standard for Fire Doors and Fire Windows
Quincy, MA: National Fire Protection Association.

NFPA. (2000)
NFPA 130 - Standard for Fixed Guideway Transit and Passenger Rail Systems
Quincy, MA: National Fire Protection Association.

NFPA. (2001)
NFPA 502-Standard for Road Tunnels, Bridges, and Other Limited Access Highways
Quincy, MA: National Fire Protection Association.

NFPA. (2002a)
NFPA 70 - National Electrical code
Quincy, MA: National Fire Protection Association.

NFPA. (2002b)
NFPA 110 - Standard for Emergency and Standby Power
Quincy, MA: National Fire Protection Association.

NFPA. (2002c)
NFPA 70B - Recommended Practice for Electrical Equipment Maintenance

Quincy, MA: National Fire Protection Association.

NFPA. (2003)
Fire Protection Handbook
19th edn. NFPA, Quincy, MA: National Fire Protection Association.

Numajiri, F., & Furukawa, K. (1998)
Short Communication: Mathematical Expression of Heat Release Rate Curve and Proposal of Burning Index
Fire and Materials, 22, 39-42.

NVF - Nordic Road Association. (1993)
Ventilation of Road Tunnels
Report No.6, Sub-Committee 61.

Oka, Y., Atkinson, G. T., Jagger, S. F. & Deall, A. (1996)
Control of smoke flow in tunnel fires
Fire Safety Journal, 25, 305-322.

Owen, M. S. (ed.) (2003)
ASHRAE Handbook - Heating, Ventilating and Air-conditioning (HVAC) Applications
Atlanta, Georgia: American Society of Heating, Refrigerating and Air-conditioning Engineers (ASHRAE)

Patankar, S. V. (1980)
Numerical Heat Transfer and Fluid Flow
USA: Hemisphere

Peard, M. (1996)
Statistics on breakdowns, accidents and fires in French road tunnels
Proceedings of the Ist International Conference on Tunnel Incident Management, 347-365, May 13-15, Korsor, Denmark.

Perard, M. & Brousse, B. 1994)
Full size tests before opening two French tunnels
8th Int. Symp. on Aerodynamics and Ventilation of Vehicle Tunnels, 383-408, July 6-8, Liverpool.

Perard, M. (1992)
Organization of fire trials in an operated road tunnel
1st Int. Conf. on Safety in Road and Rail Tunnels, 161-170, November 23-25, Basel, Switzerland.

PIRAC. (1975)
Road Tunnels Committee. Report to the XVth World Road Congress, Mexico city, Mexico
Permanent International Association of Road Congresses (now World Road Association (PIARC)), Paris, France.

PIRAC. (1979)

Technical Committee on Road Tunnels
Proceedings of the XVIth World Road Congress, September 16-21, Vienna Austria. Permanent International Association of Road Congresses (now World Road Association (PIARC)), Paris, France.

PIRAC. (1983)
Technical Committee on Road Tunnels
Proceedings of the XVIIth World Road Congress, October 8-15, Sydney Australia. Permanent International Association of Road Congresses (now World Road Association (PIARC)), Paris, France.

PIRAC. (1987)
Technical Committee on Road Tunnels
Report No.5 Proceedings of the XVIIIth World Road Congress, September 13-19, Brussels, Belgium. Permanent International Association of Road Congresses (now World Road Association (PIARC)), Paris, France.

PIRAC. (1991)
Technical Committee on Road Tunnels
Proceedings of the XIXth World Road Congress, September 22-28, Marrakech, Morocco. Permanent International Association of Road Congresses (now World Road Association (PIARC)), Paris, France, 19.05.B.

PIRAC. (1995a)
Committee (C5), Road tunnels
Proceedings of the XXth World Road Congress, September 3-9, Montreal Canada. (PIARC), 20.05.B.

PIRAC. (1995b)
Classification of Tunnels, Existing Guidelines and Experiences. Recommendations
World Road Association (PIARC), Paris, France, 05.03.B.

PIRAC. (1995c)
Road Safety in Tunnels
World Road Association (PIARC) Paris, France, 05.04.B.

PIRAC. (1999)
Fire and Smoke Control in Tunnels
Report no. 05.05.B
The World Road Association (PIARC / AIPCR)

PIRAC. (2005)
Systems and Equipment for Fire and Smoke Control in Road Tunnels
World Road Association (PIARC), Paris, France.

Planning Guidelines. (2002)
Austria Guideline Code for the Planning, Construction and Maintenance of Roads (RVS)
Operation and Safety Facilities (OaSF), Tunnel Equipment, RSV 9.282.

Pucher, K. (1994)
Fire Tests in the Zwenberg Tunnel (Austria)
Proc. Int. Conf. on Fires in Tunnels, 187-194, 10-11 October, Boras.

PWRI - Public Works Research Institute. (1993)
State of Road Tunnel Equipment Technology in Japan - Ventilation, Lighting, Safety Equipment
Japan.

RABT-Road and Transportation Research Association. (2002)
Guidelines for Equipment and Operation of Road Tunnels (RABT 02)
Germany: Federal Ministry of Traffic.

Rhodes, N. (1994)
Review of Tunnel Fire and Smoke Simulations
Proceedings of the 8th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels. BHR Group Conference Series Publication No.12, 471-486, London: Mechanical Engineering Publications Limited.

Rhodes, N. (1996)
Smoke Modeling & Critical Velocity in Tunnels
ITC Conference, London.

Rodi, W. (1980)
Turbulence Models and Their Application in Hydraulics
State-of-the-Art paper of the IAHR, Delft Hydraulics, The Netherlands.

RWS - Ministry of Transport, Public Works and Water Management (1998)
Technical Standards for the Provision and Installation of Fire Protection for Tunnels
Rijkwaterstaat, Netherlands, 1998.

RWS - Ministry of Transport, Public Works and Water Management. (1991)
Technical Standards for the Provision and Installation of Ventilation of Road Tunnels
Rijkwaterstaat, Netherlands.

Saito, N., Yamada, T., Sekizawa, A., Yanai, E., Watanabe, Y. & Miyazaki, S. (1995)
Experimental Study on Fire Behaviour in a Wind Tunnel with a Reduced Scale Model
2nd Int. Conf. on Safety in Road and Rail Tunnels, 303-310, Granada, Spain.

Schabacker, J., Bettelini, M., & Rudin, C. H. (2001)
CFD study of temperature and smoke distribution in a railway tunnel with natural ventilation system
Third International Conference on Tunnel Fires and Escape from Tunnels, Washington DC, USA

Simcox, S., Wilkes, N. S., & Jones, I. P. (1992)
Computer simulation of the flows of hot gases from the fire at King's Cross underground station
Fire Safety Journal, 18(1), 49-73.

Sinai Y. L. (2003)
Field Modeling of a Steckler Experiment: An Example of the Relationship between the Level of Modeling and Accuracy
Presented at Fourth International Seminar on Fire & Explosion Hazards, September 8-12, Londonderry, UK: University of Belfast.

Spalding, D. B. (1971a)
Mixing and Chemical Reaction in Steady Confined Turbulent Flames
Proc. of 13th International Combustion Symposium, 649-657. The Combustion Institute.

Spalding D. B. (1971b)
Concentration fluctuations in a round turbulent free jet
Chemical Engineering Science, 26, 95-107.

Steckler, K. D., Quintiere, J. G., & Rinkinen, W. J. 1982
Flow Induced by Fire in a Compartment
National Bureau of Standards, NBSIR 82-2520.

Steinert, C. (1994)
Smoke and Heat Production in Tunnel Fires
Proc. Int. Conf. on Fires in Tunnels, 123-137, October 10-11, Boras.

Studiengesellschaft Stahlanwendung e.V. (Ed.) (1995)
Fires in Transport Tunnels: Report on full-scale tests
EUREKA-Project EU499: FIRETUN. Dusseldorf.

Suzuki, K., Harada, K., & Tanaka, T. (2002)
A multilayer zone model for predicting fire behavior in a single room
Fire safety science -Proceedings of the Seventh International Symposium, 851–862.

Suzuki, K., Tanaka, T., Harada, K., & Yoshida, H. (2004)
An application of a multilayer zone model to a tunnel fire
Proceedings of the Sixth Asia Oceania Symposium on Fire Science and Technology, March 17–20, Daegu, Korea.

Swedish National Road Association. (1999)
Tunnel 99-General Technical Specification
Sweden.

Te Velde, K. (1988)
A computer simulation for longitudinal ventilation of a road tunnel with incoming and outgoing slip roads

Proc, of 6th Int. Symp. on the Aerodynamics and Ventilation of Vehicle Tunnels (179-201), September 27-29, Durham, UK, BHRA Fluid Engineering Centre.

Thomas, P. H. (1970)
Movement of smoke in horizontal corridors against an air flow
Inst. Fire Enqrs, Q. 30, 45-53.

Thomas, P. H. (1995)
The growth of fire ignition to full involvement
In G. Cox (Ed.), Combustion Fundamentals of Fire (273–328) London: Academic Press.

Tuovinen, H., Holmstedt, G., and Bengtson, S., (1996)
Sensitivity Calculations of Tunnel Fires Using CFD.
Fire Technology, Second Quarter, 99-119.

United States Department of Transportation. (1975)
Subway Environmental Design Handbook, Volume I-Principles and Applications
Associated Engineers. Report No. UMT A-DC-06-00 I 0-74-1, Transit Development Corporation, Washington, DC.

Vantelon, J. P., Guelzim, A., Quach, D., Kim Son, D., Gobay, D., & Dallent, D. (1991)
Investigation of fire-induced smoke movement in tunnels and stations: an application to the Paris Metro
Proc. of 3rd Int. Symp. on Fire Safety Science, 907-918, Edinburgh, UK.

Vauquelin, O., & WU, Y. (2006)
Influence of tunnel width on longitudinal smoke control
Fire Safety Journal. Accepted 2006, Available online www.elsevier.com

Versteeg, H. K., & Malalasekera, W. (1995)
An Introduction to Computational Fluid Dynamics: The Finite Volume Method.
Longman.

Vistnes, J. (2004)
Validation of Phoenics 3.5 for Modelling Tunnel Ventilation Systems Under Fire Conditions
PHOENICS 10th International User Conference, May 3-5, Melbourne, Australia.

Wang J., Hua J., Kumar K., & Kumar S. (2006)
Evaluation of CFD Modeling Methods for Fire Induced Airflow in a Room
Journal of Fire Sciences, 24, 393-411.

Woodburn, P. J., & Britter, R. E. (1996 a)
CFD Simulations of a Tunnel Fire – Part I
Fire Safety Journal, 26, 35-62.

Woodburn, P. J., & Britter, R. E. (1996 b)
CFD Simulations of a Tunnel Fire – Part II
Fire Safety Journal, 26, 63-90.

Wu Y., Bakar M. Z. A. (2000)
Control of Smoke Flow in Tunnel Fires Using Longitudinal Ventilation Systems – A Study of the Critical Velocity
Fire Safety Journal, 35(4), 363–390.

Wu, Y., Stoddard, J. P., James, P., & Atkinson, G. T. (1997a)
Effect of slope on control of smoke flow in tunnel fires
Proceedings of 5th Int. Conf. on Fire Safety Science, 1225-1236, Melbourne, Australia.

Wu, Y., Bakar, M. Z. A., Atkinson, G. T., & Jagger, S. (1997b)
A study of the effect of tunnel aspect ratio on control of smoke flow in tunnel fires
9th Int. Conference on Aerodynamics and Ventilation of Vehicle Tunnel, 573-587, October 6-8, Aosta Valley, Italy.

Xiaojun, C. (2008)
Simulation of temperature and smoke distribution of a tunnel fire based on modifications of multi-layer zone model
Tunnelling and Underground Space Technology, 23, 75–79.

Xue, H., Hihara, E., & Saito, T. (1993)
Turbulence model of fire-induced air flow in a ventilated tunnel
Int. Journal of Heat & Mass Transfer, 36, 1739-1748.

Xue, H., Ho, J. C., & Cheng, Y. M. (2001)
Comparison of different combustion models in enclosure fire simulation
Fire safety Journal, 36, 37-54.

Yang, K. H., & Lee, S. K. (1999)
Smoke Management and Egress Design Analysis of an Underground Railway Station
Journal of Applied Fire Science, 9(2), 153-171.

Yuan, & You. (2007)
CFD simulation and optimization of the ventilation for subway side-platform
Tunnelling and Underground Space Technology, 22, 474-482.

Websites

http://europa.eu.int/comm/transport/road/roadsafety/roadinfra/tunnels/index_en.htm.
Safety in European Road Tunnels.
Retreived on 10th Dec 2003.

http://europa.eu.int/scadplus/leg/en/lvb/l24146.htm.
SCADPlus. *Minimum levels of Safety in European Road Tunnels.*
Retreived on 10.12.2003.

http://news.bbc.co.uk/2/hi/europe/1858436.stm
Inside the Mont Blanc Tunnel
Retrevied on 20.12.2003.

www.delhimetrorail.com/

www.en.wikipedia.org/wiki/Namma_Metro

www.gruxa.ch/gotthard_road.htm
History of GotthardStrasse
Retrevied on 25.12.2003.

www.janes.com/articles/Janes-Urban-Transport-Systems/Chennai-Metro-Rail-Limited-India.html

www.kolmetro.com

www.konkanrailway.com

www.mapsofindia.com/kochi/metrorail-project.html

www.mrtunnel.com/page3.htm
Report of Task force for Technical Investigation of the 24th March 1999 Fire in the Mont Blanc Vehicular Tunnel, France
Retrevied on 28.11.2003.

www.tunneldufrejus.com/english/presentation.htm
In Between France & Italy, working to your Dimension
Retreived on 1.12.2003