COMPUTATIONAL FLUID DYNAMICS SIMULATION OF EXHAUST VENTILATION AND JET FAN SYSTEMS DURING A FIRE EVENT IN VELODROME

SAYED MOJTABA TABIBIAN

DEPARTMENT OF MECHANICAL ENGINEERING
FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2017

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF EXHAUST VENTILATION AND JET FAN SYSTEMS DURING A FIRE EVENT IN VELODROME

SAYED MOJTABA TABIBIAN

DISSERTATION SUBMITTED IN FULFILMENT OF THE REQUIRMENTS FOR THE MASTER OF MECHANICAL ENGINEEIRNG

DEPARTMENT OF MECHANICAL ENGINEERING
FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2017

UNIVERSITY OF MALAYA
ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: **Sayed Mojtaba Tabibian**

(I.C/Passport No:

Matric No: **KGY 150025**

Name of Degree: **Master of Mechanical Engineering**

Title of Project Paper/Research Report/Dissertation/Thesis ("this Work"):

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF EXHAUST VENTILATION AND JET FAN SYSTEMS DURING A FIRE EVENT IN VELODROME

Field of Study: **CFD (Computational Fluid Dynamics) - Fluid Mechanics**

I do solemnly and sincerely declare that:

(1) I am the sole author/writer of this Work;
(2) This Work is original;
(3) Any use of any work in which copyright exists was done by way of fair dealing and for permitted purposes and any excerpt or extract from, or reference to or reproduction of any copyright work has been disclosed expressly and sufficiently and the title of the Work and its authorship have been acknowledged in this Work;
(4) I do not have any actual knowledge nor do I ought reasonably to know that the making of this work constitutes an infringement of any copyright work;
(5) I hereby assign all and every rights in the copyright to this Work to the University of Malaya ("UM"), who henceforth shall be owner of the copyright in this Work and that any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of UM having been first had and obtained;
(6) I am fully aware that if in the course of making this Work I have infringed any copyright whether intentionally or otherwise, I may be subject to legal action or any other action as may be determined by UM.

Candidate's Signature Date:

Subscribed and solemnly declared before,

Witness's Signature Date:

Name:

Designation:

ABSTRACT

Ensuring human safety in the case of fire in tunnels, buildings, car parks or any other premises is one of the objectives of fire engineering design. Many researchers have focused on fire safety elements like heat, velocity and smoke dynamics because many fire victims are killed by toxic gases and smoke items such as carbon monoxide induced by fire. Carbon monoxide poses the most deadly risk to people and will not easily be discharged. In this study, the effectiveness of ventilation system in fire safety is investigated by using computational fluid dynamics (CFD) simulation results on smoke spill. The case study was conducted at Velodrome (Indoor) to determine the smoke propagation and air flow pattern during fire. For this purpose, the full scale model has been developed in the CFD with the total of four million elements for mesh generation. The velocity inlet was adopted for fresh air inlets form doors as well as mass flow of fire. Outflow was selected for exhaust ducting system. About 4 MW fire at two different positions were simulated at the Velodrome and the time based simulation was done for total of 0-90 minutes considering evacuation time. The effect of the smoke spill by simulating with exhausted system for both switched “ON” and “OFF” of the simulator. Also the results were investigated and analyzed in 3-dimensional plane. The results for velocity, smoke, and temperature distributions are presented at 2m, 4m, 10m, and 13m height from floor level for both of fire position scenarios. This study focuses on the problem of smoke evacuation and the possibility of operating the fans and exhausted system, during different fire at Velodrome. The primary goal of smoke management is to facilitate safe exit in the case of fire and it is also crucial in saving a property since it is more costly to maintain it than to build it. Thus, the control and removal of smoke and gases from burning building is a vital component in any fire protection scheme. The findings demonstrate that by activating a fan and exhaust system, the risk of people’s life and damaged property can be reduced because the source of fire can be removed completely and thus, smoke poses no threat.

ABSTRAK

Memastikan keselamatan manusia dalam hal api dalam terowong, bangunan, tempat letak kereta atau mana-mana premis lain adalah salah satu objektif reka bentuk kejuruteraan kebakaran. Ramai penyelidik telah menumpukan kepada elemen keselamatan kebakaran seperti haba, halaju dan dinamik asap kerana banyak mangsa kebakaran dibunuh oleh gas toksik dan barangan asap seperti karbon monoksida yang disebabkan oleh kebakaran. Karbon monoksida menimbulkan risiko paling berbahaya kepada manusia dan tidak akan mudah dilepaskan. Dalam kajian ini, keberkesanan sistem pengudaraan dalam keselamatan kebakaran disiasat dengan menggunakan keputusan simulasi cecair pengkomputeran (CFD) terhadap tumpahan asap. Kajian kes itu dijalankan di Velodrome untuk menentukan penyebaran asap dan corak aliran udara semasa kebakaran. Untuk tujuan ini, model skala penuh telah dibangunkan dalam CFD dengan jumlah sebanyak empat juta elemen untuk penjanaan mesh. Saluran halaju telah digunakan untuk saluran udara segar yang membentuk pintu serta aliran jisim kebakaran. Outflow dipilih untuk sistem saluran ekzos. Kira-kira 4 MW api di dua kedudukan berbeza disimulasikan di Velodrome dan simulasi berasaskan masa dilakukan selama 0-90 minit memandangkan masa pemindahan. Kesan tumpahan asap dengan simulasi dengan sistem yang habis-habis untuk kedua-dua beralih "ON" dan "OFF" simulator. Hasilnya juga disiasat dan dianalisis dalam bidang 3-dimensi. Keputusan untuk halaju, asap dan pengagihan suhu dibentangkan pada ketinggian 2m, 4m, 10m, dan 13m dari tingkat lantai untuk kedua-dua senario kedudukan api. Kajian ini memberi tumpuan kepada masalah pemindahan asap dan kemungkinan mengendalikan peminat dan sistem yang lelah, semasa api yang berbeza di Velodrome. Matlamat utama pengurusan asap adalah untuk memudahkan keluar selamat dalam hal kebakaran dan juga penting dalam menyelamatkan harta kerana ia lebih mahal untuk mengekalkannya daripada membinanya. Oleh itu, kawalan dan penghapusan asap dan gas dari bangunan terbakar adalah komponen penting dalam mana-mana skim perlindungan kebakaran. Penemuan menunjukkan bahawa dengan mengaktifkan kipas dan sistem ekzos, risiko kehidupan orang dan harta yang rosak dapat dikurangkan kerana sumber api dapat dihapus sepenuhnya dan dengan demikian, asap tidak menimbulkan ancaman

ACKNOWLEDGEMENT

In the name of Allah The Most Gracious The Most Merciful, first and foremost, all praises to Allah who has created us and gave us intelligence and guidance. And peace is upon our prophet the teacher of all mankind and peace is upon his family.

First off all, I would like to thank and convey my sincere appreciation to my Supervisor Dr. Nik Nazri Bin Nik Ghazali for the encouragement, guidance and support in the completion of this project.

I am also thankful to my wife who has supported me in all levels of my study and life.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	:	Computational fluid Dynamics
HRR	:	Heat Release Rate
AR	:	Aspect Ratio
PPM	:	Part Per Million
CFM	:	Cubic Feet per Minute
NFPA	:	National Fire Protection Association
NVF	:	Naturally Ventilated Floors
MVF	:	Mechanical Ventilated Floor
SHEVS	:	Heat Exhaust Ventilation System
TWA	:	Time Weighted Average
CO2	:	Carbon Dioxide
CO	:	Carbon Monoxide
ON	:	on
OFF	:	off
MW	:	Mega Watt
SHC	:	Smoke and Heat Control
UM	:	University of Malaya

CHAPTER 1: INTRODUCTION

1.1 Research Background

The majority of deaths in fire are due to the inhalation of smoke. Smoke causes direct visual obscuration by absorbing and scattering light, reduces the visibility of escape signs and may cause pain to the eyes and respiratory tract. Smoke may also decrease or eliminate the capacity for building occupants to escape due to reduced visibility and thermal hazards (NFPA, 2015). Another consideration is the toxic hazard of gases such as carbon monoxide, carbon dioxide and hydrogen cyanide.

Different studies on fire hazards, show that the predominant cause of hazard is smoke, not the temperatures. So the smoke extraction systems or fire ventilation systems are a main point for thermal design in Velodrome, car park, tunnel safety and etc. In recent years, many researchers focused on ventilation systems or smoke extraction systems during a fire event, including Smoke back-layering flow length (Du *et al*., 2016), maximum temperature, critical velocity of fire. Controlling the fire events and its influences to the fire environments are very important, typically by ventilation systems. Various fire environments such as tunnels, car parks, buildings and etc. have different philosophies based on fire ventilations. On the other hand, different types of smoke control systems exist. In general, some fire research group focus on preventing the move of smoke upstream of the fire. That is, smoke back layering flow, while others opt to reduce smoke propagation rates by maintaining low tunnel air velocities. Another important consideration is to study on the temperature distribution along car parks, tunnels or etc. with mechanical ventilation. (Li *et al*., 2013).

According to Li *et al*., (2012), in a long corridor fire such as tunnel, ceiling jet can be formed to reduce temperature through boundary layer in the smoke layer contacting the tunnel ceiling (Merci, B., & Shipp, M.,2012). Stated that due to low ceiling height in car parks, natural vertical venting systems cannot be very common. Therefore, one of vital studies in design of ventilation systems is the optimized control of smoke distribution. In order to reach successful design of ventilation systems, it is important to comprehend the characteristics of smoke distribution. For example, in order to recognize the ventilation system control it is necessary to have correct information about smoke or air movement within the fire environments. Monitoring of smoke or air movement strongly depend on correct smoke or air velocity readings, i.e. in fire tunnels on their location in tunnels and trustiness of the sensors.

1.2 Background of Problem

Effective smoke control system or ventilation systems during fire events is very important for saving lives since that bifurcation flow of smoke can reduce smoke propagation rates and give more time for passengers to escape (Barbato *et al*., 2014). The important parameters of smoke extraction in case of fires including characteristics of velocity or smoke layer thickness are needed to be investigated by researchers in detail.

Research done by Barbato *et al*. (2014) showed that the most deadly factor in fire events is due to smoke where a huge quantity of toxic gases as a result of an incomplete combustion is released. Thus, the development of an effective ventilation system is a key element for personnel safety during evacuation in fire events. Hence, an increasing number of researches on smoke exhaust methods have been presented in recent years. The existence of various toxic gases, low oxygen content and high temperatures threat to personnel safety in fire events. While the ability to firefighting and rescue as well as, high radiation heat, low visibility poses risk evacuation and high temperatures also result in an extension of the fire. So the smoke management system is essential:

- To save lives by facilitating human evacuation and extinguishment;
- To reduce risk of explosions;

- To support firefighting operations and rescue;
- To decrease damage to structure, equipment and surrounding facilities in fire environments;

This research was motivated to present a smoke exhaust method for personnel safety during fire events in Velodrome.

1.3 Problem Statement

Ventilation systems with their proper operation play the main role in fire safety. Ventilation systems should able to provide the acceptable air quality for personnel safety in fire event. These systems facilitate rescue conditions and provide tenable environment during a smoke or fire event. Ventilation systems provide sufficient fresh air, while dealing with ventilating environment during a fire emergency. Although fire tests in different environments such as car park, tunnels, mine, buildings have been performed since the early 1960s, but there are still some issues on fire ventilation systems and little interest was given to fire ventilation in Velodrome. Therefore, in order to fill this gap, more researchers are required to use fire case studies in other environments such as Velodrome to determined the smoke propagation and air flow pattern during fire condition. The design and operation of the fire ventilation in Velodrome during a fire event is a major topic. This research will develop the CFD open source model developer on the smoke spill at Velodrome to control fire and smoke incidents.

1.4 Research Objectives

The research seeks to address the following objectives:

1) To develop a ventilation system during a fire event
2) To investigate the smoke propagation and air flow pattern during fire condition with CFD simulation

1.5 Research Scope

In order to design a ventilation system during fire incidents, computational fluid dynamics (CFD) will be used and performed.

The scope of this research is to identify effectiveness of ventilation systems in fire safety and to investigate computational fluid dynamics (CFD) simulation results on smoke spill. A case study had been conducted at Velodrome to determined the smoke propagation and air flow pattern during fire condition. However, the real size model is developed in the CFD simulation.

1.6 Research Contributions and Significance

The major contributions of this study can be summarized as in the following paragraphs:

1) A study of the effectiveness of exhaust ventilation and jet fan systems in Velodrome during a fire event.

Based on the reviews done on the research articles and studies conducted on the trend of smoke extraction system or ventilation systems during a fire event, this research has provided airflow velocity at Velodrome for fire ventilation.

2) Computational Fluid Dynamics (CFD) simulation on the smoke spill at Velodrome.

This research work will be involved with CFD model to simulate the fire situations and smoke spread at Velodrome. CFD is a simulation tools that research works apply to solve fundamental flow equations for modeling fire and smoke behaviors.

1.7 Organization of Thesis

In this report, five chapters are involved to illustrate the modeling a fire ventilation system and flow of smoke in fire situations from its conception to its simulation. A brief introduction to ventilation systems and smoke extraction system was given in the Chapter 1. Chapter 2 presents a literature reviews where it discusses prior studies and the gap that exists in the studied area. The project methodology and instruments applied in this study is described in Chapter 3. Chapter 4 describes and presents a smoke propagation and air flow pattern during fire event at Velodrome. Chapter 5, reports about the conclusion of the project.

CHAPTER 2: LITERATURE REVIEW

The purpose of this chapter is to reveal literature related to one's particular area of study and shows some of the fundamental aspects of the project.

2.1 Introduction to Fire Ventilation and Smoke Control Systems

The exhaust ventilation system is meant to eliminate contaminants. It must provide the necessary control of the air which is full of contaminants and their sources. In this system, particulates, vapors and gases are controlled by controlling the air. The components of a typical exhaust system usually include a hood, a duct, an air cleaner, an air mover and a vent or an outlet. Since the hood is where the air is drawn into the system and it is relatively close to the source of the contaminants, the design of the hood must be effective. The major categories of hood are the enclosed hood, the partially enclosing hood and the exterior hood (Cao *et al*., 2017). Exhaust ventilation and smoke in buildings with large enclosed spaces is generally provided by a Smoke and Heat Exhaust Ventilation System (SHEVS). Hot smoky gases are collected at high level and vented to the outside. Supply of inlet replacement air below the smoke layer is crucial and must be included in the design along with the sizing of the smoke venting system. (NPFA, 2015).

The most mortal factor during fire events is smoke. The development of an effective design of ventilation systems and smoke exhaust method is the most important protection measures for human health during evacuation in fire events. In the context of this research thesis, fire ventilation systems including smoke production will be named fire ventilation. Smoke control during fire ventilation is achieved by dilution and evacuation of smoke. It is needed that smoke filled air can be replaced by clean air, which is created mechanically in through the portals. Dilution can reduce the concentrations of toxic gases to improve tenability.

Fire ventilation system uses the extract ventilation to catch the contaminants from being breathed by personnel in workstations. The three main elements of Fire ventilation system are hood, duct, fan and discharge as showed at Figure 2.1.

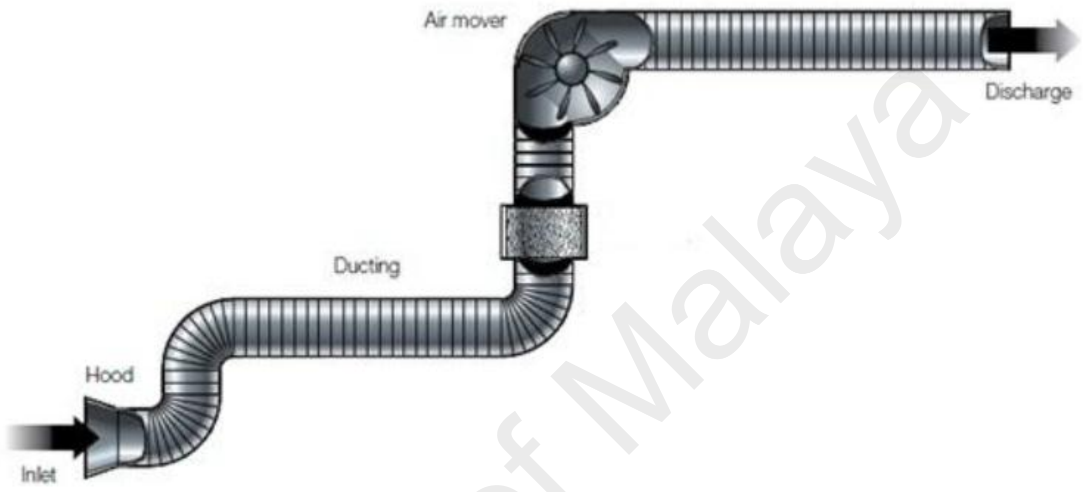


Figure 2.1: Common parts of ventilation system

2.1.1 Hood

The entry point of contaminants into the exhaust ventilation system is the hood. The hood is designed to grant the necessary amount of air in order to control contaminants and draw them into the system. The hood can be of a simple round or rectangular opening or it can be specially designed for the mentioned purpose. There are three types of hood that can be fixed into an exhaust ventilation system. The first is the enclosed hood. In this type of hood, the amount and speed of air allowed into the enclosed cover prevents the contaminants and their sources to escape. As for the next type of hood, which is the exterior hood, it is usually installed outside and hence, far from the source of the contaminant. The right amount and speed of air is released to capture any contaminant at its furthermost distance from the hood and then, draws the contaminant into the system. This amount and speed of air is called the capture velocity of the hood. The next type of hood is the partial-enclosing hood or also known as the receiving hood. It can be defined as the hood that receives contaminants. In order for this type of hood to function properly, the flow of air must be accurate to remove contaminants so that they do not escape (Cao *et al*., 2017).

Exhaust hood is widely utilized in many industries like metallurgy, mineral, mechanical, chemical, textiles medicine, health and tobacco. Since exhaust hood is widely used in the tobacco industry like the other industries. It is apparent that exhaust hoods are only efficient when the sources of contaminants are enfolded. Moreover, the efficiency of the hoods decreases rapidly with the distance of the sources of contaminants. This is due to the non-directional of the airflow entering the hood and clean air within the hood is also sucked in at the same time. The exhaust hoods are often becoming inefficient since they are placed at the app position due to access requirement (Cao *et al*., 2017).

2.1.2 Duct

This chapter presents the discussion on numerous components of a duct system. It also presents the general design and the structural characteristics of these components. Just as important as selecting the appropriate hood, selecting the components of a duct system should not be taken for granted. The purpose of a duct system is to surround and direct the flow of air in a ventilation system from one point to another. In an exhaust system, the duct prevents the contaminated air from mixing with the workroom air as it is removed. In a supply system, the duct directs the supply air to the point where it is required. The duct can be considered as a pathway which the air in a ventilation system travels. The most common duct used in ventilation systems has a round cross section because round ducts result in a more uniformed speed profile within the system. However, in certain situations, the other cross-section configurations can be used.

2.1.3 Fan

A machine used to create Flow, be called Fan. The fan includes of a rotating arrangement of blades which act on fluid or any mass. The rotating assembly of blades and hub is known as an impeller, a runner or a rotor. Usually, it is placed in housing or a case. This may direct the airflow and also increase safety by preventing any object from contacting the fan blades. Most fans are powered by electric motor; however, other sources of power may also be utilized like hydraulic motors and internal combustion engines. General applications include personal thermal comfort and climate control e.g., floor fan or an electric table, vehicle engine cooling systems e.g. in front of a radiator, machinery cooling systems e.g., inside computers, ventilation, fume extraction, winnowing e.g., separating chaff of cereal grains, removing dust e.g. in a vacuum cleaner, provide draft for a fire. There are three main types of fans. They are axial, cross flow (also called tangential) and centrifugal (also called radial).

2.2 Common Fire Ventilation Methods

The fire and smoke distribution in fire situations must be controlled by ventilation systems or smoke extraction system. Different environments such as car park, tunnels, buildings and etc. Base on fire ventilation with fire situations have various philosophies. Some researchers prefer to prevent movement of smoke upstream of the fire to be called back-layering, others focus on maintaining low air velocities to reduce fire smoke movement. Among types of smoke extraction methods, jet fan is one of the most effective fire ventilation systems that suppression of the fire and enables early evacuation. Thus, a number of scalars encourage the fire brigades in equipment with a movable jet fan in order to improve the fire environment for rescue and extinguishment fires (Kashef *et al*., 2013). The studies on movable fans have showed that the functionality of them is better than the fixed fans located at the ceiling because its distance can be freely modified to adapt to the complicated fire situations.

Table 2.1: recent studies summary (2007–2017) that focus on common ventilation

Reference	Environmental	Solution	Method
Zhong *et al.*(2013)	Tunnel	FDS	longitudinal ventilation
	Remark: A bifurcation flow of fire smoke in tunnel with longitudinal ventilation - The smoke bifurcation flow and the generation of low temperature region in the middle of tunnel ceiling have a significant influence on the effectiveness of tunnel smoke extraction and Reverse smoke flows which were formed by the impact of the ceiling jet with sidewalls. The reverse smoke flows would be brought to downstream by the longitudinal Ventilation. The smoke vent should not be located in the low temperature region.		
Fan *et al.*(2014)	Tunnel	Large eddy	Natural ventilation
	Remark: Vertical shaft arrangement effects on natural ventilation performance in tunnel fires- the reasons influence the natural ventilation performance are boundary layer separation and plug-holing. the shaft dimensions and Shafts on the tunnel ceiling influence the natural ventilation		
Merci, B., & Shipp, M.(2012)	Car park	CFD	Mechanical ventilation
	Remark: A study of heat and smoke control during fires in large car parks- this study is effective for forced ventilation as air flow reach the fire source with presence of beams in various directions and water effects.		
Deckers *et al.*(2012)	*Car park*	CFD	Mechanical ventilation
	Remark: Full-scale experiments Smoke management in fire situation - The flow patterns and the smoke extraction flow rate and The exact position of extraction fans on the smoke pattern and the presence of a transversal beam are effects of jet fans.		
Kashef *et al.*(2013)	Tunnel	One dimensional	Natural ventilation
	Remark: Ceiling temperature distribution and smoke diffusion in tunnel fire throughout natural ventilation- the smoke mass flow rate through the shafts at the ceiling and ratio of inertia force provided by the incoming fresh air to the buoyancy force provided by the hot smoke.		
Ura *et al.*(2014)	Tunnel	experimental results	Natural ventilation
	Remark: Behavior of smoke extraction base natural ventilation during a Fire in a shallow urban road tunnel - natural ventilation through the openings in the ceiling given the natural buoyancy of the hot smoke.		
Yao *et al.*(2016)	Tunnel	experimental results	Longitudinal ventilation
	Remark: smoke back-layering flow length effects by vertical Shaft distance from fire source Smoke control of tunnel fires by combining longitudinal ventilation and transverse exhausting, particularly involving the issues of smoke back-layering flow length.		
Du *et al.*(2016)	Tunnel	experimental results	Longitudinal &transverse ventilation
	Remark: Efficiency evaluation of longitudinal and transverse ventilation for smoke control and thermal in tunnel- compatibility of transverse and longitudinal ventilation. The critical velocity in the fire branch and the velocity for preventing smoke penetration into the downstream branch that is adjacent to the smoke discharge route should be guaranteed. For a transverse ventilation mode applied multiple operational modes for jet fan systems.		
Li *et al.*(2012)	Tunnel	experimental results	Longitudinal ventilation
	Remark: Fire-induced flow temperature along tunnels with longitudinal ventilation - Estimate temperature of fire flow beneath the ceiling of tunnels or corridors for design of fire detectors or sprinklers.		
Heidarinejad *et al.*(2016)	Tunnel	FDS	Longitudinal ventilation

	Remark: study of two fire sources in a road tunnel: Considering different arrangement of obstacles - The effects of the distance between two fire sources and also distance between vehicles obstruction from the fire on the critical ventilation velocity		
Beard, A. N. (2016)	Tunnel	Theoretical result	Longitudinal ventilation
	Remark: water mist effect on the critical heat release rate for fire to spread from an initial fire.		
Yi *et al.*(2015)	Tunnel	experimental results	Transverse ventilation
	Remark: Define heat exhaust coefficient by the proportion of the heat exhausted from smoke duct and individual exhaust inlet and exhaust fans in total heat released by the fire.		
Zhang *et al.*(2016)	Tunnel	FDS	Longitudinal ventilation
	Remark: Prediction of smoke back-layering length in the subway tunnel under different longitudinal ventilations with metro train - Predict the smoke back-layering length based on influence of vehicle length on the smoke back-layering.		
Li *et al.*(2013)	Tunnel	experimental results	Longitudinal ventilation
	Remark: Effect of ceiling extraction system on the smoke thermal stratification in the longitudinal ventilation tunnel - air flow affected on the ceiling extraction on the smoke thermal stratification.		
Willstrand *et al.*(2015)	*Bus &Toilet*	experimental results	N/A
	Remark: Detection of fires in the toilet compartment and driver sleeping compartment of buses and coaches-based on full scale tests Install a smoke detector in the ceiling and heat or smoke detector in the concealed space of the fan.		
Su, C., & Yao, C.(2016)	*Building*	experimental results	Mechanical &Natural
	Remark: Performance measurement of a smoke extraction system for buildings in full-scale hot smoke test - This method used a string of vertical smoke layer measuring instruments composed of several approved photoelectric smoke detectors, as well as a light attenuation measuring device composed of luminance meters to conduct tests on the effect of makeup air.		
Zhang *et al.*(2016)	*Tunnel*	experimental results	Longitudinal ventilation
	Remark: effect on blockage of metro train on the smoke back-layering in subway tunnel fires - ventilation velocity base on metro train length is developed to predict the smoke back-layering length.		
Ji *et al.*(2016)	*Tunnel*	experimental results	Longitudinal ventilation
	Remark: flame merging behaviors from two pool fires along the longitudinal centerline of model tunnel with natural ventilation- By using dimensional analysis and introducing a correlation factor. The effective ceiling flame length is developed, involving the heat release rate, pool size, spacing and effective tunnel height.		
Meroney *et al.*(2013)	*Military firing range*	CFD	Firing range ventilation
	Remark: Simulation of ventilation and smoke movement in a large military firing range - test three alternative ventilation arrangements and Removing intermediate exhaust vents along the length of the tunnel greatly to Providing inlet guides and some venting around the entrance to the tunnel.		
Hidalgo *et al.*(2015)	*Building*	experimental results	N/A
	Remark: Performance criteria for the fire safe use of thermal insulation in buildings - Design of insulation systems in buildings which is based on the design of thermal barriers for controlling the onset of paralysis.		
Hull *et al.*(2016)	*Building*	experimental results	N/A
	Remark: Quantification of toxic hazard from fires in buildings - estimation of the toxic fire hazard By combining the toxic product yields with the mass loss range. A		

	methodology is proposed for quantifying the volume of toxic effluent produced by burning construction materials within an enclosure.		
Li *et al.*(2016)	*Building*	experimental results	Natural ventilation
	Remark: Smoke spread velocity along a corridor induced by an adjacent compartment fire with outdoor wind - A model of smoke velocity in the corridor with outdoor wind established and Proposed correlations for smoke spread velocity in the corridor under the effect of outdoor wind.		
Chen *et al.*(2016)	*Building*	experimental results	Shaft ventilation
	Remark: Vertical temperature distributions in ventilation shafts during a fire- determining temperature distributions in ventilation shafts and presents a model for forecasting vertical temperature distributions in ventilation shafts where the influence of both shaft walls and ventilation.		
Wang *et al.*(2016)	*Coal Mine*	FDS	Mine ventilation
	Remark: Information fusion of plume control and personnel escape during the emergency rescue of external- A ventilation system Setting up smoke control measures to determine the personnel escape conditions and routes.		
Fan *et al.*(2013)	*Mine Laneway*	FDS	Mine ventilation
	Remark: Smoke movement characteristics under stack effect in a mine. Investigating of fire on smoke movement under stack effect inside an inclined laneway by increasing either angle or length of the inclined laneway contribute to reducing the backflow length in the horizontal laneway and thus leading to more smoke flowing into the inclined laneway.		
Yuan *et al.*(2016)	*Mine Laneway*	FDS	Mine ventilation
	Remark: Modelling CO spread in underground mine fires – considered the airflow leakage effect on CO concentration reduction.		
Zhang *et al.*(2011)	*Coal Mine*	FDS	Mine ventilation
	Remark: numerical simulation in coal mine fire for escape capsule installation – Determined the escape capsule installation location in the passageway.		
Mei *et al.*(2016)	*Tunnel*	experimental results	Mechanical venting
	Remark: Evolution characteristics of fire smoke layer thickness in a mechanical ventilation tunnel with multiple point extraction- investigate the characteristics of smoke layer thickness and plug-holing phenomenon in a mechanical ventilation tunnel with multiple point extraction system.		
Wang *et al.*(2016)	*Coal Mine*	Numerical method	Mine ventilation
	Remark: A study of Mining-induced void distribution and application in the hydro-thermal investigation and control of an underground coal fire - Proposed a mathematical model of the three-dimensional heterogeneous and anisotropic void rate distribution of mining-induced voids by establishing a series of distribution equations of the void rate in the disturbed overburden.		
Gao *et al.*(2014)	*Tunnel*	CFD	Hybrid ventilation
	Remark: An analysis of Carbone monoxide distribution in large tunnel fires that to estimate the horizontal distribution of CO by effect of heat release rate (HRR) of fire and tunnel's aspect ratio (AR) on CO and temperature stratification in a large tunnel fire.		
Tang *et al.*(2017)	*Tunnel*	CFD	Longitudinal ventilation
	Remark: Effect of blockage-heat source distance on highest temperature of buoyancy-induced smoke flow under ceiling in a longitudinal ventilated tunnel- investigated the effect of blockage-heat source distance on the highest gas temperature under the ceiling.		
Tilley *et al.*(2011)	*Tunnel*	CFD	Longitudinal ventilation
	Remark: CFD simulations in small-scale tunnel and atrium fire configurations - Both cases concerned the formation of a quasi-steady-state smoke layer and smoke movement.		
Huang *et al.*(2009)	*Building*	CFD&GA	Natural ventilation
	Remark: Optimum design for smoke-control system in buildings considering robustness using CFD and Genetic Algorithms - the optimum design method		

	considering the robustness of smoke-control systems in buildings is developed using a coupled approach combining CFD and GA.		
Deckers *et al*.(2013)	*Car Park*	CFD	*Mechanical Ventilation*
	Remark: Smoke management in case of fire in a large car park. As long as the flow is unidirectional, reduced smoke extraction rates lead to more smoke back-layering and increased fire HRR and consider the smoke extraction rate effect being much stronger than the fire HRR.		
Węgrzyński, W., & Krajewski, G.(2017)	*Building*	*CFD*	Natural ventilation
	Remark: Influence of wind on natural smoke and heat exhaust system performance in fire conditions that shows that the local performance of ventilators differs, depending on their location within the building.		
Weng *et al*.(2014)	*Tunnel*	*CFD*	Natural & *Mechanical*
	Remark: Full-scale experiment and CFD simulation on smoke control and smoke movement in a metro tunnel with one opening portal.		
Wang, F., & Wang, M.(2016)	*Tunnel*	*CFD*	Longitudinal ventilation
	Remark: the effects of fire location on smoke movement in a road tunnel with a numerical simulation is carried out the effects of cross-sectional fire locations on the critical velocity and the smoke flow characteristic.		
Gao *et al*.(2016)	*Building*	*CFD*	Hybrid ventilation
	Remark: A CFD study on Fire-induced smoke management via hybrid ventilation in subway station- Hybrid ventilation in a subway station is studied with the dispersion of fire-induced buoyancy driven smoke and Four different grid systems are compared and the sensitivity study of those grid systems is performed.		

This study discusses a case study undertaken on smoke propagation and air flow pattern during fire condition. This report focuses on Computational Fluid Dynamics (CFD) simulation on the smoke spill at Velodrome. There are the different techniques for smoke control depending on the type of fire ventilation system to be followed at sub-sections:

2.2.1 Common Fire Ventilation Methods in Tunnels

Tunnels may require ventilation for different reasons. The reasons can be for example, to ensure a sufficient air quality, to manage the spread of smoke in case of fire or to decrease temperatures to passable limits. Vehicular tunnels e.g. rail, road and metro usually require high air quality during smoke control and normal operation in case of fire, while cable tunnels require smoke control, cooling and a certain amount of air exchange. Station tunnels and mine tunnels also require sufficient ventilation for physiological, smoke control and cooling requirements. Ventilation is essential in most road tunnels to limit the concentrations of contaminants to passable levels in tunnel. Ventilation systems are also used to manage heated gases and smoke that are produced during a tunnel fire emergency. Some short tunnels are ventilated naturally without applying fans, however, such tunnels could necessitate a ventilation system to combat a fire emergency (NFPA, 2015). Common ventilation system in tunnels are transverse, semi-transverse and longitudinal as described as below:

2.2.1.1 Longitudinal Ventilation System

Longitudinal ventilation system can be installed in much longer tunnels, Depending on the maintenance and fire risk mitigation of sufficient air quality. For short tunnels that are three kilometers or less in length, longitudinal ventilation system is usually applied due to lowest construction cost. Exerting smoke management using longitudinal ventilation involves extraction smoke through a predetermined route downstream of the fire and preventing the smoke from spreading upstream. Thus, the minimum air speed is required to prevent smoke from spreading against the longitudinal ventilation flow. This is important in the longitudinal ventilation mode. The critical speed should be guaranteed to prevent smoke back-layering. Moreover, a suitable airflow velocity should be produced to prevent smoke from infiltrating the branches that are used for pedestrian evacuation, especially for the tunnel branches that are downstream of the fire and adjacent to the smoke evacuation route (Du *et al.*, 2016). Longitudinal ventilation in its simplest form consists of fresh air introduced within the entry point and exhaust air expelled out of the exit portal. The pollution level increases along the tunnel because this is the direction of air flow, and vehicles continue to generate emissions as they pass from one end to the other. In reality, tunnels in urban areas of Australia are normally graded downhill at the start of the tunnel and then uphill toward the exit, as they generally pass through relatively flat terrain. The relatively high engine load on the uphill section tends to result in higher exhaust emissions near the end of the tunnel. The design of a longitudinal ventilation system is dictated by the allowable pollution limit inside the tunnel (Beard, A. N., 2016).

The way this is controlled is by ensuring that the volume of fresh air coming into the tunnel at the entry portal adequately dilutes the pollutants. This air volume can be induced by the vehicles, and is sometimes referred to as the ‘piston effect’. For longer tunnels the air flow can be supplemented by ventilation fans in cases when the traffic speed is inadequate to generate sufficient portal inflow to keep pollutant levels below the allowable limit (Yi *et al*., 2015 and Beard, A. N., 2016).

2.2.1.2 Transverse or semi transverse ventilation system

Transverse ventilation systems employ remote-controlled dampers to extract the smoke close to the fire location. In these systems, the location of evacuating the concentrated smoke will be limited to the location of the smoke source. The effectiveness of these systems in control of air/smoke flow depends on confining smoke within a short region and on the capacity of smoke extraction. Monitoring of air or smoke movement in controlled operation of transverse ventilation related on correct air or smoke velocity readings, i.e. location of the sensors and their validity in fire situations (Li *et al*., 2012 and Zhang *et al*., 2016).

2.2.1.3 Natural Ventilation Systems

In this type of tunnel, the smoke produced by a fire was aerated through the openings in the ceiling, providing a natural buoyancy of hot smoke. The distance from the fire to the incline position of the spreading smoke and the thickness of smoke layers along the ceiling were investigated by changing the heat release rate and using two types of median structures experimental parameters. It was clarified that the smoke spreading distance was constant and independent of the heat release rate of the fire under the experimental conditions. Moreover, it was confirmed that the thickness of the smoke layers in the tunnel thinned out quickly due to the natural ventilation (Yao *et al*., 2016).

2.2.2 Common Fire Ventilation Methods in Buildings

Ventilation systems move outdoor air into the buildings, and distribute the air within them. The building ventilation systems generate clean air for breathing by diluting the pollutants originating in the buildings and eliminating the contaminant from them. There are three basic elements in building ventilation systems:

- Airflow path: the overall airflow path in a building which should be from clean zones to dirty zones;

- Ventilation rate: the quantity of outdoor air provided into the space and the quality of outdoor air

- Air distribution or Airflow pattern: the external air should be delivered to each part of the space in an efficient method and the airborne contaminants provided in each part of the space should be eliminated in an efficient method.

Three ventilation system is applied in buildings: natural, mechanical and hybrid (mixed-mode) ventilation as described here:

2.2.2.1 Mechanical Ventilation System

Mechanical ventilation systems are driven by Mechanical fans. Fans can be installed in windows or walls, or exhausting air from a room or installed in air ducts for supplying air. The kind of mechanical ventilation systems depends on climate. For example, a positive pressure mechanical ventilation system is used, in warm and humid climates. In these climates, infiltration needs to be prevented to decrease the interstitial condensation. Conversely, in cold climates, negative pressure ventilation is used as exfiltration is prevented to reduce interstitial condensation. For a room with locally generated pollutants, such as a kitchen, toilet or bathroom, the negative pressure system is often used (Zhang *et al*., 2016).

Ventilation systems are used in air-conditioned residential buildings in order to keep an acceptable indoor air quality. Based on results of evaluating various ventilation strategies, it was concluded that short-term mechanical ventilation is the most appropriate ventilation strategy for air-conditioned residential buildings. However, there is still no a general design framework of short-term mechanical ventilation strategy for determining the appropriate design parameters, including ventilation frequency, start concentration of ventilation and ventilation period based on various combinations of indoor CO2 generation rate, infiltration rate, net room volume, and mechanical ventilation rate (Ai, Z. T., & Mak, C. M., 2016).

2.2.2.2 Natural Ventilation System

An effective strategy for reducing the use of energy in buildings is natural ventilation system. The effect of natural ventilation system is significant for buildings with high internal heat generation, such as commercial office buildings. This is because naturally ventilated buildings are becoming increasingly popular in Japan. According to review paper done by Nomura, M., & Hiyama, K. (2017). The design of naturally ventilated buildings were analyzed to compare the representative air change rates. The measurement results from studies shows that ventilation performance depends highly on the design and no strong correlation is found between the air change rates and floor areas. It is noted that, the performance of natural ventilation systems are considerably dependent on the building shape, that during the early stages of building design is generally discussed. It is important to provide a clear target air change rate in range of achievable values for natural ventilation in early design stage and consider this target throughout the building design process. Natural ventilation design for building usage is one of the best strategies for reducing the energy consumption (Nomura, M., & Hiyama, K., 2017).

2.2.2.3 Hybrid Ventilation System

The active ventilation systems in building, which eliminate excess contaminants, heat and humidity from indoor environment, could be large energy consumers. In order to provide desired ventilation flow rates for all of the floors of a multi-story building and reduce the energy consumption is proposed a stack-based hybrid ventilation scheme. The most advantages of this hybrid scheme is when the required ventilation flow rate is beyond the one that pure buoyancy-driven ventilation schemes or the building has many floors. the optimal interface between the MVFs (mechanically ventilated floors), NVFs (naturally ventilated floors) and the vent sizes of different NVFs which guarantee an balance between the desired ventilation flow rate, room air temperature, and the heat inputs within the occupants' spaces, are derived. There are the differences between the applicability of this hybrid ventilation scheme and the other two low-energy ventilation schemes. The design procedure is presented for stack-based hybrid ventilation scheme (Yang, D., & Li, P., 2015). These hybrid ventilation systems adjusting the use of each system based on the time of day or season of the year have drawn worldwide attention. Hybrid ventilation technology provides sustainable development and energy saving and fulfils high requirements for indoor environmental performance by optimizing the balance between energy use, indoor air quality, environmental impact and thermal comfort. (Lim *et al*., 2015).

2.2.3 Common Fire Ventilation Methods in Car park

Underground car parks are common in urban or densely populated areas. These car parks can be associated with being exposed to risks such as fire and explosions. As such, fire safety is an important issue in managing underground car parks. Studies related to this issue such as those conducted on car park ventilation systems and available statistics on heat release rate from recent car fire experiments with modern cars and various setups show that fires in car parks should be a cause of concern even though car fires usually do not spread and therefore, there are less injuries and few deaths. However, a fire that consumes cars can bring detrimental effects to car owners and substantial structural damage can result in cases in which fire spreads between vehicles. The full-scale experiments on new cars have showed high fire HRR amounts which exceeds 16 MW when three cars were on fire. The constant fire spread between cars and high heat release rates were due to the severe heat transferred to the neighboring cars. However, there were a number of fires in various car parks in countries where these situations have been applied and the fire has extend to a large number of cars. Ventilation systems effect in large car parks causes a decrease of the temperatures and thus, in order for a slower fire spread from the initial burning car to the neighboring cars, the air flow must reach the fire source. Placing the position of fire in a recirculation zone shows that air flow will basically by pass it and effect of the ventilation will be very limited. In addition, fire sources near a wall provide a more challenging condition for heat control (SHC) system and smoke. This is because the fire-induced flows are stronger and the fire development is faster. These affects the forced ventilation in which the air flow can reach the fire source and the air flow momentum can be strong enough to defeat the flow resistance provided by the fire-induced smoke flow (Węgrzyński, W., & Krajewski, G., 2017 and Kashef *et al*., 2013).

2.2.3.1 Jet-Fan - Based Ventilation System

Jet fan ventilation systems are preferred over traditional ducted systems as ventilating pollutants from large spaces such as car parks. This ventilation system induces additional airflow within the environment by producing a high discharge thrust and velocity using the axial fans located at the ceiling of environment. Smoke and heat will be discharged from exiting portal within environment. It is very important to consider the selection and situation of jet fans inside environment for controlling the smoke/air velocity and avoiding smoke penetration through open cross-passage doors. Jet fan ventilation systems induce the turbulence in air and smoke movement. Thus, installed fans destroy the existing smoke layer within the smoke filled zone. Jet fan ventilation systems activate upstream fans by activation of fans downstream of the fire location (Kashef *et al*., 2013). These ventilation systems provide a low pressure region downstream and an overpressure upstream of the fire.

2.2.3.2 Horizontal mechanical Ventilation System

A method for controlling the heat and smoke generated by the fire source be called Horizontal mechanical ventilation. One selection is that use the ductwork to 'trap' the smoke and eliminate heat and smoke through the ductwork. Heat and smoke generated by fire is removed and the risk of fire spread is reduced. Another selection in horizontal mechanical ventilation systems is that use the natural vertical venting with aim at a guaranteed smoke-free height. However, this system is not very common due to the usually low ceiling height in car parks (Deckers *et al*., 2012).

2.2.3.3 Natural Ventilation System

Natural ventilation system is the preferred method of ventilation systems within car parks and these systems require openings to fresh air being provided to equal a percentage of the floor area of the car park.

2.2.4 Common Fire Ventilation Methods in Mine

The main ventilation circuit in mine is flow-through (mine) ventilation. The air is distributed through the mine from surface via shaft, internal ventilation raises and ramps. Then, flows are controlled by regulators and permanently mounted ventilation fans. The auxiliary ventilation systems take air from the flow-through system and distribute it to the mine workings via temporarily mounted ventilation fans, and disposable steel ducting. Duct systems and auxiliary fan may be either exhaust systems that draw out contaminated air or forcing systems, where fresh air is pushed into mine headings (Fan *et al*., 2013).

A basic component in the design of subsurface facility such as underground mine is the quantified planning of the distribution of airflows, with the location of fans and their duties and other ventilation controls required to get acceptable environmental conditions throughout the system. It is essential to plan ahead throughout the life of an underground operation in order that fans, new shafts or other airways are available in a timely manner for effective ventilation of extensions to the workings. Ventilation planning should be a continuous and routine process, as any operating mine is a dynamic system with new workings continually being generated and older ones coming to the end of their productive life. Analysis of Ventilation network is related with the interactive characteristic of air flows within pipe or duct and the linked branches of an integrated and complete network. (Mei *et al*., 2016).

2.3 Concluding Remarks

This chapter reviewed the various types of ventilation systems and the existing common fire ventilation methods. It discussed the basic ideas behind ventilation systems with a centralize on the methodologies as well as on the requirements for systems of fire ventilation in various environmental for example mine, car park, tunnel, and building. As result, one of the objectives of fire engineering design is life safety in the case of tunnel, buildings, car park, Velodrome fires or other area. The fire events and its effects to the fire environments should be controlled, usually by ventilation systems. These systems play a key role in human safety and provide tenable environment. Ventilation system should provide passable air quality for the safe passage of users in order to simplify rescue situations during fire event. Next chapter (Chapter 3) will discuss the research methods involved in the acquisition of the objectives and completion the research successfully.

CHAPTER 3: RESEARCH METHODOLOGY

This chapter provides the research plan of this study and describes the activities necessary for the completion of the research. The research methodology is foundation as it layout approaches and measurement that make sure that the research will handle appropriately. This chapter describes the fire simulation and smoke spread using the CFD model in a Velodrome environment.

3.1 Introduction

In order to optimize a smoke-control system, a CFD simulation for modeling requires multiple input parameters, a geometric setup and related physical models representing physical phenomena that will be used with focus on the safety issue. The simulation of the smoke spread using CFD in case of fire in Velodrome will be done due to human safety (see Figure 3.1). Therefore, the obtaining trust in CFD and boundary conditions and grid size are important. CFD models usually require large capacity computer workstations or mainframe computers. In CFD models, the space is divided into many cells and use the governing equations to solve the move of mass and heat between the cells. The governing equations include the equations of conservation of momentum, mass and energy. These partial various equations can be solved numerically by algorithms specifically developed for that purpose. For smoke management applications, the number of cells is generally in the range from tens of thousands to millions. Due of the very large number of cells, CFD models avoid the more generalized engineering equations used in zone models. Through the use of small cells, CFD models can test the situation in much greater detail and account for the impact of irregular unusual air movements and shapes that cannot be addressed by either algebraic equations or zone models. (NFPA, 2015).

Figure 3.1: Example of Fire in Velodrome

3.2 Project Methodology

The main objective of this section is to outline research methodology of this study. It presents and explains the list of steps researcher undertook to carry out this research from data collection through data analysis. Figure 3.2 provides an illustration of steps involved and the connections between them for CFD simulation during a Fire Event in Velodrome environment. The purpose of this section is to explain the methodology we are using for achieving the objectives of our research.

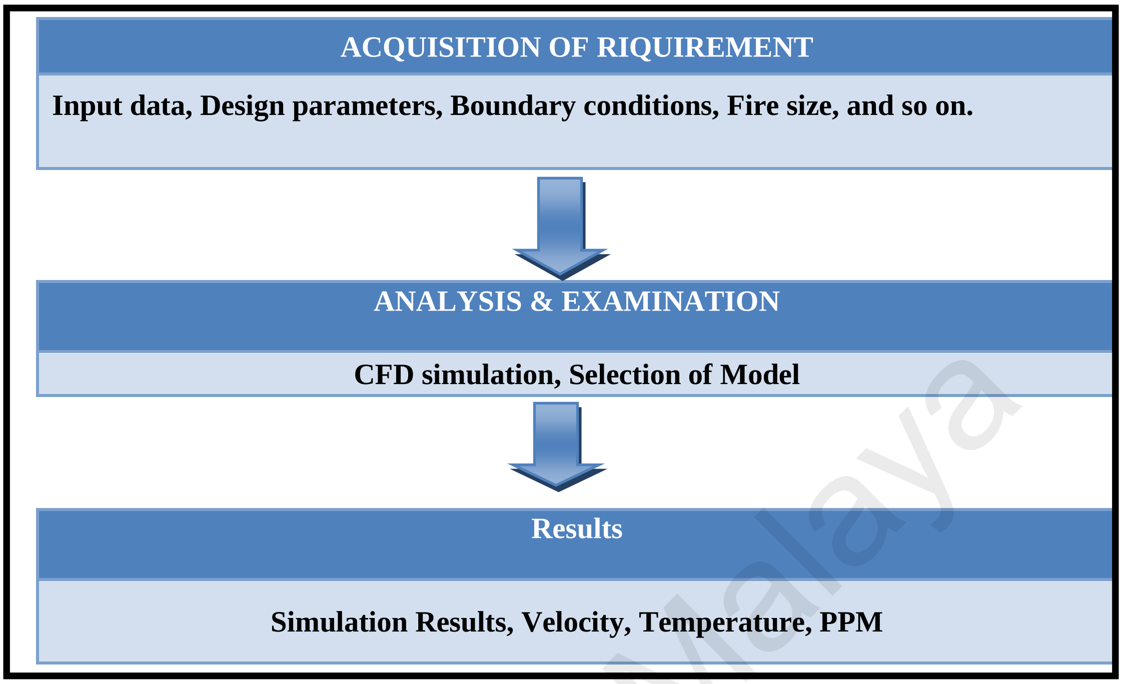


Figure 3.2: Methodology procedure framework

Parts per Million (PPM) measure the level of Carbon Monoxide (CO) concentration. For example, 100 PPM CO means that for every 999,900 molecules of air, there are 100 molecules of CO. Carbon Monoxide effects differently depending on the concentration (NFPA, 2015). Table 3.1 summarizes some health effects because to prolonged exposure to different concentrations of CO, as well as Pocket CO alarm levels and some government recommended limits by NFPA, (2015).

Table 3.1: Pocket CO alarm levels (NPFA, 2015)

Level of CO	Health Effects, and Other Information
0 PPM	Normal, fresh air.
9 PPM	Maximum recommended indoor CO level
10-24 PPM	Possible health effects with long-term exposure.
25 PPM	Max TWA (time-weighted average) Exposure for 8 hour work-day **Pocket CO TWA warning sounds each hour.**
50 PPM	Maximum permissible exposure in workplace. **First Pocket CO ALARM starts (optional, every 20 seconds).**
100 PPM	Slight headache after 1-2 hours.
125 PPM	**Second Pocket CO ALARM starts (every 10 seconds).**
200 PPM	Dizziness, headache after 2-3 hours of exposure.
400 PPM	Headache and nausea after 1-2 hours of exposure. Life threatening in 3 hours. **Third Pocket CO ALARM starts (every 5 seconds).**
800 PPM	Headache, nausea, and dizziness after 45 minutes; collapse and unconsciousness after 1 hour of exposure. Death within 2-3 hours.
1000 PPM	Loss of consciousness after 1 hour of exposure.
1600 PPM	Headache, nausea, and dizziness after 20 minutes of exposure. Death within 1-2 hours.
3200 PPM	Headache, nausea, and dizziness after 5-10 minutes; collapse and unconsciousness after 30 minutes of exposure. Death within 1 hour.
6400 PPM	Death within 30 minutes.
12,800 PPM	Immediate physiological effects, unconsciousness. Death within 1-3 minutes of exposure.

3.2.1 Fundamentals of Computational Fluid Dynamics (CFD)

CFD as a simulation tool has been used for modeling fluid-flow problems and solving the governing flow equations. CFD is an analysis tool and sophisticated design that apply the modern computation power of computers to simulate heating (chilling, sterilization, cooking), fluid flow, mass transfer (dissolution or transpiration), phase change (melting, boiling, freezing), mechanical movement (impellers, fans, pistons, or rudders), stress or deformation of related structures, chemical reactions (combustion or rusting), and interactions between solids and fluids. The accuracy of CFD simulations and their reliability are being constantly improved by considering the rapid development of computing power and commercial CFD packages. This numerical CFD simulation has been widely used in more researches for fire smoke simulation. However, it is noted that considering the accuracy of such simulations is needed for making traditional measurements, which is impossible without disturbing the packaging arrangement (Zhao *et al.*, 2016).

CFD models are utilized in the Fire Protection Engineering field. The tool applies the fundamental laws of physics to offer a versatile approach to solving the challenges of fire dynamics. CFD is used by engineers and scientists in a wide range of fields. Typical applications include:

- **Process industry**: chemical reactors, Mixing vessels
- **Building services**: Ventilation of buildings, such as atriums
- **Health and safety**: Investigation on the effects of smoke and fire
- **Motor industry**: car aerodynamics, Combustion modeling
- **Electronics**: Heat transfer within and around circuit boards
- **Environmental**: Dispersion of pollutants in water OR air
- **Power and energy**: Optimization of combustion processes
- **Medical**: Blood flow through grafted blood vessels

3.2.2 Numerical simulation set-up

A schematic presentation of set-up for simulations set-up has been shown at Figure 3.3. As seen at Figure 3.3, we have 4 point that fresh air jet fan and also fresh air doors come from 4 side of Velodrome. We have installed exhaust systems as longitudinal in Velodrome in 4 zone. Fire cases were created in two positions for simulating behavior of smoke propagation in various times.

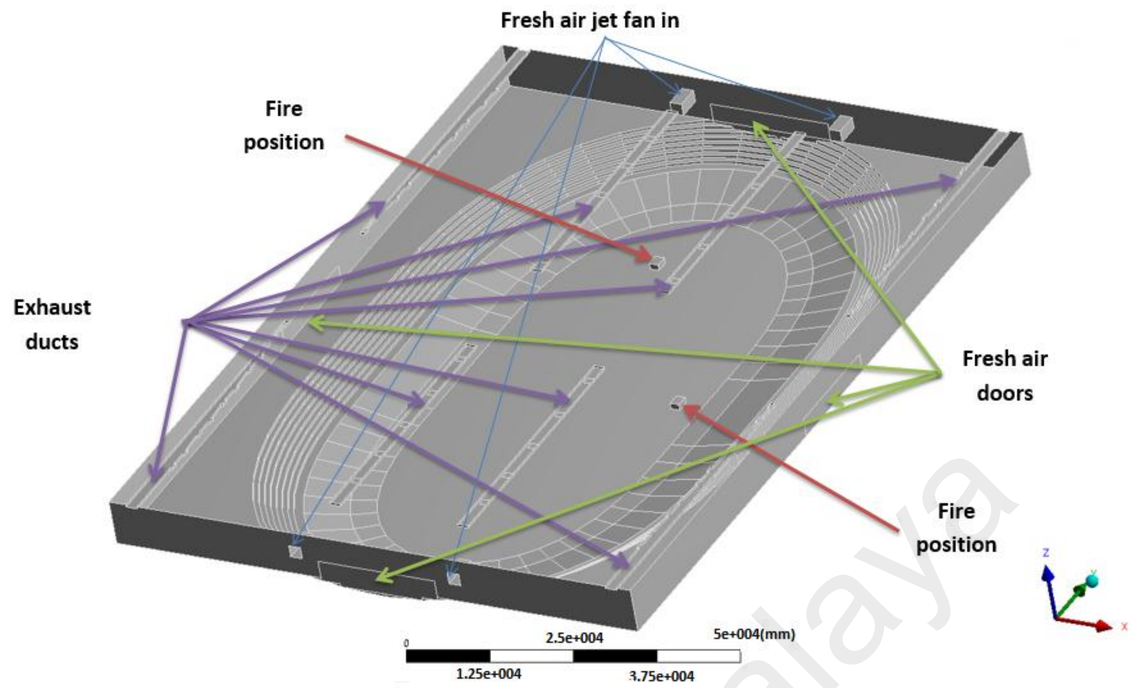


Figure 3.3 Schematic presentation of Velodrome as modeled in the simulations

3.2.3 Geometry of the Computational Model

Figure 3.4 shows a geometric view of the Velodrome. The computational model for an air circulation system design with dimensions 85.5m (width) _124m (length) _13.5m (height) was created based on the size of a representative single shelf from a commercial scale indoor Velodrome building. Volume properties of model is 1.2327e+014 mm³ and analysis type of model is 3 Dimensional.

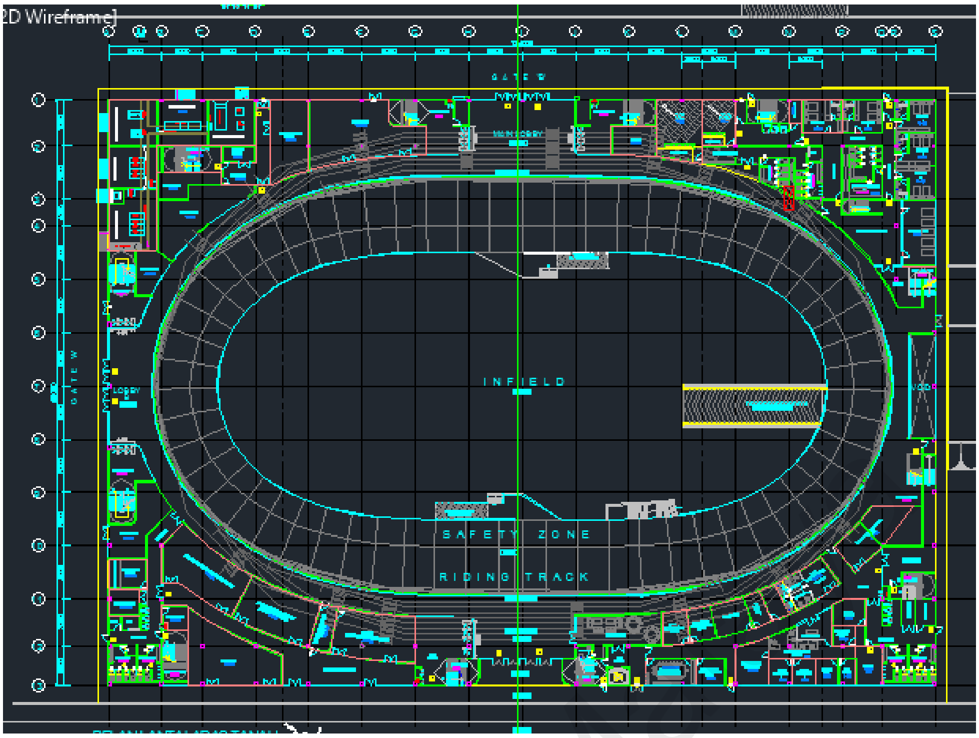


Figure 3.4: CAD GEOMETRY

Figure 3.5 shows four heights for velodrome that we will consider in our simulation that minimum height is floor and maximum height is 13.5 meter.

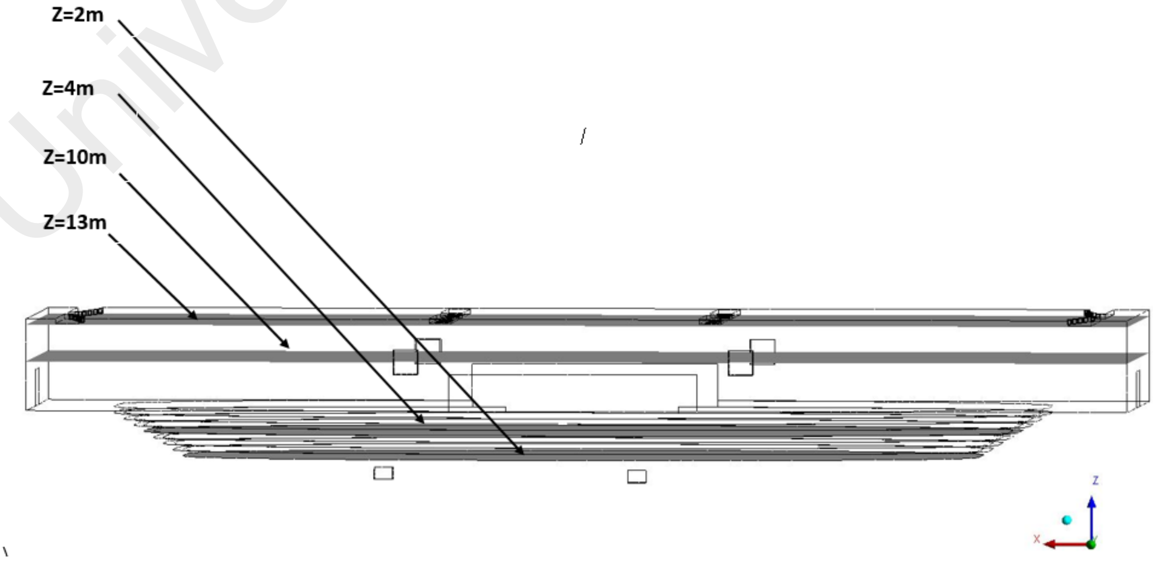


Figure 3.5: Cut planes (Z=0 IS FLOOR AND Z= 13.5M IS CEILING)

3.2.4 Mesh Generation

The Scaling (Grid Independence) is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid convergence. The method we carry out a mesh independence study described as below: (Wang, F., & Wang, M., 2016).

Step 1: Run the simulation on initial mesh and ensure convergence of residual error to 10-4, monitor points are steady if not, correct the mesh and repeat.

Step 2: Once we have gain the convergence criteria above for first simulation, correct the mesh globally so that you have finer cells throughout the domain. Usually we would aim for around 1.5 times the initial mesh size. Run the simulation and ensure that the residual error drops below 10-4, that the monitor points are steady. At this point we need to compare the monitor point amount from Step 2 against the amount from Step 1. If the amount at Step 2 is not within passable amount of the Step 1 result, then this means that our solution is changing because of our mesh resolution, and hence the solution is not yet independent of the mesh. In this case you will need to move to Step 3.

Step 3: Due to your solution is changing with the filtration of mesh, we have not yet achieved a mesh independent solution. We need to correct the mesh more, and repeat the process until you have a solution that is independent of the mesh. We should then always use the smallest mesh that gives you this mesh independent solution (to reduce your simulation run time).

To ensure that numerical solutions are independent with selected mesh size, simulations are performed with three different grid sizes. As we get three sizes for fine, coarse and medium grid. The Table 3.2 and table 3.3 depicted parameters of mesh that selected in CFD software for simulation of Velodrome. Total number of the elements in mesh generation is reached up to four million.

Table 3.2: Parametric View of Mesh Generation

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default(22.5910 mm)
Max Face Size	2000.0 mm
Max Size	2000.0 mm
Growth Rate	1.180
Minimum Edge Length	94.850 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5

Growth Rate	1.2
Table 3.2 continued	
Inflation Algorithm	Pre
View Advanced Options	No
Assembly Meshing	
Method	None
Patch Conforming Options	
Triangle Surface Masher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Shape Checking	CFD
Element Midsize Nodes	Dropped
Straight Sided Elements	
Number of Retries	0
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Disfeaturing	
Pinch Tolerance	Default (20.3320 mm)
Generate Pinch on Refresh	No
Automatic Mesh Based Disfeaturing	On
Disfeaturing Tolerance	Default (11.2960 mm)
Statistics	
Nodes	929922
Elements	3983048
Mesh Metric	None

Table 3.3: Parametric view mesh controls

Object Name	*Face Sizing*	*Face Sizing 2*	*Edge Sizing 2*
State	Fully Defined		
Scope			
Scoping Method	Geometry Selection		
Geometry	208 Faces	14 Faces	1558 Edges
Definition			
Suppressed	No		
Type	Element Size		
Element Size	200. mm	150. mm	500. mm
Behavior	Soft		
Curvature Normal Angle	Default		
Growth Rate	1.170	1.150	1.10
Local Min Size	Default (22.591 mm)		
Bias Type			No Bias

The Figure 3.6 and figure 3.7 showed that 3-D and 2-D View of mesh generation respectively in Velodrome Model. As you can see in these views, mesh cell near fire are very small to create accuracy of model for simulation.

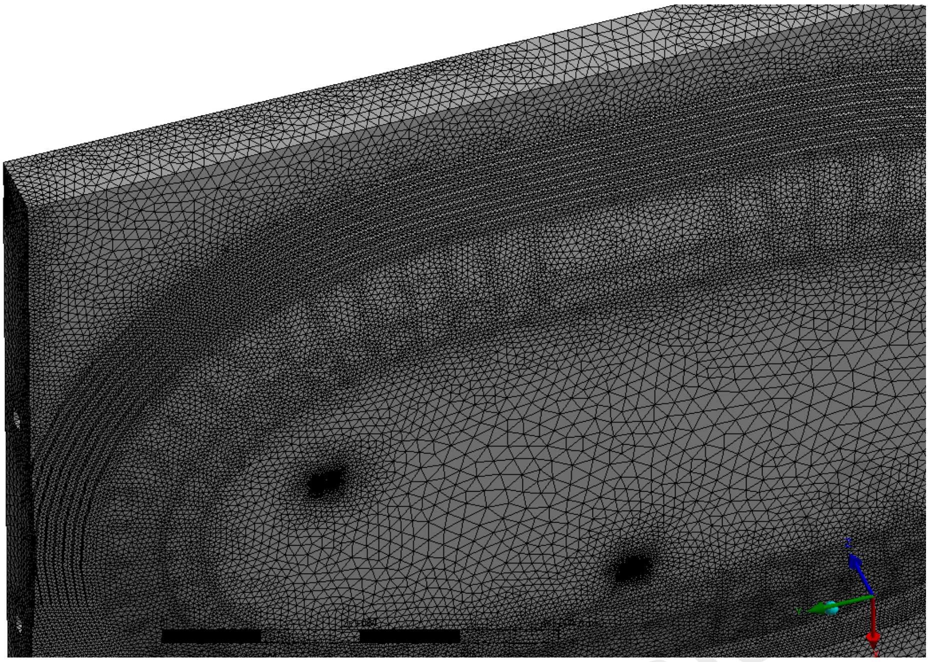


Figure 3.6: 3-D Mesh generations

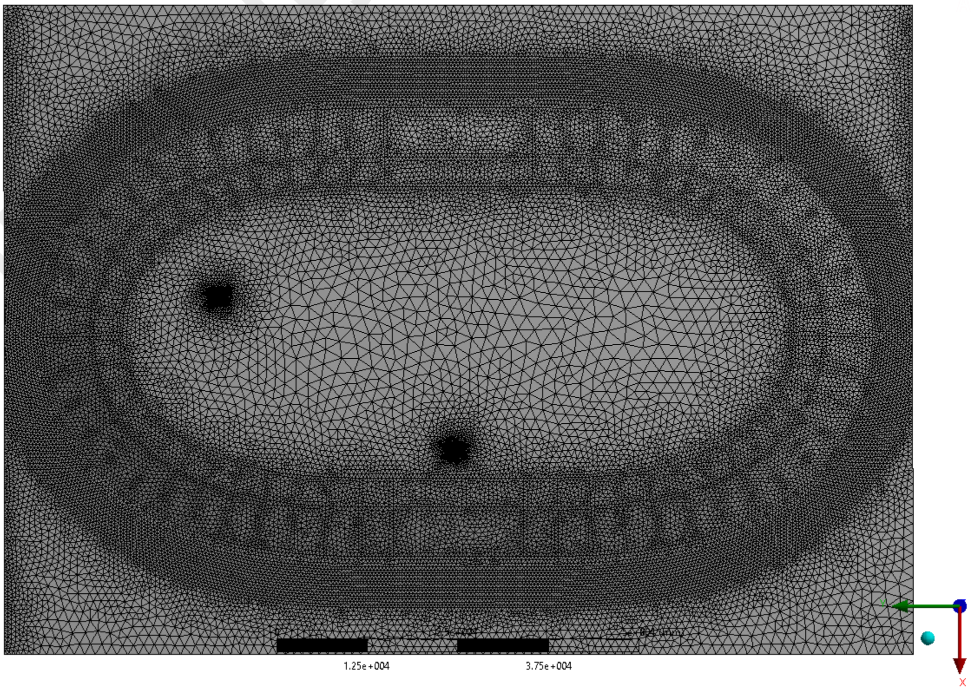


Figure 3.7: 2-D View Mesh generation

3.2.5 Boundary Condition

The most integral section of any Computational fluid dynamics (CFD) problem is the description of its boundary conditions. Therefore, it is required that the user understands and uses the boundary conditions correctly, wisely and effectively and also comprehend its role in the numerical algorithm. If the boundary conditions are not determined correctly, then the solution might result in blunders and if they are not applied wisely, then the problem solving time may increase multiple. Different types of boundary conditions are used in CFD for Various conditions and purposes are as follows:

- Inlet boundary condition
- Outlet boundary condition
- Wall boundary conditions,
- Constant pressure boundary condition

The design fire is assumed to be 4 MW by rectangular duct with 3 m^3 with mass flow rate of 11.8 kg/s and with density1.165kg/m^3 and velocity of fire 3.376m/s. In this model created the model of fire as a fluid zone with heat generation source with amount of 4 MW. The properties of fire fluid as illustrate in table 3.4 in two different positions in Velodrome as we set it accordingly.

Table 3.4: Fire properties

Mass flow rate of smoke (kg/s)	Area of Fire (m^2)	Density (kg/m^3)	Velocity of Fire (m/s)	Smoke temperature (k)
11.8	3	1.165	3.376	629.9

We defined 5 zone of boundary condition as you can see in table 3.5 and named: Outlet, Fire – Inlet 1, Fire - Inlet 2, Fresh-air-doors, fresh-air-in.

Considering to boundary condition, velocity inlet is adopted for fresh air inlets form doors. Outflow was chosen for exhaust ducting system. The wall boundary condition was taken to be adiabatic, and was applied on the solid walls of Velodrome. A velocity boundary condition was prescribed at the Velodrome Inlet. A free pressure outlet boundary condition was used for the Velodrome Outlet. The temperatures of the ambient air were assumed to be 300 °K.

Table 3.5: named selected in simulation according boundary condition

Object Name	*outlet*	*fire-in1*	*fire-in2*	*fresh-air-doors*	*fresh-air-in*
State	Fully Defined				
Scope					
Scoping Method	Geometry Selection				
Geometry	60Faces	1 Face		4 Faces	
Definition					
Send to Solver	Yes				
Visible	Yes				
Program Controlled Inflation	Exclude				
Statistics					
Type	Imported				
Total Selection	60 Faces	1 Face		4 Faces	
Suppressed	0				
Used by Mesh Worksheet	No				

Velodrome consist floors, ceilings, walls, was assumed as the concrete. Four unit of jet fan were defined as showed in figure 3.3 to blow air inside and air velocity was defined at the each fan is 31405 CFM and velocity 3.66m/s and temperature of 300k as depicts in Table 3.6 that conduct throughout duct with Area of 4 m^2 into Velodrome.

Table 3.6: Fresh air intakes

Air flow (cfm)	Velocity (m/s)	Area of Duct (m^2)	Temperature (k)
31405	3.66	4	300

Table 3.7: Fresh air from doors

Velocity (m/s)	Temperature (k)
0.2	300

There is four door can assist to intake fresh Air that assumed velocity of each door 0.2m/s and Temperature of 300K was specified as an air inlet in the natural ventilation situations as shown table 3.7. as you can see in figure 3.8 we assumed eight unit exhausted system to remove smoke from Velodrome with each fan capacity 22175 CFM and extracted smoke throughout Duct as outlet. Besides, the mechanical ventilation included smoke exhaust mode and air supply mode. A rectangular duct protrudes from the wall to create a velocity initiated close to the fire. This duct-mounted makeup air vent serves as the makeup air that will produce the maximum interaction between the flame zone and the fire. The magnitude of the velocity, as well as the location and size of the duct vary in each simulation configuration.

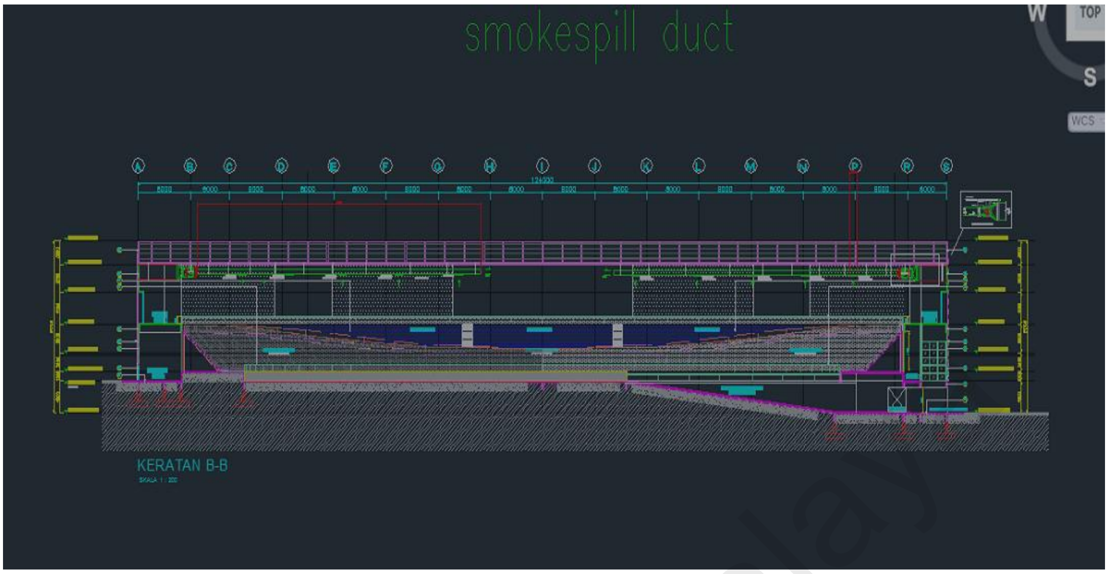


Figure 3.8: Smoke Ducts

Figure 3.9 shows top view of scheme of ducting system in velodrome throughout jetfans that velocity of fan inlet is 31405 CFM and each outlet has 2000 CFM.

CFM stands for cubic feet per minute (airflow). Put simply, CFM is how much air a fan moves. The measurement is taken when the ceiling fan is on its highest speed and uses both the volume of air and the rate at which it moves.

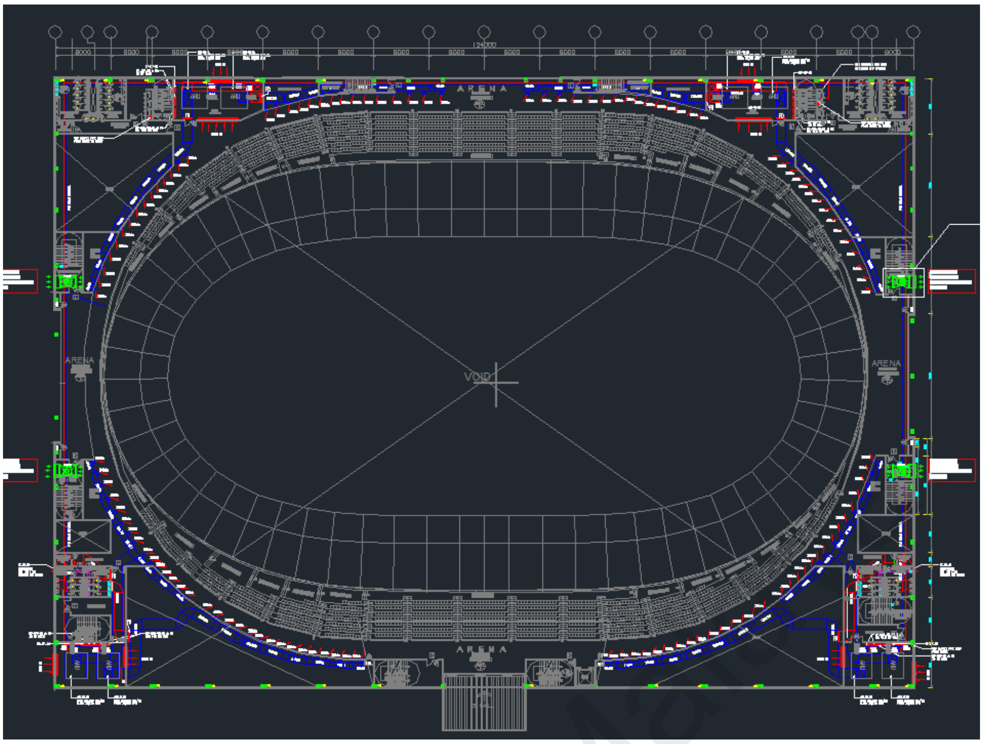


Figure 3.9: Fresh Air Intake

3.2.6 Design methodology and parameters

For this model we have to describe model for simulation and also material we selected. We select turbulent model due to use fan in system according software we have to select viscous model (k- epsilon) a material for this model is Air with density and velocity constant. According our system we have to choose energy equation due to have Temperature in our system. In this model all equipment and parameters consider together to get result.

3.3 Concluding Remarks

The purpose of this chapter was to explain the methodologies we are using for achieving the objectives of our research. Research method involved in this thesis is CFD simulation on the smoke spill at Velodrome. The objective of the investigation is to determined the smoke propagation and air flow pattern during fire condition at Velodrome. CFD modeling methods were employed to simulate the smoke and fire spread in Velodrome case. CFD results depend on mesh selected for the simulation. In this model we define all primary parameter for solving problem and create result according this primary condition. As describes primary condition as below:

- Geometry of Model
- Mesh generation
- Boundary condition

Computational Fluid Dynamics (CFD) simulation results are presented of full-scale Velodrome fire experiments with smoke and heat control (SHC) by ventilation system in next chapter (chapter 4).

CHAPTER 4: RESULTS AND DISCUSIONS

The results are used to determine the impact of exhaust system on the rate of smoke production and consequent smoke layer position. Further the results are used to develop an engineering tool that assists in accounting for the impact of smoke management system. NFPA, (2015) describes a smoke control system as ‘‘an engineering system that contain all methods that can be used separately or in combination to modify smoke movement. “Effective smoke control system or ventilation systems during fire events is very important for saving lives. Since that bifurcation flow of smoke can reduce smoke propagation rates and give more time for people to escape. In this chapter, the results of CFD simulation are presented smoke, velocity and temperature by applying exhaust ventilation and jet fans in case of a Velodrome fire System Design Overview.

4.1 CFD modeling and Analysis

CFD simulations are analyzed the effects of the duct mounted ventilation on the fire. This analysis concentrates on the effect of the increased air velocity on the mass flow rate of the fire as well as the smoke layer interface height within the Velodrome. The computational area includes a generic Velodrome segment with a dimension of 85.5m (width) _124m (length) and 13.5m (height) with Total number of the elements in mesh generation is reached up to four million. Boundary conditions are assumed for the four side of the Velodrome segment that was modeled. Total effective extraction rate achieved at the fire location was calculated based on the velocity and the downstream air flow as well. Four unit of jet fan were defined to blow air inside and air velocity was defined at each fan to be 31405 CFM, with the velocity of 3.66m/s and temperature of 300^{o}k that is conducted throughout duct into the Velodrome. There are four door that assist to intake fresh Air with assumed velocity of each door to be 0.2m/s and temperature of 300^{o}K was described as the air velocity inlet of the natural ventilation system. Eight unit exhaust system is assumed to remove smoke from Velodrome with the fan capacity of 22175 CFM to extract the smoke throughout the duct as outlet. Besides, the mechanical ventilation included for smoke exhaust mode and air supply mode. This model is analyzed in three different modes of placing fire far from exhaust ducts (Case A), Placing fire between the two exhaust ducts (Case B) and placing a Jet fan to expel the smoke from Velodrome (Case C) that all three models is being analyzed below:

4.2 Case A

Base on Case A that position of fire is located in far from exhaust system ducts in ceiling and Center of Velodrome that smoke propagation, velocity and Temperature are analyzed according to exhaust system. The results of this case study are graphically expressed in Figure 4.1 to Figure 4.14.

4.2.1 Smoke Analysis of Case A

Figure 4.1 to Figure 4.5 are 3D views of the smoke pattern and propagation in different times 0-90 minutes when exhaust system turned “OFF” (a), and exhaust systems turned “ON” (b).

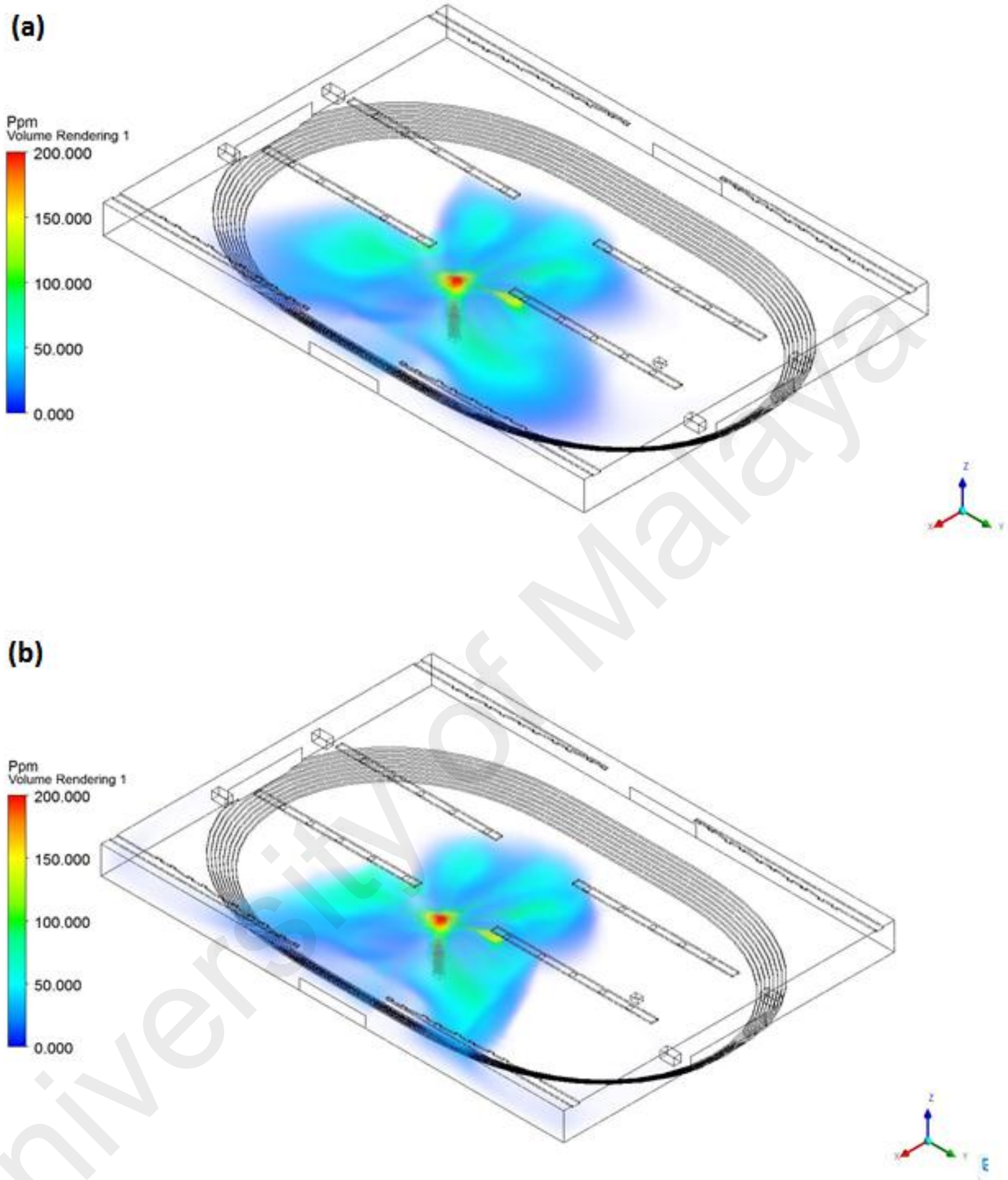


Figure 4.1: 3-D view of the smoke propagation in PPM, in t=10min, Exhaust system "OFF" (a), and Exhaust system "ON" (b)

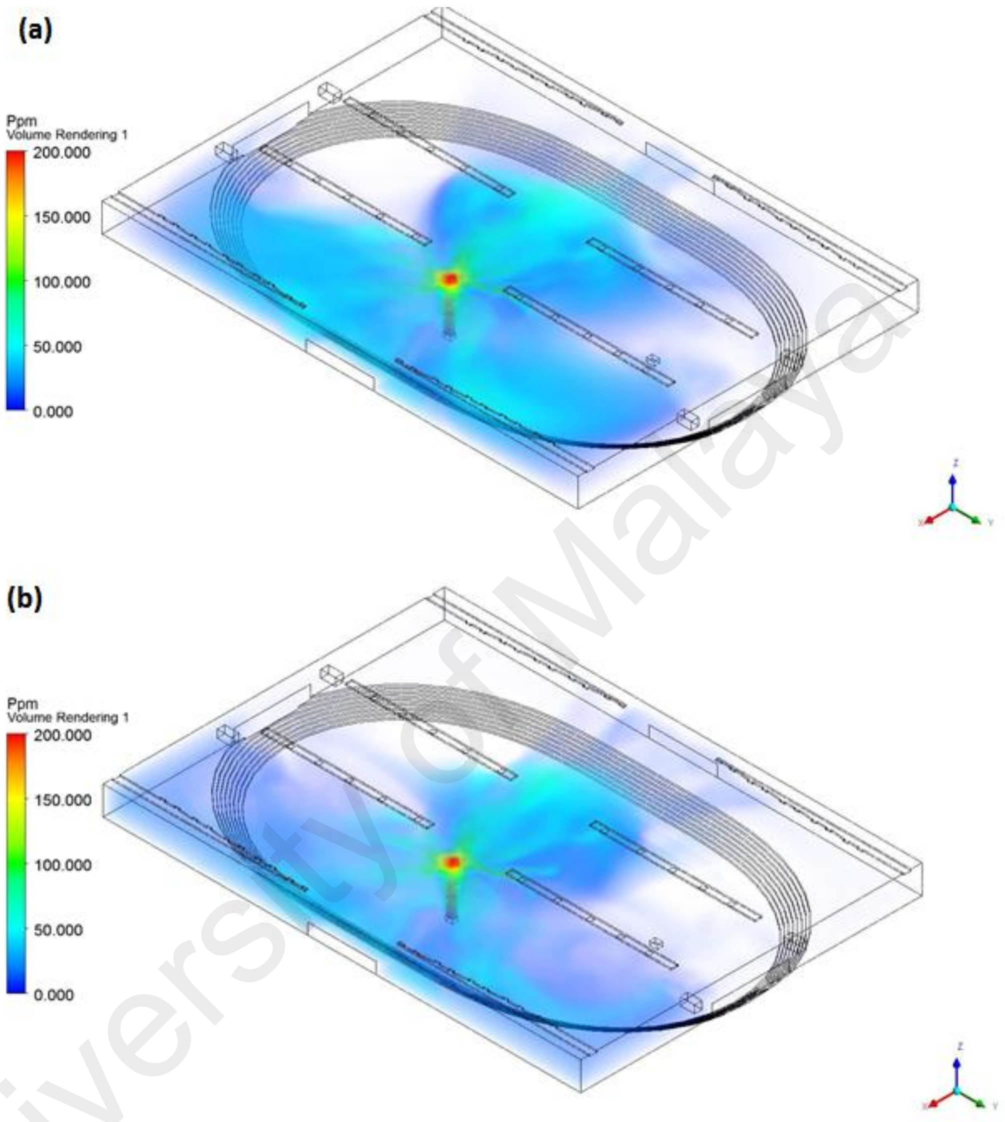


Figure 4.2: 3-D view of the smoke propagation in PPM, in t=20min, Exhaust system "OFF" (a), and Exhaust system "ON" (b)

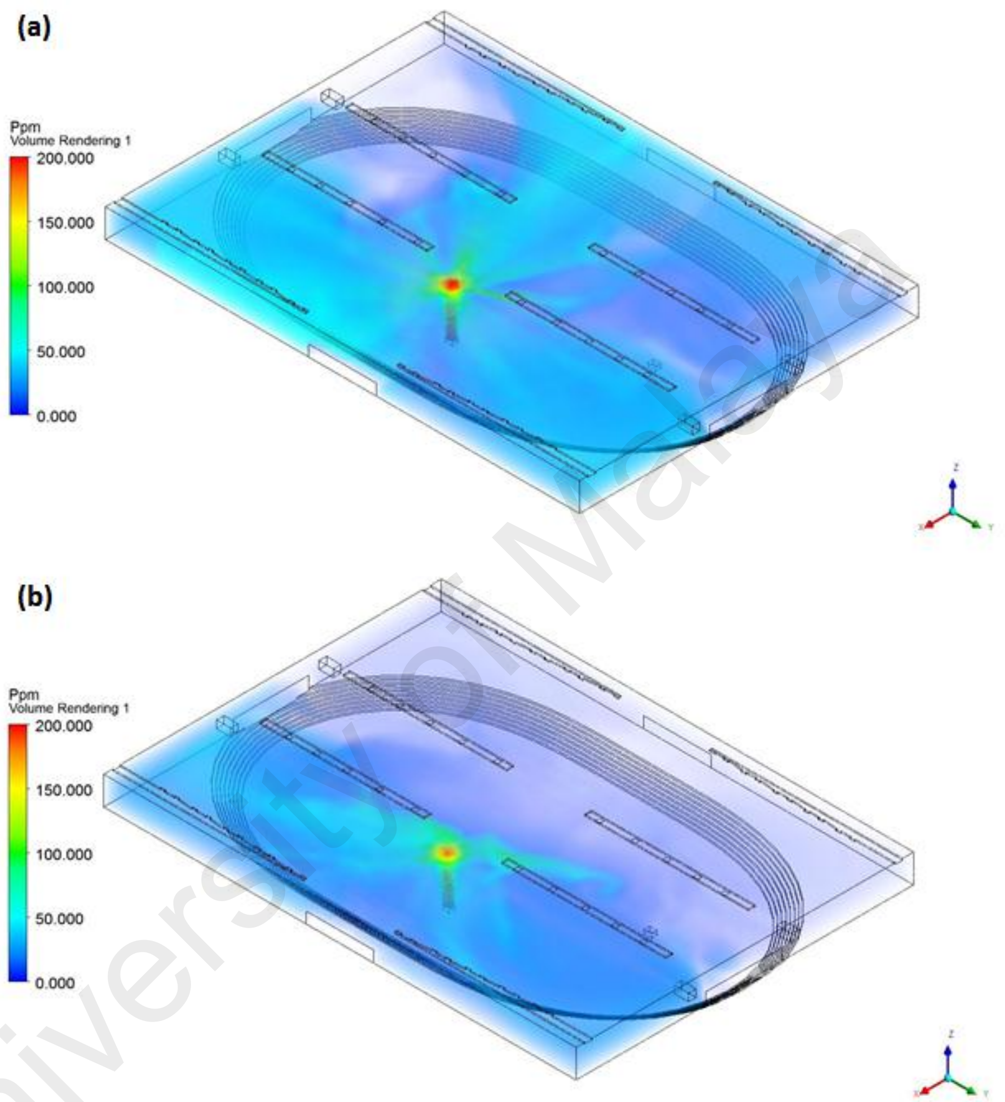


Figure 4.3: 3-D view of the smoke propagation in PPM, in t=50min, Exhaust system "OFF" (a), and Exhaust system "ON" (b)

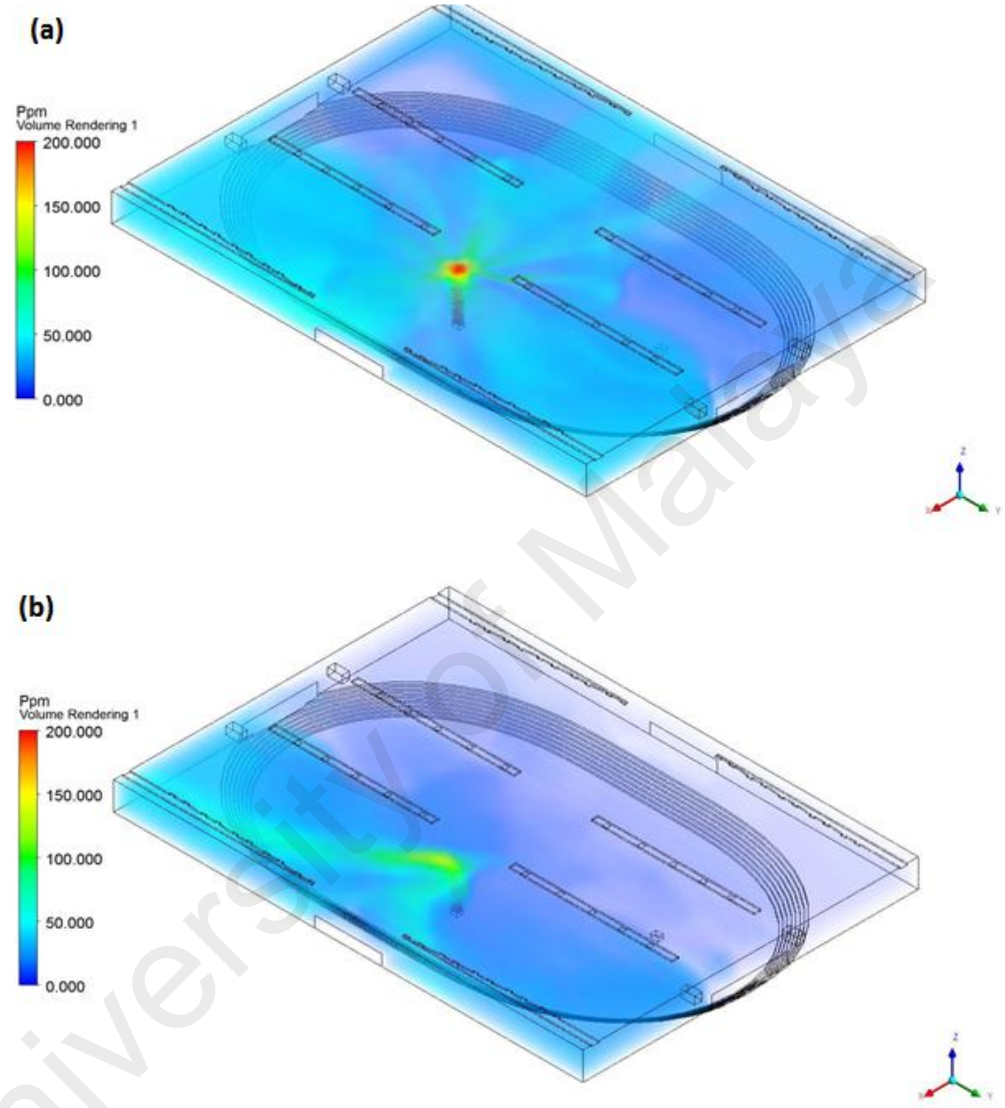


Figure 4.4: 3-D view of the smoke propagation in PPM, in t=70min, Exhaust system "OFF" (a), and Exhaust system "ON" (b)

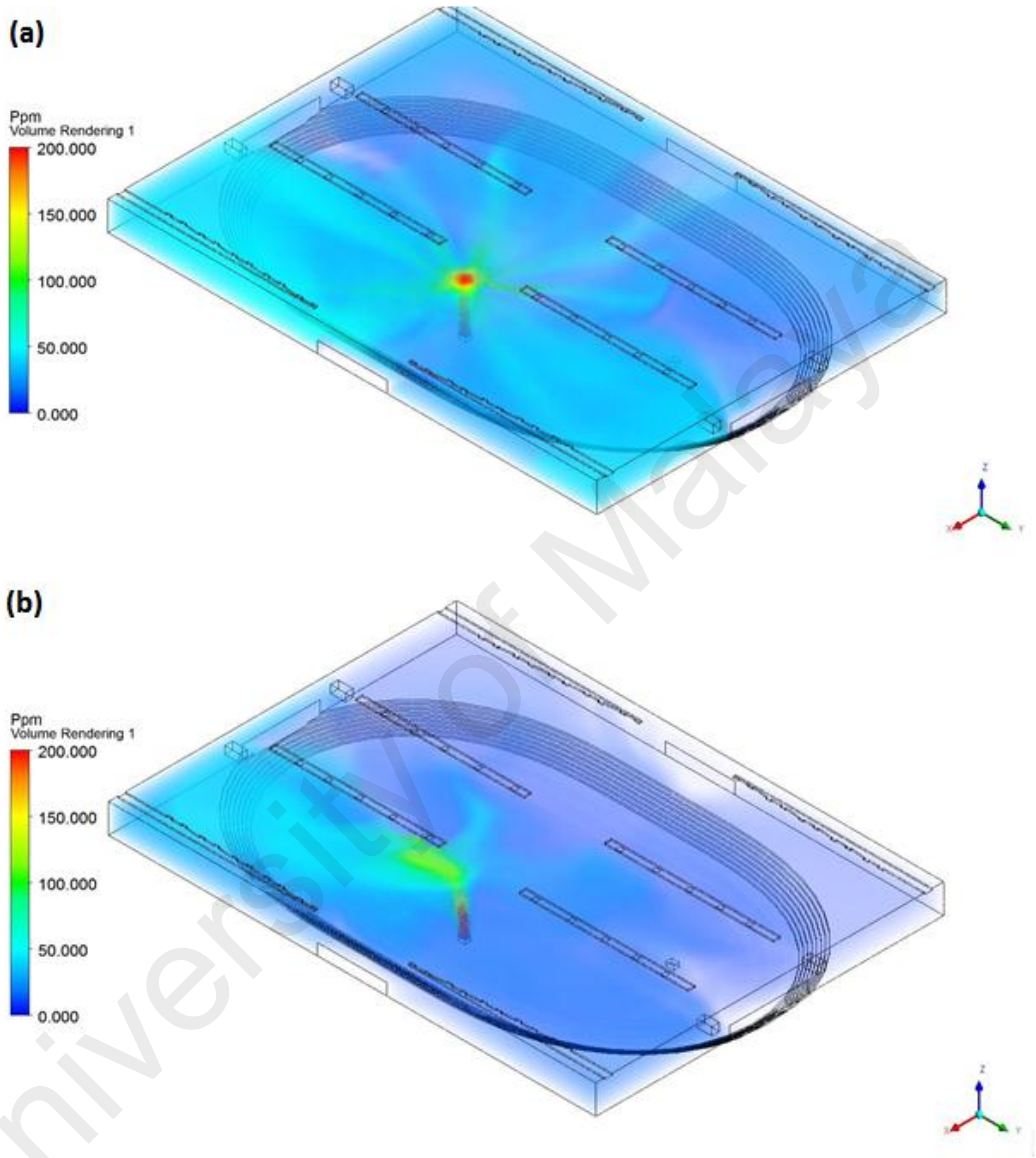


Figure 4.5: 3-D view of the smoke propagation in PPM, in t=90min, Exhaust system "OFF" (a), and down Exhaust system "ON" (b)

Observing Figure 4.1 to Figure 4.5 show that maximum concentration of CO (PPM) is around 50 PPM, when exhaust system is “OFF” and near fire position it goes up to 200 PPM. By implementation of the exhaust system in this case the concentration of CO is reduced by around 70%. It is also observed that the orientation of the vent has a major influence on smoke production; as additional simulation is run to compare the same size vent with the height and width length switched in both directions. As the fire source constantly requires fresh air, air is constantly moving towards the fire. While, fresh air is constantly entrained into the smoke.

The objective of zone pressurization is to limit the movement of smoke outside the fire or the smoke control zone by providing higher pressure areas adjacent to the smoke zone. Zone pressurization can be accomplished by exhausting the high concentrated smoke zone. In the event of a fire, when the doors are opened to the fire or smoke control zone, the adjacent zones are pressurized. The smoke zone is only a part of an area and all the rest of the Velodrome areas are pressurized.

Figure 4.6 to Figure 4.8 show the top view of the smoke concentration in different time 0-90 minutes when Exhaust system turned “OFF” (a) and Exhaust system turned “ON” (b) for the height of 2 meter.

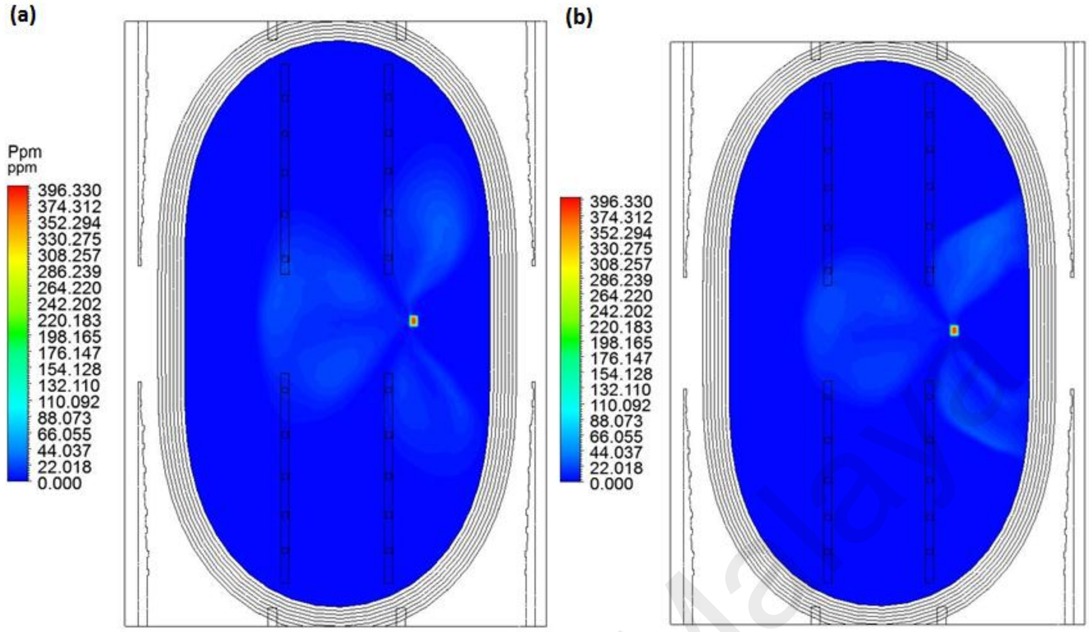


Figure 4.6: Smoke propagation in PPM in plane z=2m, t=10min, exhaust system "OFF" (a) and, exhaust system "ON" (b)

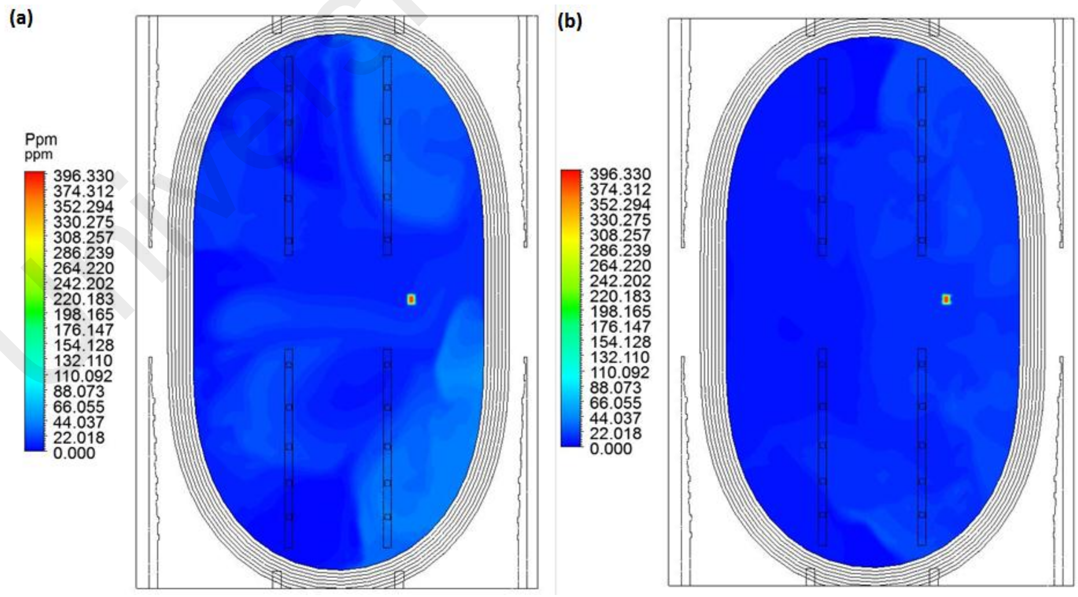


Figure 4.7: Smoke propagation in PPM in plane z=2m, t=50min, exhaust system "OFF" (a) and, exhaust system "ON" (b)

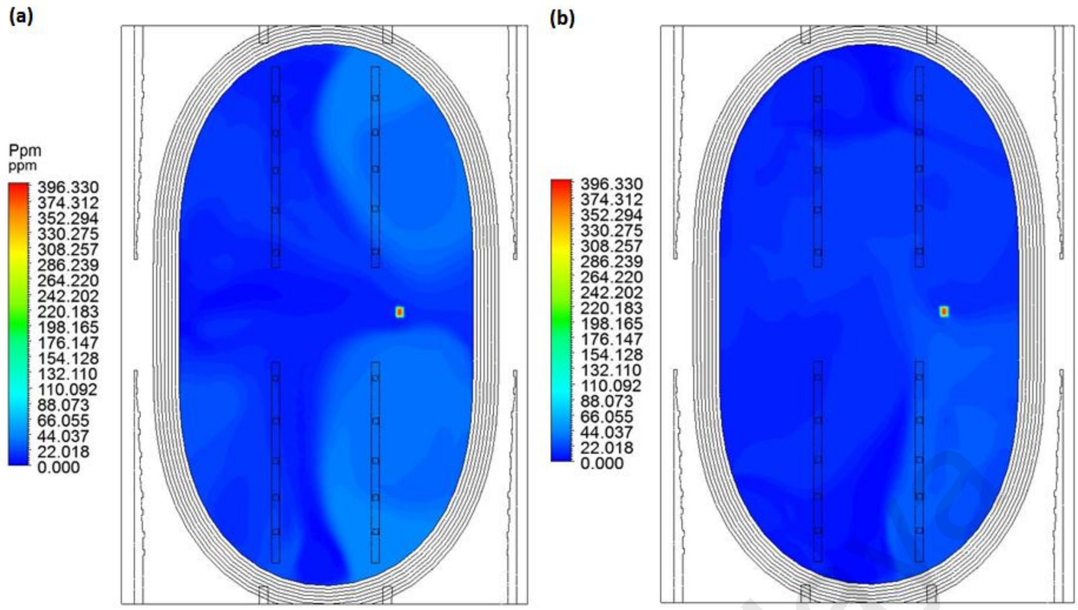


Figure 4.8: Smoke propagation in PPM in plane z=2m, t=90min, exhaust system "OFF" (a) and, exhaust system "ON" (b)

As can be seen in Figure 4.6 after 10 minute, there is not difference in concentration of CO in Velodrome between exhaust system "OFF" and "ON". But after 50 minutes and moreover, the smoke create in two zones as shown in Figure 4.7 and Figure 4.8 exhaust system reduces the CO concentration by up to 80% at the height of 2 meter in Velodrome.

4.2.2 Velocity Analysis of Case A

Figure 4.9 to Figure 4.11 show the top view of the velocity profile (exhaust system "OFF" (a) and exhaust system "ON" (b) after subsequent time from 0-90 minutes in height of Z=2 meter in Velodrome.

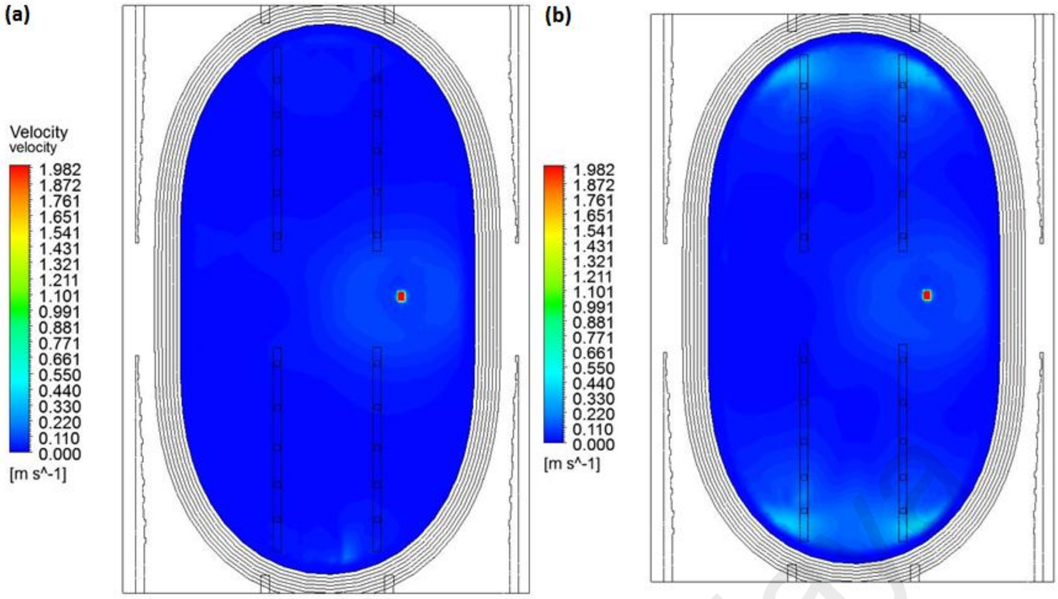


Figure 4.9: Velocity contour in plane z=2m, t=10min, exhaust system "OFF" (a) and exhaust system "ON" (b)

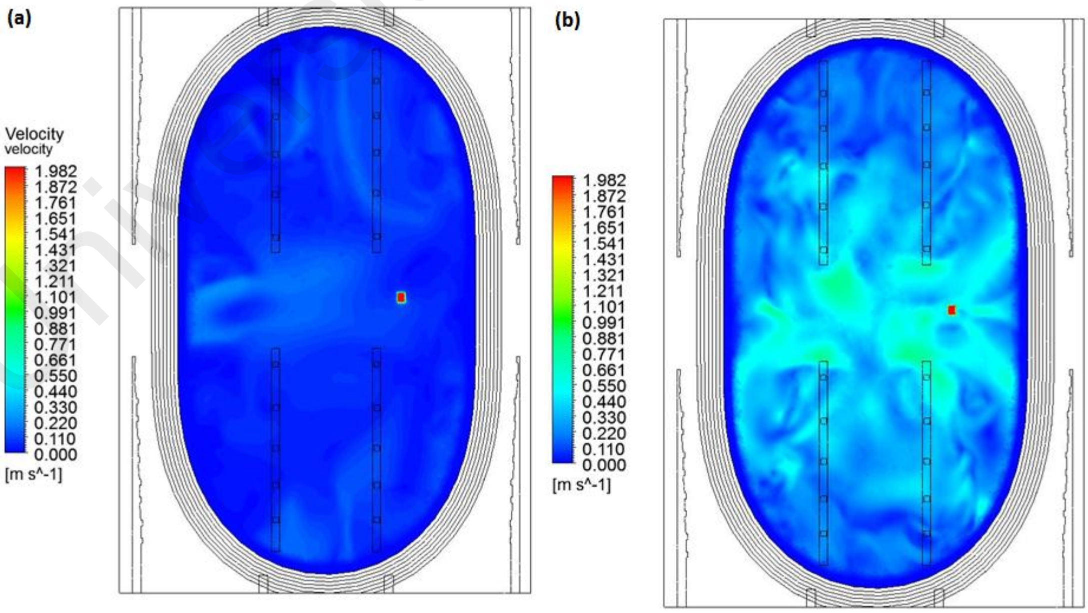


Figure 4.10: Velocity contour in plane z=2m, t=50min, exhaust system "OFF" (a) and, exhaust system "ON" (b)

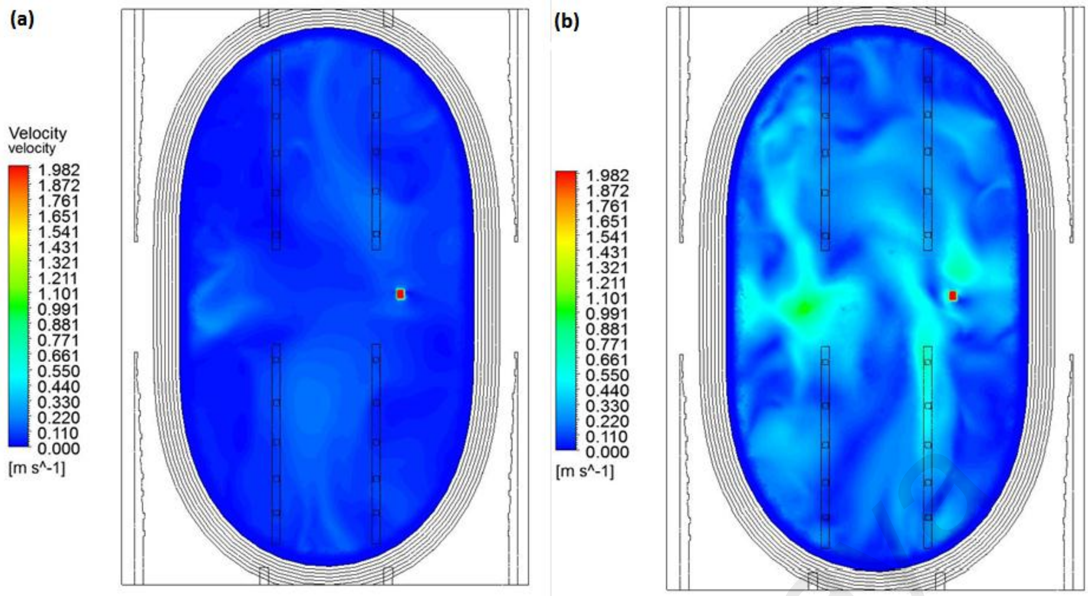


Figure 4.11: Velocity contour in plane z=2m, t=90min, exhaust system “OFF” (a) and exhaust system “ON” (b)

Velocity profiles shows by exhaust system velocity is increased in middle of model around 1.2 m/s. In this case critical velocity that is required to control the smoke in Velodrome is investigated. The modeling method was based on a mechanical ventilation system being located far enough from fire so that the velocity field is non-uniform.

4.2.3 Temperature Analysis of Case A

Figure 4.12 to Figure 4.14 show the temperature contour toggling between exhaust system turned “OFF” (a) and exhaust system turned “ON” (b) after subsequent time from 0-90 minutes in height of Z=2 meters in Velodrome.

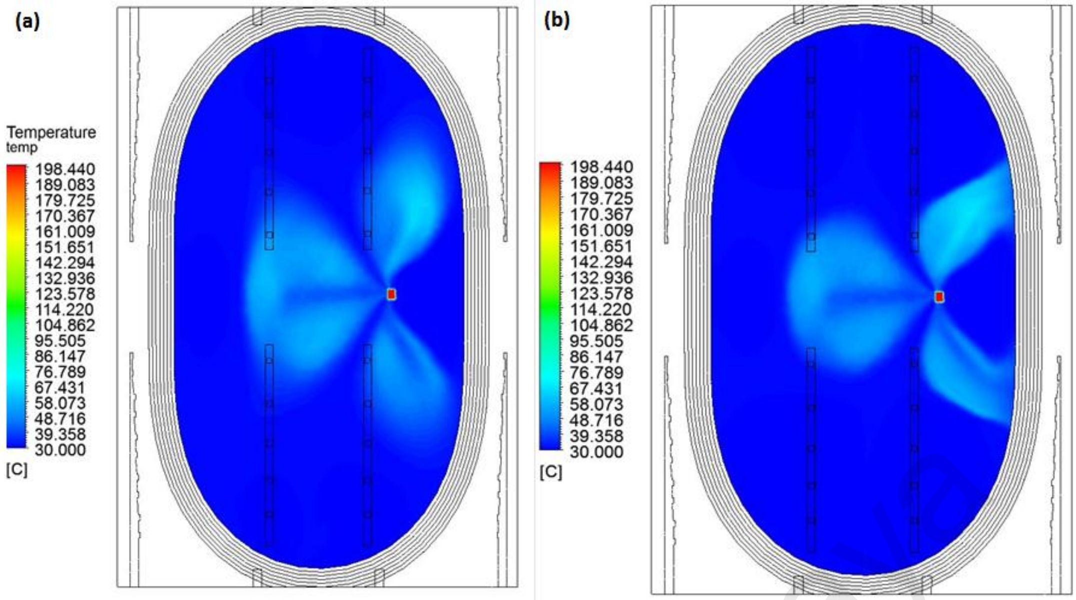


Figure 4.12: Temperature Distribution in °C in plane z=2m, t=10min, exhaust system "OFF" (a) exhaust system "ON" (b)

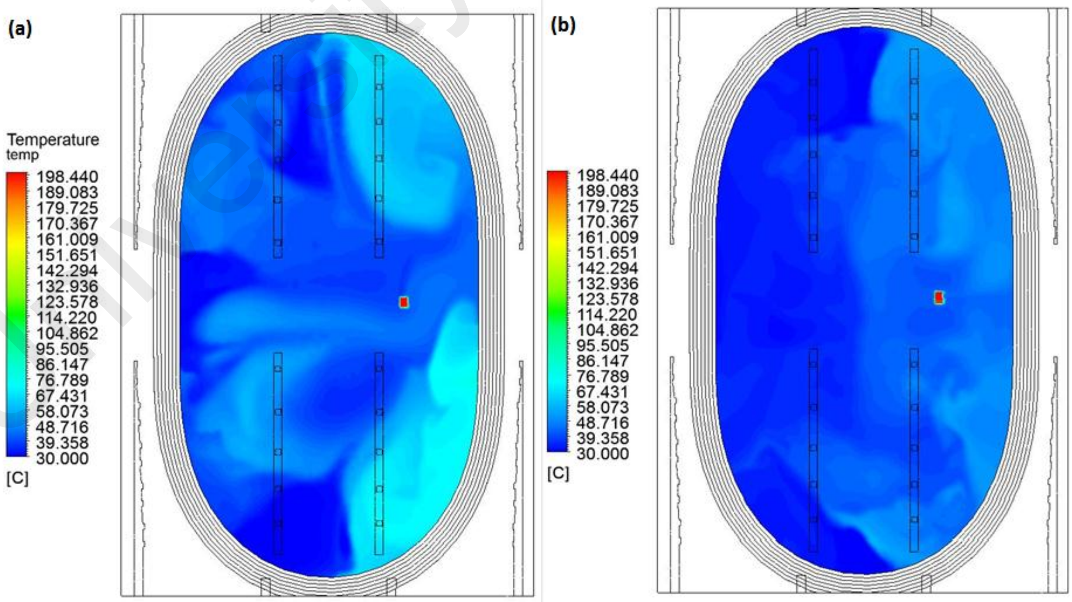


Figure 4.13: Temperature Distribution in °C in plane z=2m, t=50min, exhaust system "OFF" (a) and exhaust system "ON" (b)

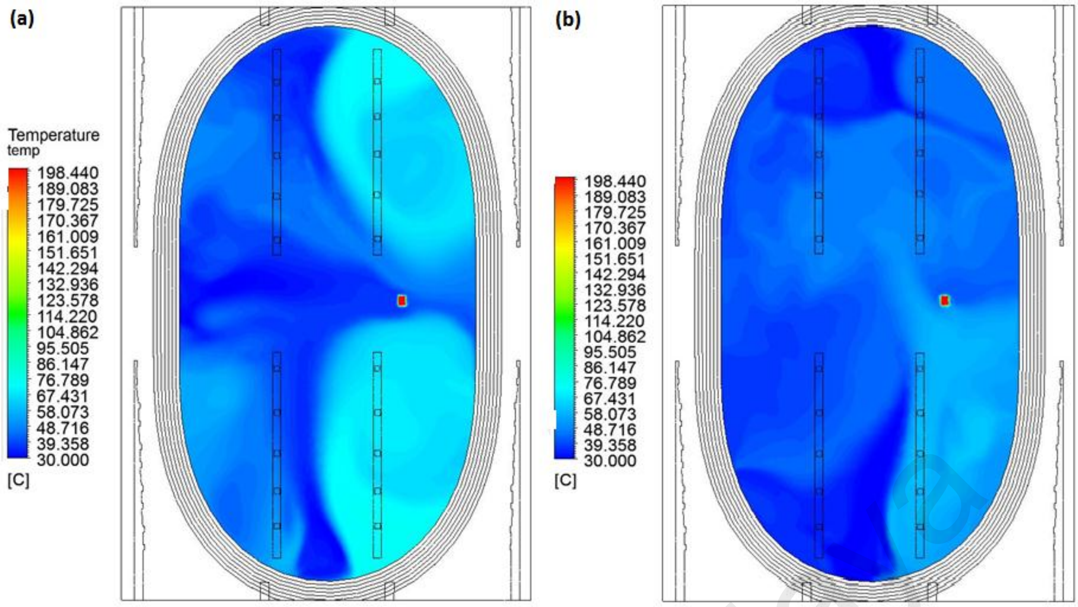


Figure 4.14: Temperature Distribution in °C in plane z=2m, t=90min, exhaust system "OFF" (a) and exhaust system "ON" (b).

It is observed that the temperature increases significantly for (10, 50, 90 minute) due to the formation of the smoke layer in two zone. The temperature is gradually increase during the remaining time of the simulation. It is clear from the Figure 4.14 that the distribution of temperature in the smoke layer features a certain level of horizontal stratification with the highest temperatures being found at the right side of Velodrome. The smoke layer height is designed for a 2 m clear height and the temperature values reported by CFD in the upper layer range from about 70°C. By applying exhaust system the reduction in temperature is estimate to be around 80% as it is directly related to the amount of smoke.

4.3 Case B

Base on Case B position of fire is located in between two exhaust system ducts in ceiling and near the corner of Velodrome that we analysis of the smoke propagation, velocity and Temperature according exhaust system. These results are also expressed graphically in Figure 4.15 to Figure 4.22.

4.3.1 Smoke Analysis of Case B

Case B was varied from Case A due to change is a fire position. Figure 4.15 to Figure 4.19 are 3D views of the smoke pattern and propagation in different times when exhaust system turned “OFF” (a), and exhaust systems turned “ON” (b)

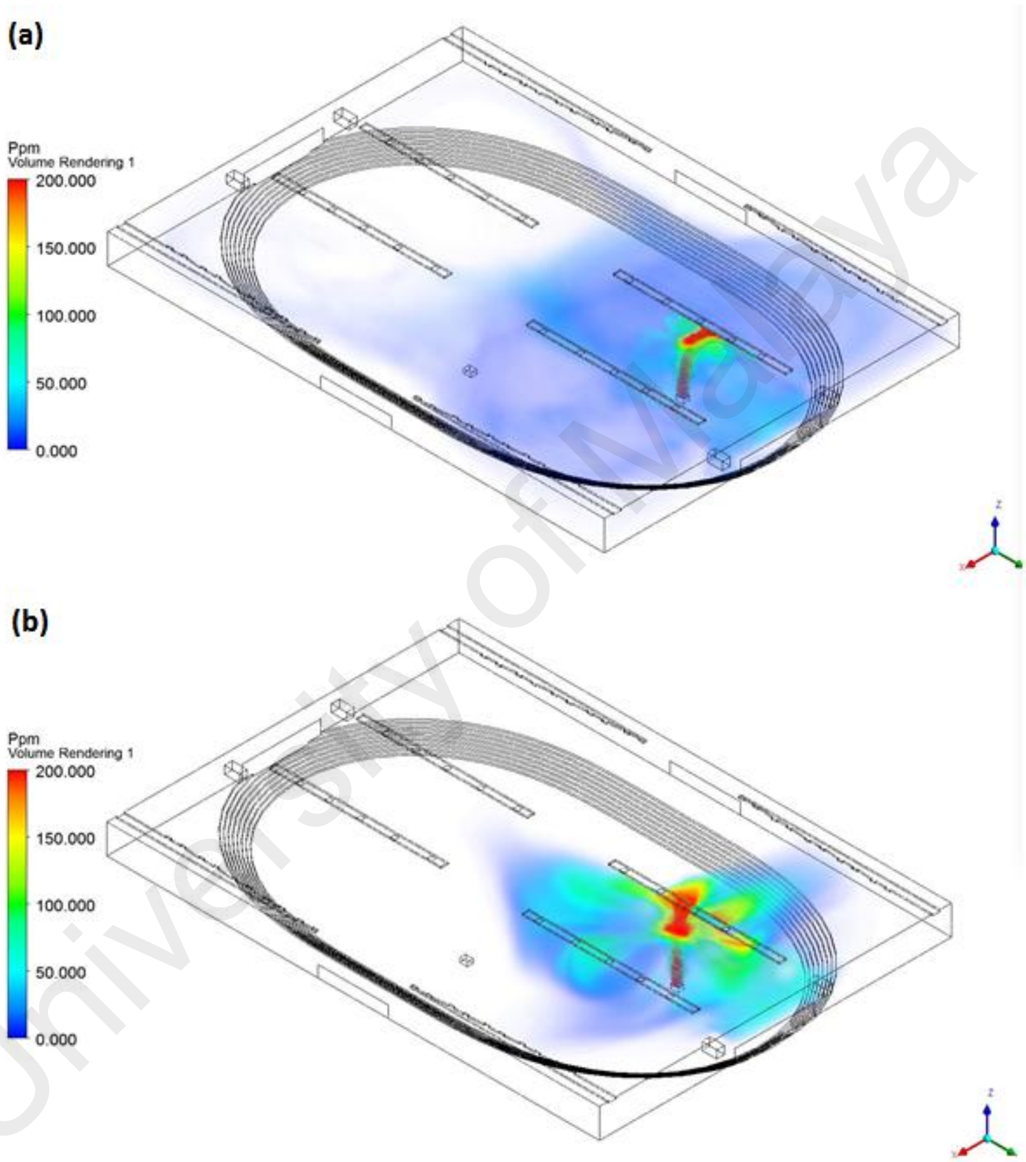


Figure 4.15: 3-D view of the smoke propagation in PPM, in t=10min, Exhaust system “OFF” (a) and Exhaust system “ON” (b)

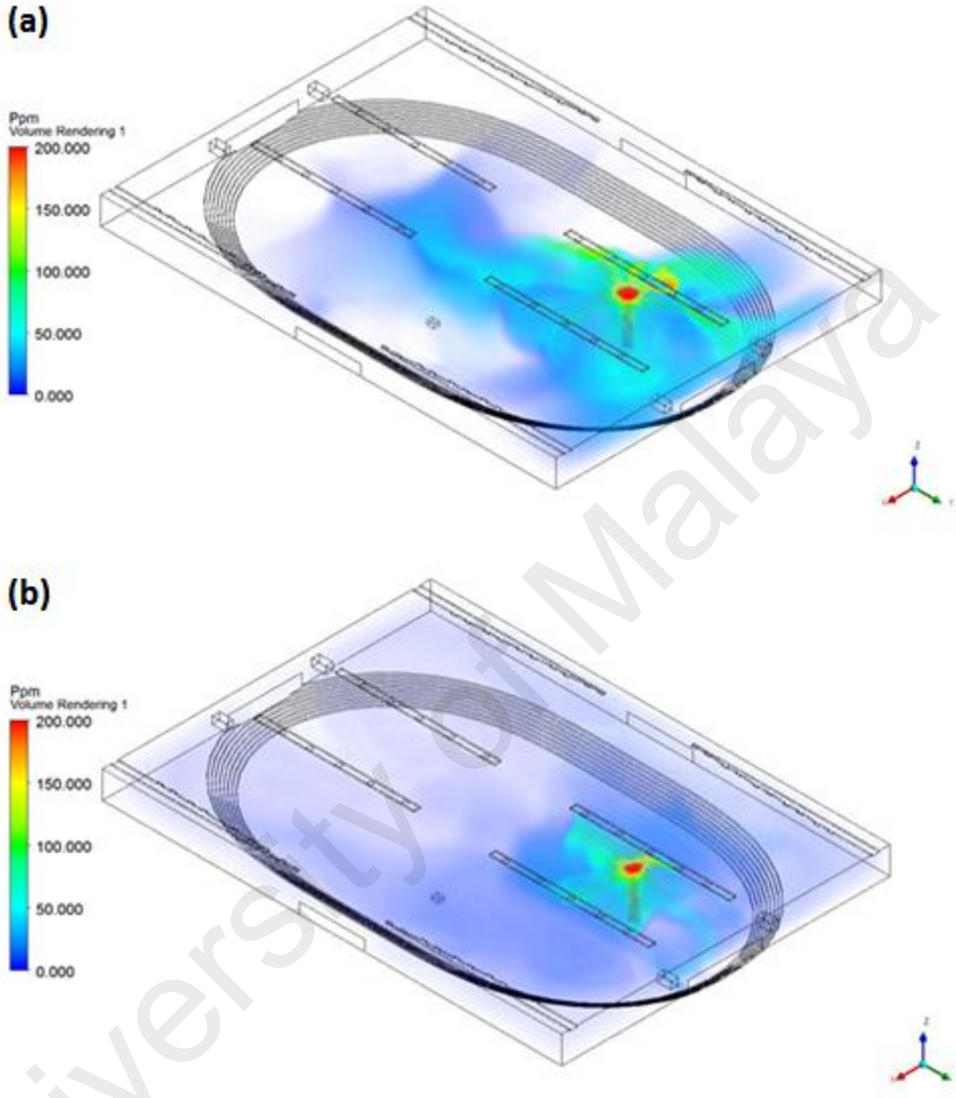


Figure 4.16: 3-D view of the smoke propagation in PPM, in t=20min, Exhaust system "OFF" (a) and Exhaust system "ON" (b)

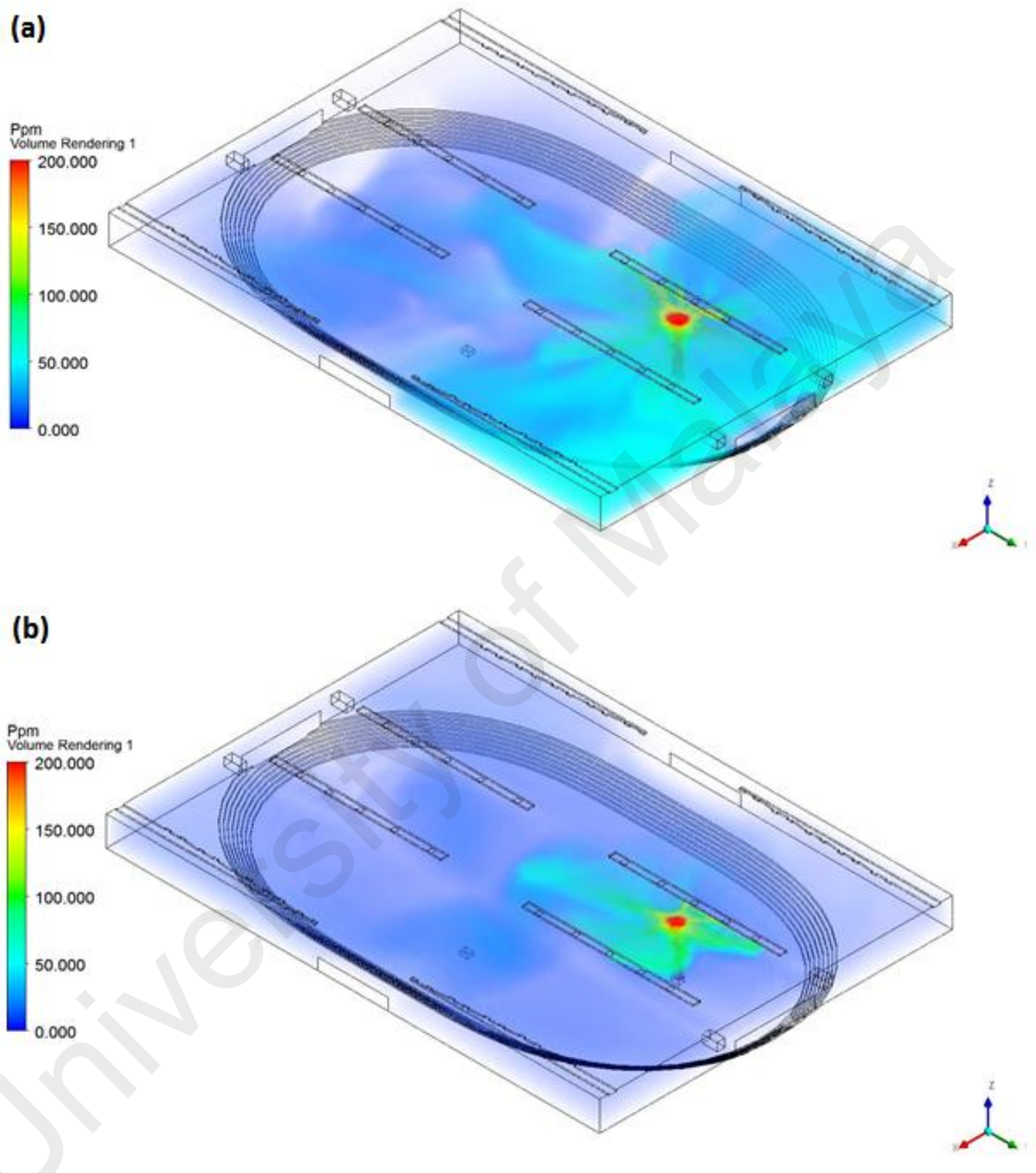
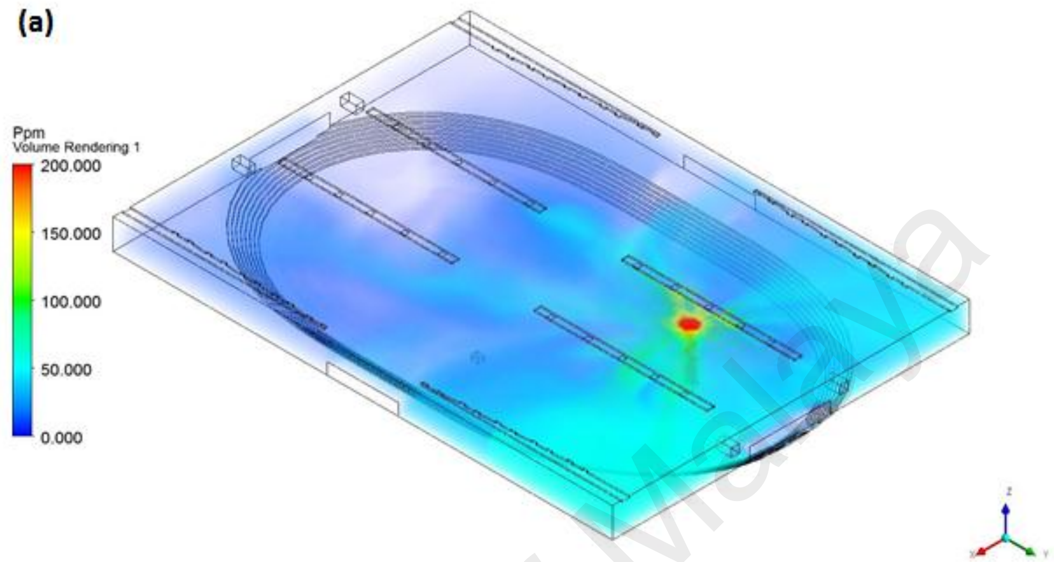


Figure 4.17: 3- D view of the smoke propagation in PPM, in t=50min, Exhaust system "OFF" (a) and Exhaust system "ON" (b)



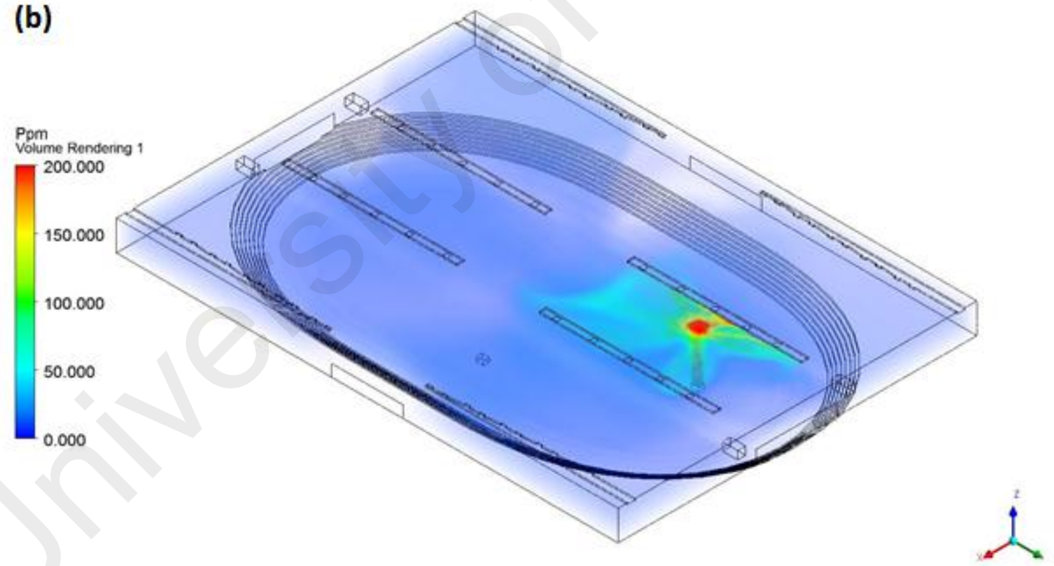


Figure 4.18: 3-D view of the smoke propagation in PPM, in t=70min, Exhaust system "OFF" (a) and Exhaust system "ON" (b)

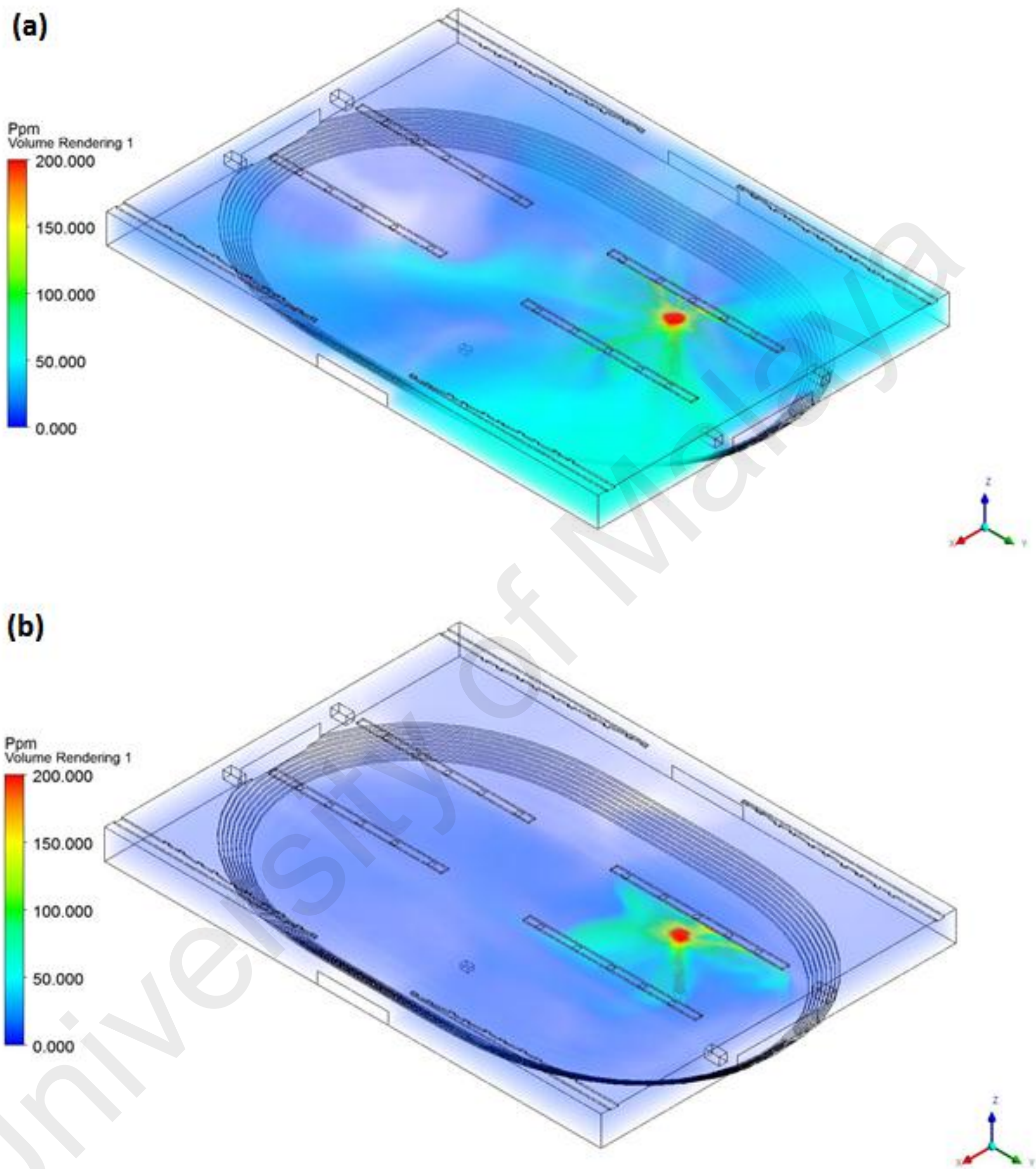


Figure 4.19: 3-D view of the smoke propagation in PPM, in t=90min, Exhaust system "OFF" (a) and Exhaust system "ON" (b)

Figure 4.15 to Figure 4.19 show the concentration of smoke in almost half of Velodrome increased to around 70 PPM when exhaust system is “OFF” but with applying exhaust system, it is reduced by 80%. In this case, the extension of smoke is happening only in one side of Velodrome. Interesting finding showed in Figure 4.15 that by applying exhaust system after 10 minute, amount of smoke in our model increased due to fire position that have negative impact the system. After 20 and 50, 70, 90 minutes as is shown Figure 4.16 to Figure 4.20 exhaust system controlled the smoke propagation in Velodrome to prevent increase of smoke concentration dramatically.

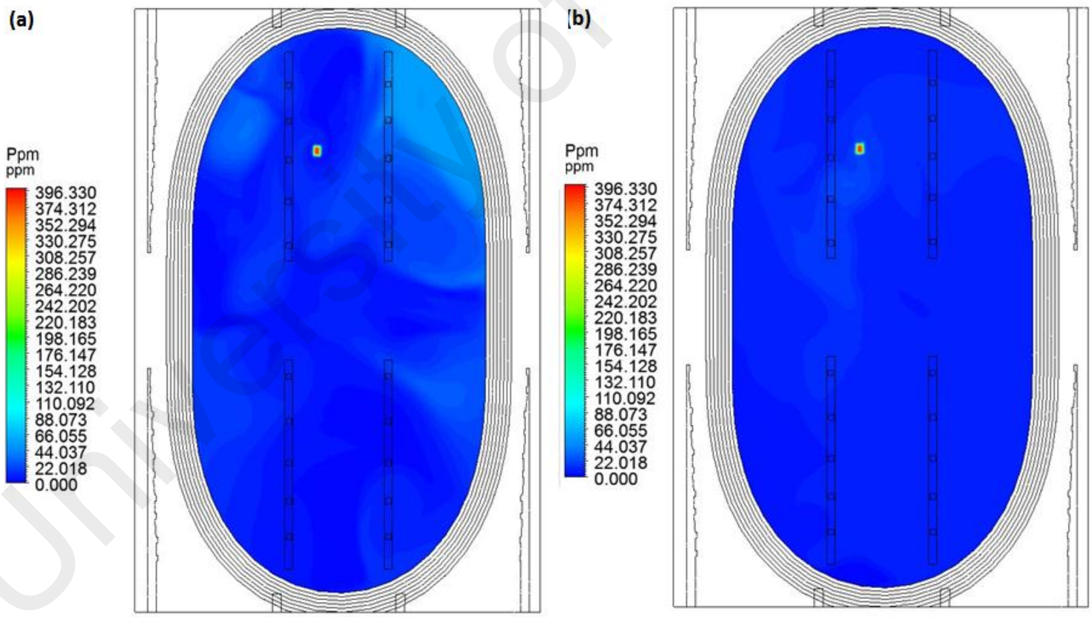


Figure 4.20: Smoke propagation in PPM in plane z=2m, t=10min, exhaust system “OFF” (a) and exhaust system “ON” (b)

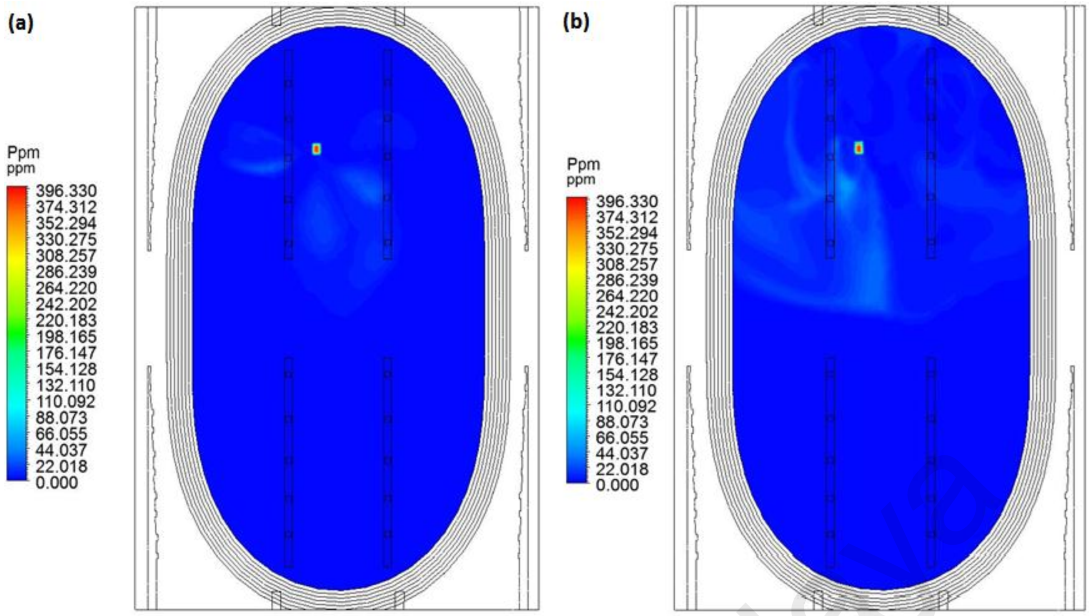


Figure 4.21: Smoke propagation in PPM in plane z=2m, t=50min exhaust system "OFF" (a) and exhaust system "ON" (b)

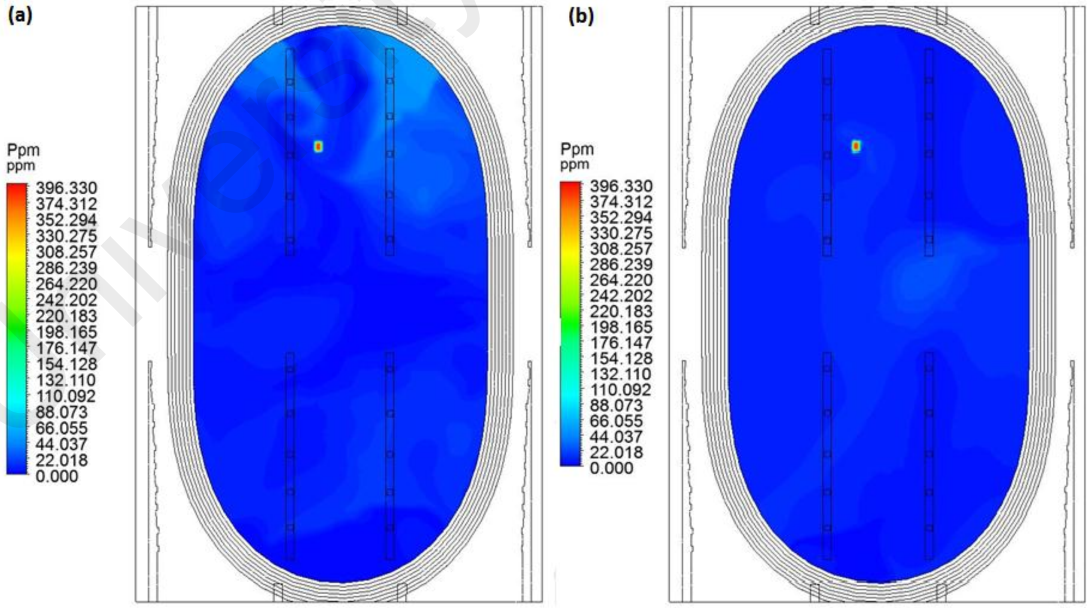


Figure 4.22: Smoke propagation (PPM) in plane z=2m, t=90min, exhaust system "OFF" (a) and exhaust system "ON" (b)

Figure 4.20 to Figure 4.22 show the top view of the smoke concentration in different time when Exhaust system turned "OFF" (a) and Exhaust system turned "ON" (b) for the height of 2 meter. Maximum amount of CO observed to be around 70 PPM that concentrate in fire position area when exhaust system is 'OFF" and with by applying exhaust system, it is reduced up to 90%.

4.3.2 Velocity Analysis of Case B

Figure 4.23 to Figure 4.25 show the top view of the velocity profile when exhaust system "OFF" (a) and exhaust system "ON" (b) after subsequent time from 0-90 minutes in height of Z=2 meter in Velodrome.

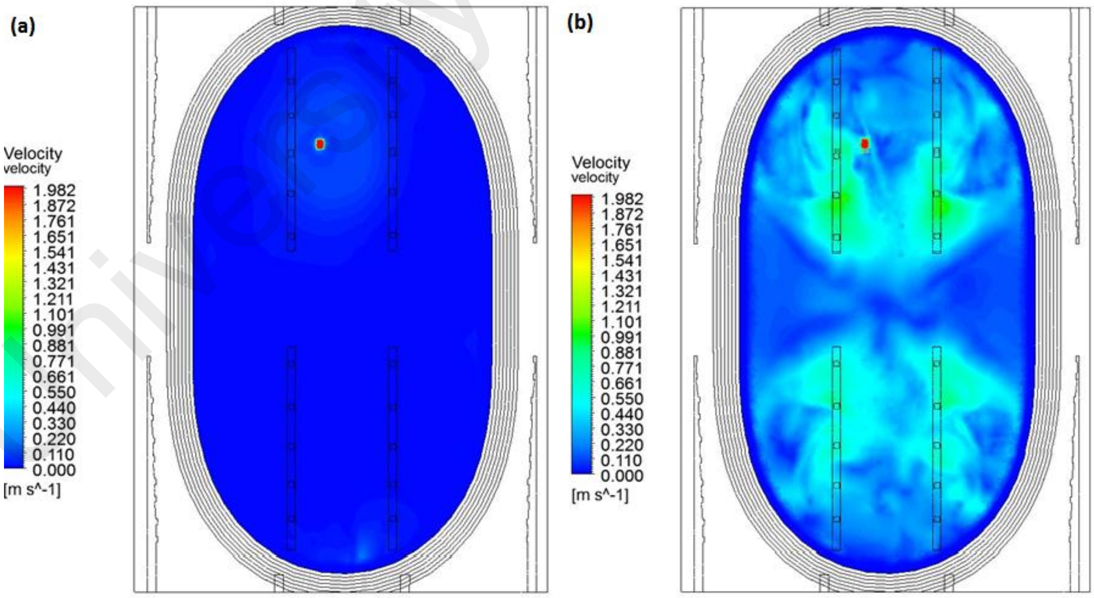


Figure 4.23: Velocity contour in plane z=2m, t=10min, exhaust system "OFF" (a) and exhaust system "ON" (b)

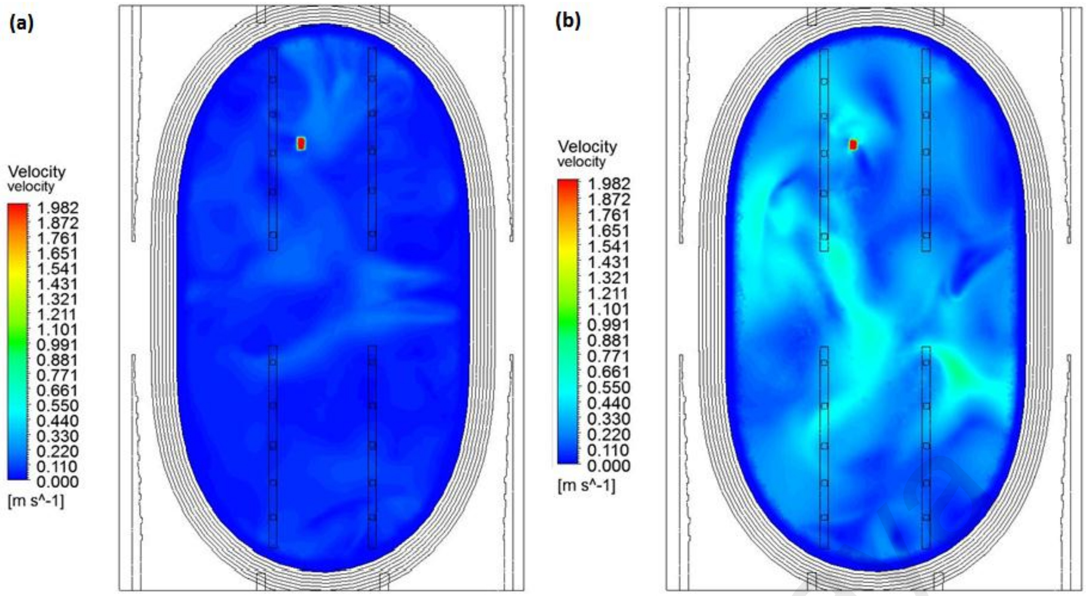


Figure 4.24: Velocity contour in plane z=2m, t=50min, exhaust system “OFF” (a) exhaust system “ON” (b)

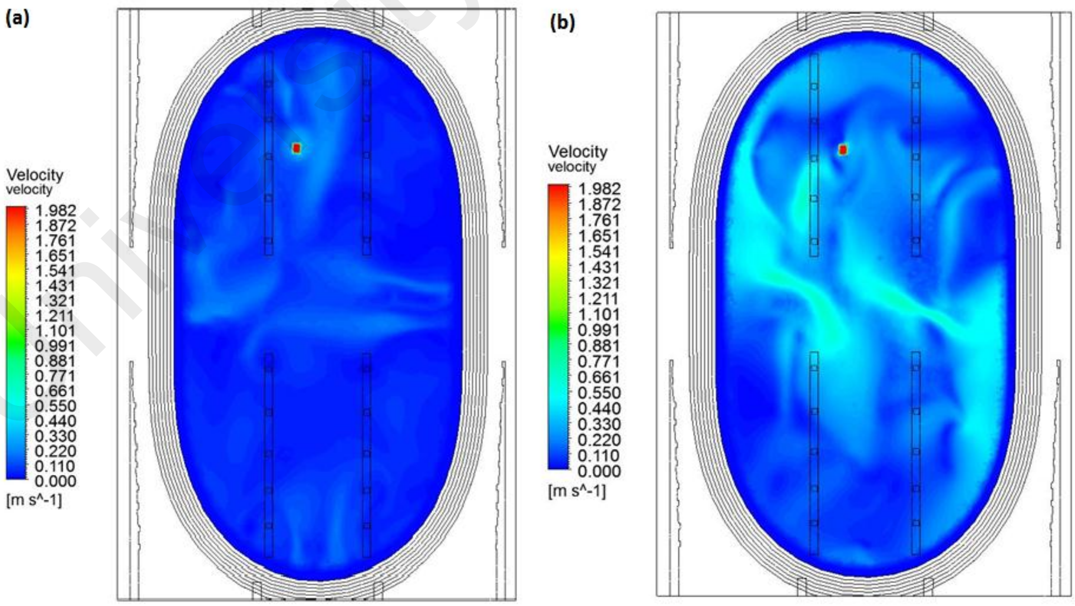


Figure 4.25: Velocity contour in plane z=2m, t=90min, exhaust system “OFF” (a) and exhaust system “ON” (b)

In Figure 4.23 to Figure 4.25, when the smoke exhaust system is operating, velocity is extended in four zone area around 1.3m/s but after 50 minutes and 90 minutes the velocity distribution show only at the center of the Velodrome.

4.3.3 Temperature Analysis of Case B

Figure 4.26 to Figure 4.28 show the temperature contour toggling between exhaust system 'OFF" (a) and exhaust system "ON" (b) after subsequent time from 0-90 minutes in height of Z=2 meters in Velodrome.

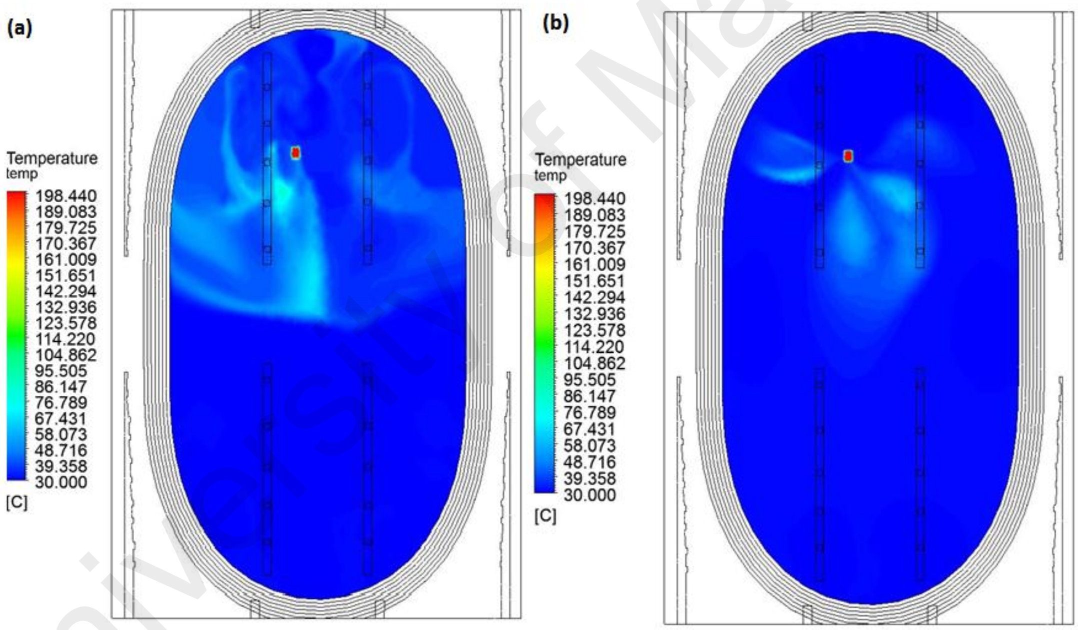


Figure 4.26: Temperature Distribution in °C in plane z=2m, t=10min, exhaust system "OFF" (a) exhaust system "ON" (b)

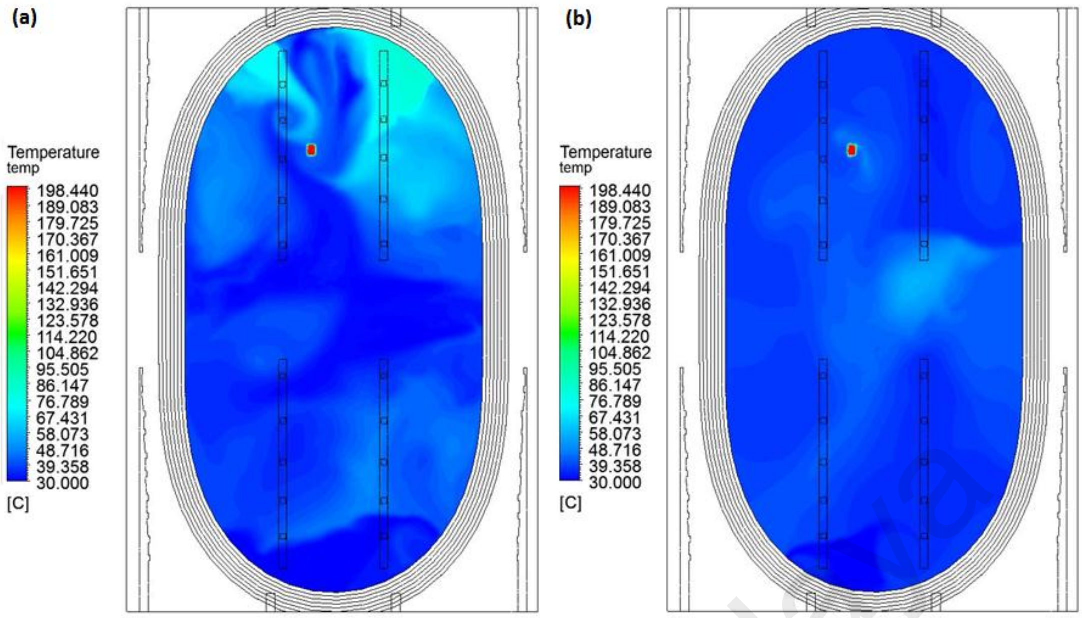


Figure 4.27: Temperature Distribution in °C in plane z=2m, t=50min, exhaust system “OFF” (a) and, exhaust system “ON” (b)

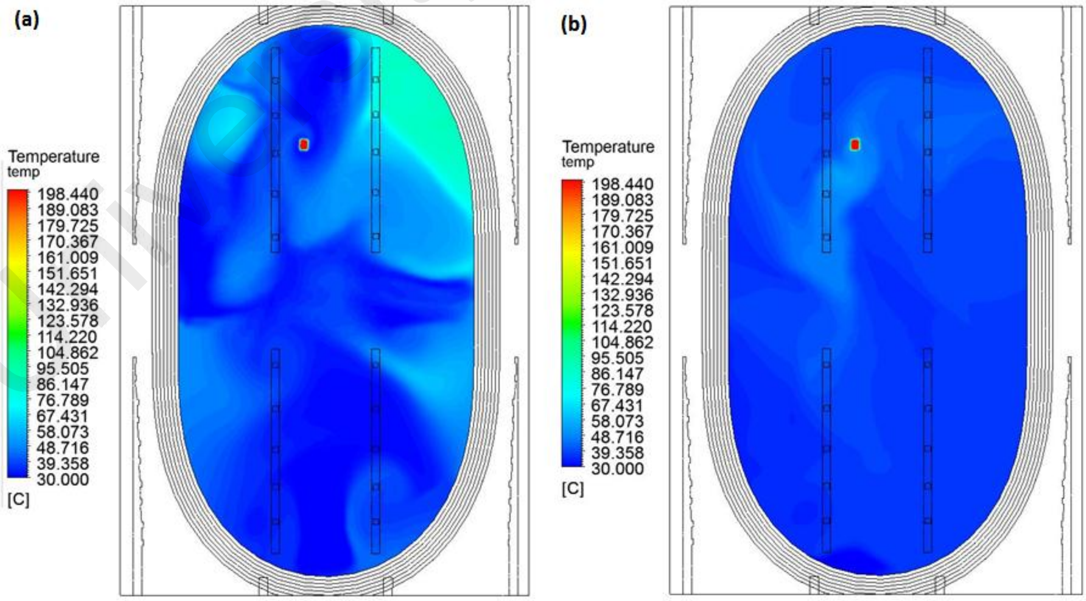


Figure 4.28: Temperature Distribution in °C in plane z=2m, t=90min, exhaust system “OFF” (a) and exhaust system “ON” (b)

Temperature control ventilation in Velodrome is a strategy that is used when the height of the smoke layer above the floor is not a critical design parameter. In this case, smoke exhaust can be used to achieve a maximum value of the temperature of the layer of emission gases. According to Figure 4.26 to Figure 4.27 with increasing time, amount of temperature increased and reached up to around 100^{o}C in some area when exhaust system is “OFF”. It can be seen in Figure 4.28, the amount of temperature increased in right corner that can be improve in future.

4.4 Case C

Selection of the jet fan used for fire system is a key factor in operation inside the Velodrome. The fans need to fulfill two major purposes. The first is to control the air or smoke velocity; and the second is to maintain pressurization in the non-affected tube to avoid smoke penetration through open cross-passage doors. Any active jet fan induces a lot of turbulence in the air or smoke movement. Thus, fans which are active within the smoke filled zone destroy any existing smoke pattern and hence fill the full Velodrome cross section with smoke. In each Case a pair of analysis are undertaken, i.e.; a) with jet fan and b) without the jet fans running. These results are also expressed graphically in Figure 4.29 to Figure 4.64. When jet fans are involved in CFD simulations, it is necessary to consider all parameters. The jet fan exit flow is horizontal also the distance between the ceiling and floor must be determined as accurately as possible. Best results have been observed between flow fields from a jet fan, mounted at the height Z=10 meter that is the reason of choice for this height.

4.4.1 Smoke Analysis of Case C

Figure 4.29 to Figure 4.40 are 3D views of the smoke pattern and propagation in different times when Jet fan turned "OFF" (a), and Jet fan turned "ON" (b) after subsequent time from 0-90 minutes with different height level of Velodrome from 2 - 13 meter.

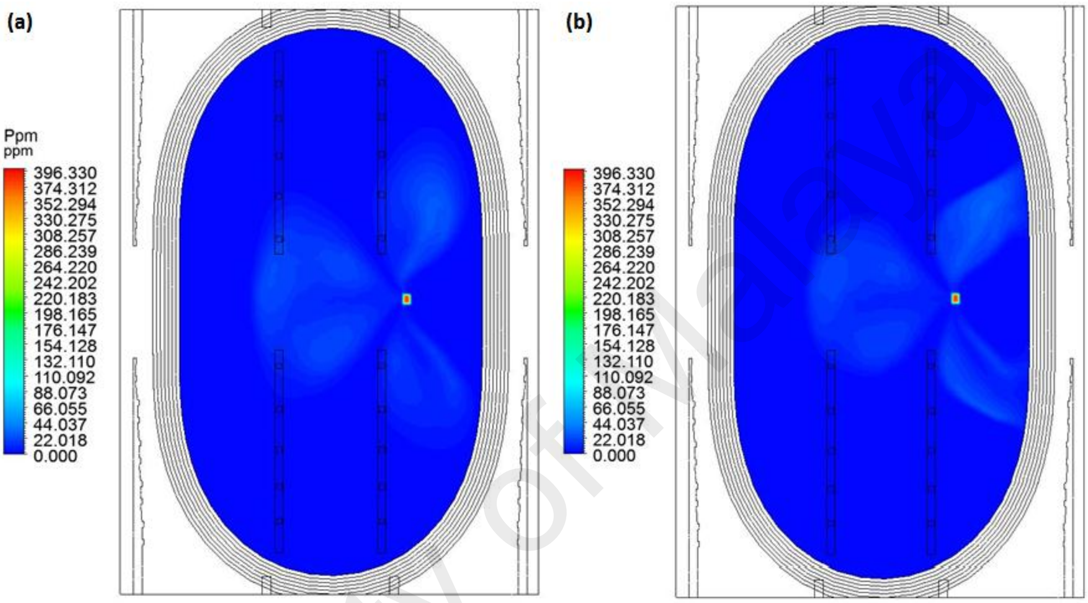


Figure 4:29: PPM Counter Z=2m, t=10 min, Jet fan "OFF" (a) Jet fan "ON" (b)

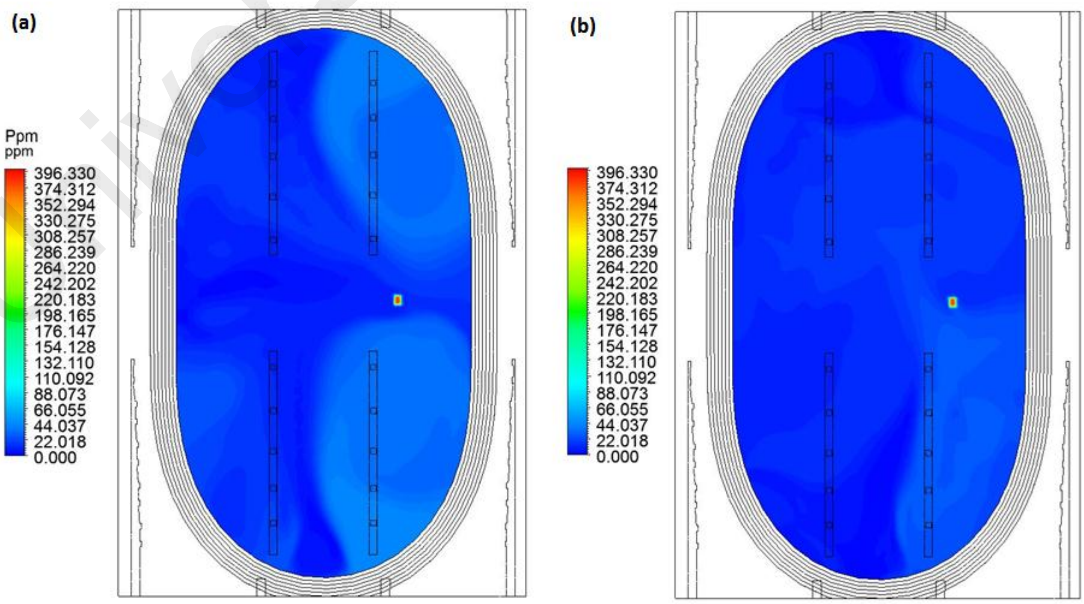


Figure 4:30: PPM Counter Z=2m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b)

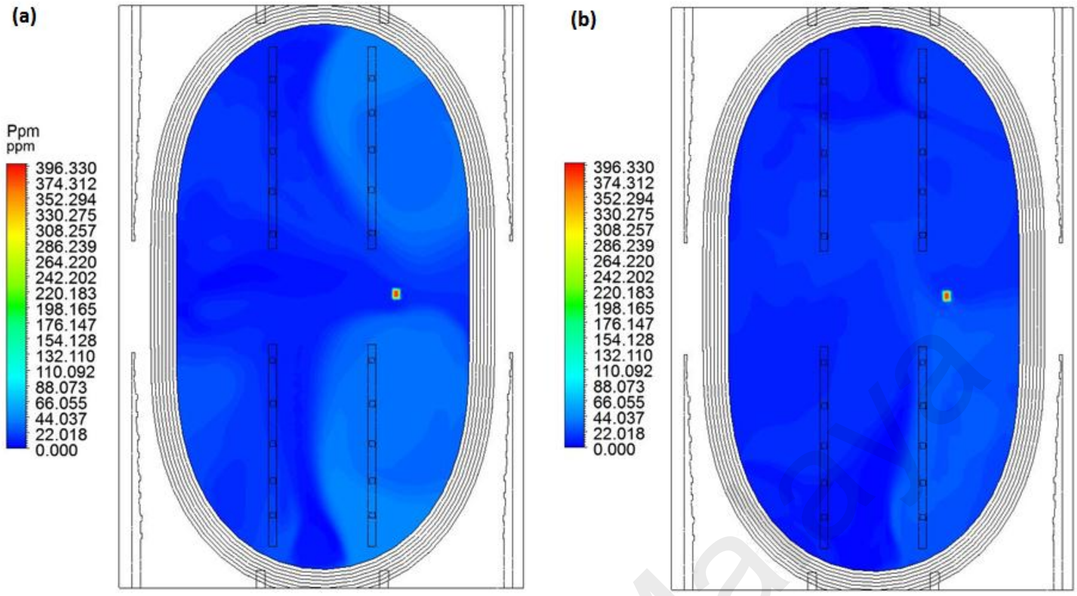


Figure 4.31: PPM Counter Z=2m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

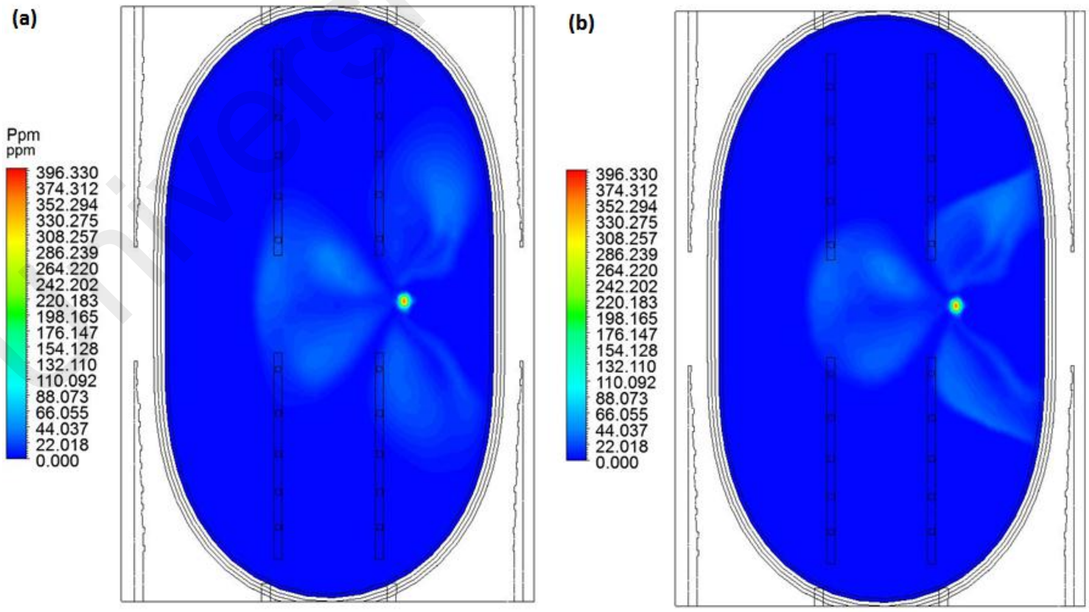


Figure 4.32: PPM Counter Z=4m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

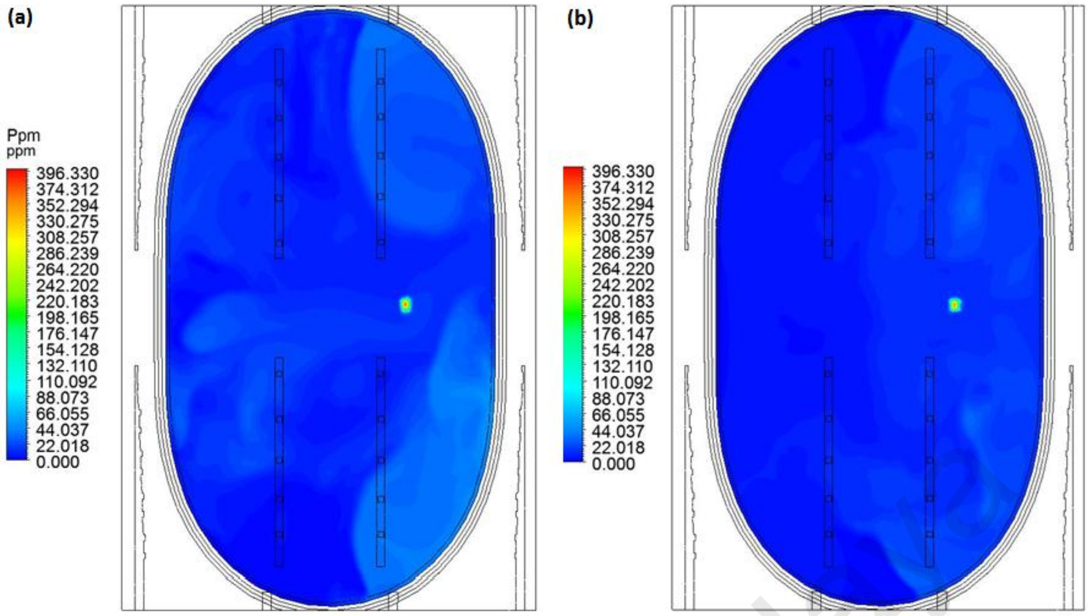


Figure 4.33: PPM Counter Z=4m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b)

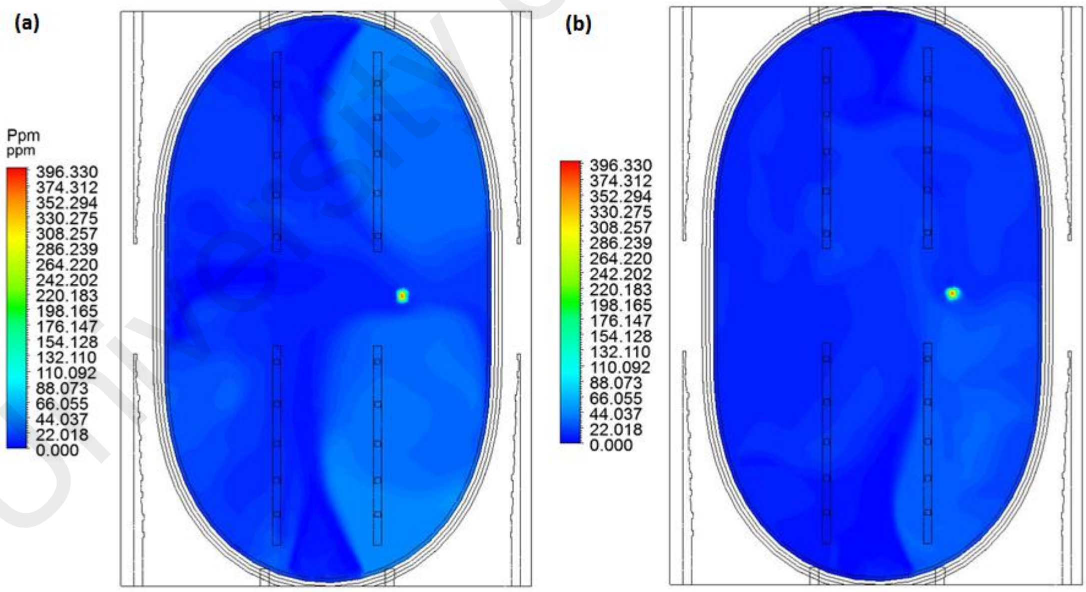


Figure 4.34: PPM Counter Z=4m, t=90 min, Jet fan "OFF" (a) Jet fan "ON" (b)

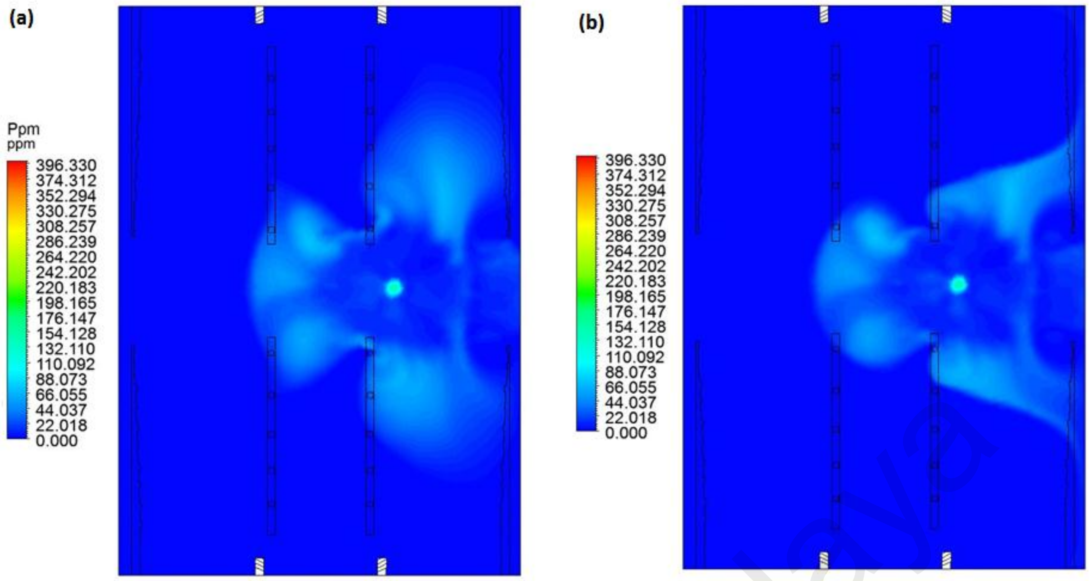


Figure 4.35: PPM Counter Z=10m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

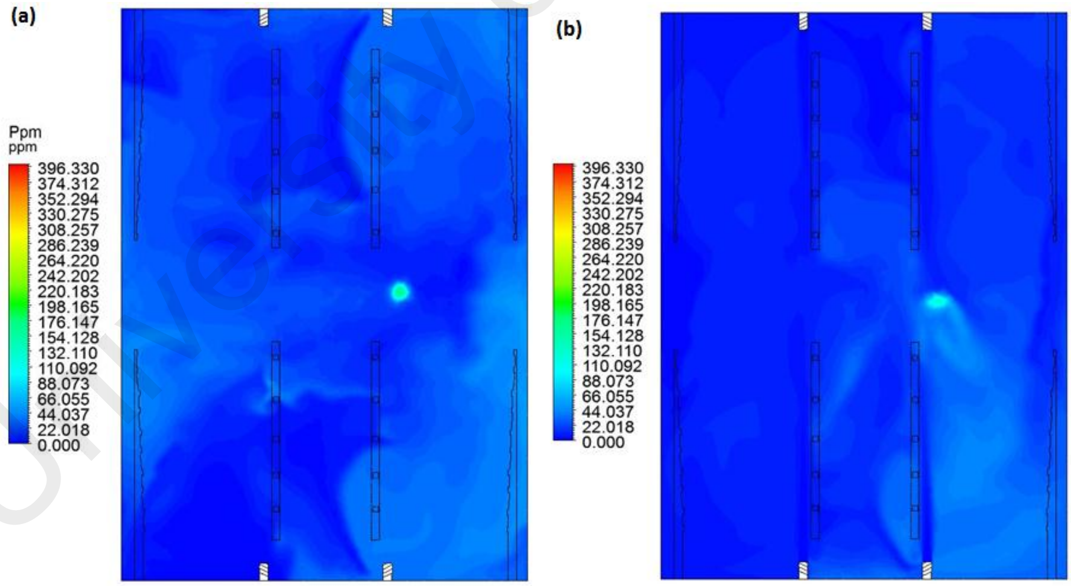


Figure 4.36: PPM Counter Z=10m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

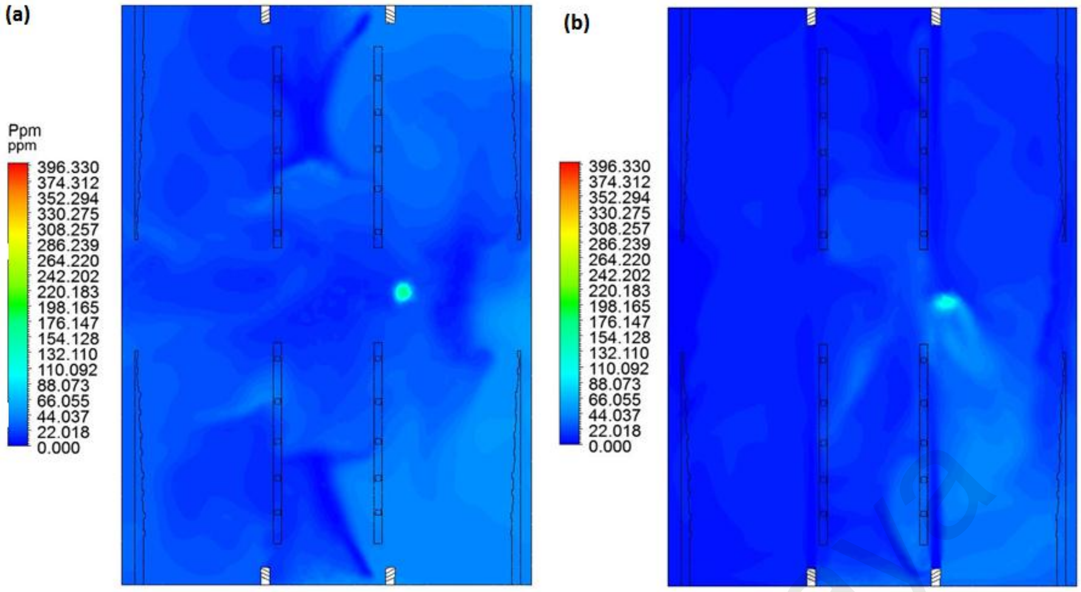


Figure 4.37: PPM Counter Z=10m, t=90 min, Jet fan "OFF" (a) Jet fan "ON" (b)

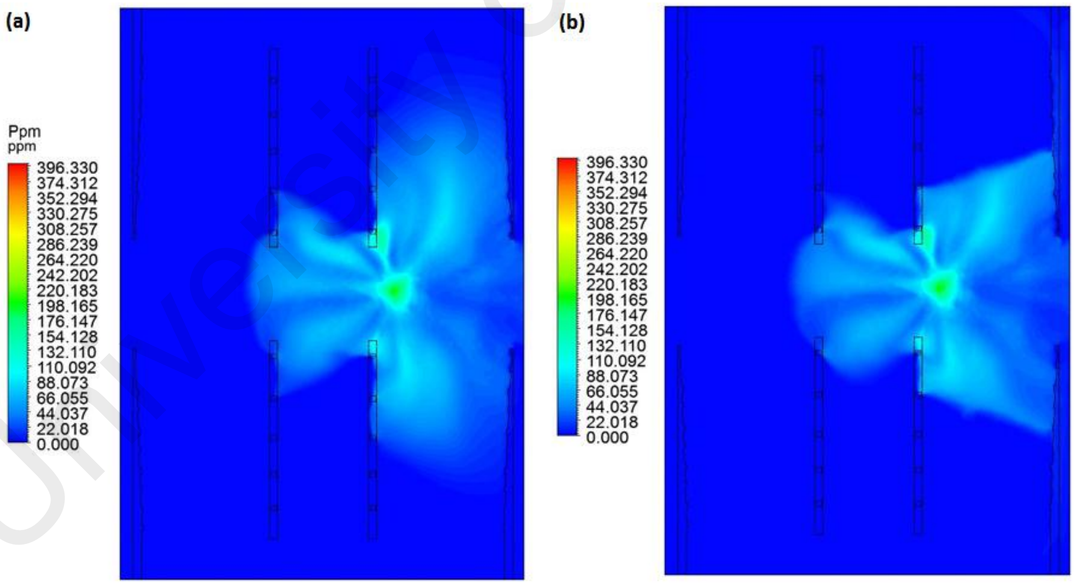


Figure 4.38: PPM Counter Z=13m, t=10 min, Jet fan "OFF" (a) Jet fan "ON" (b)

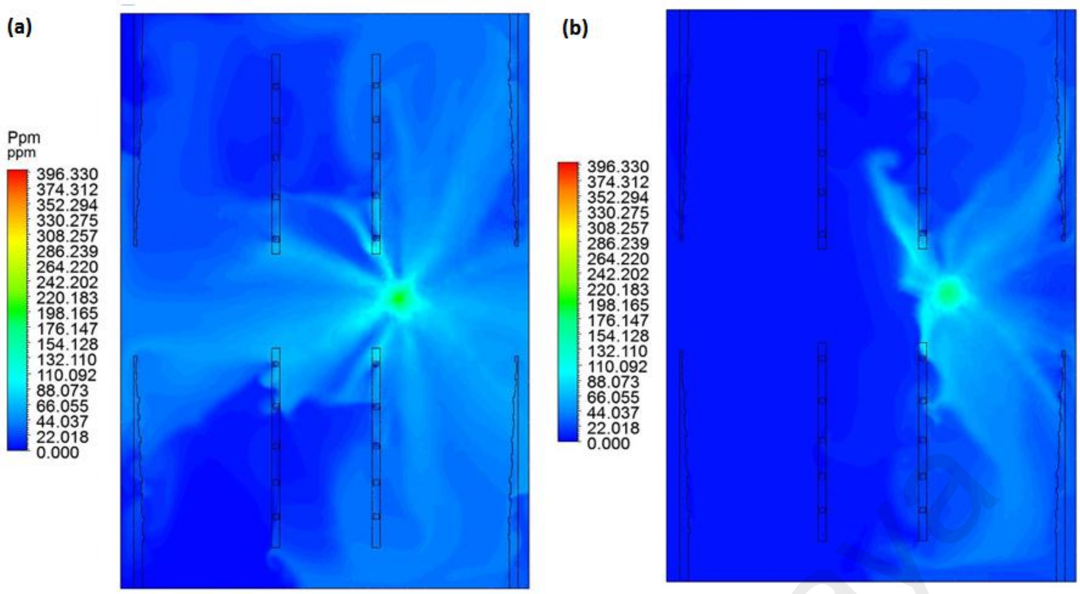


Figure4.39: PPM Counter Z=13m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

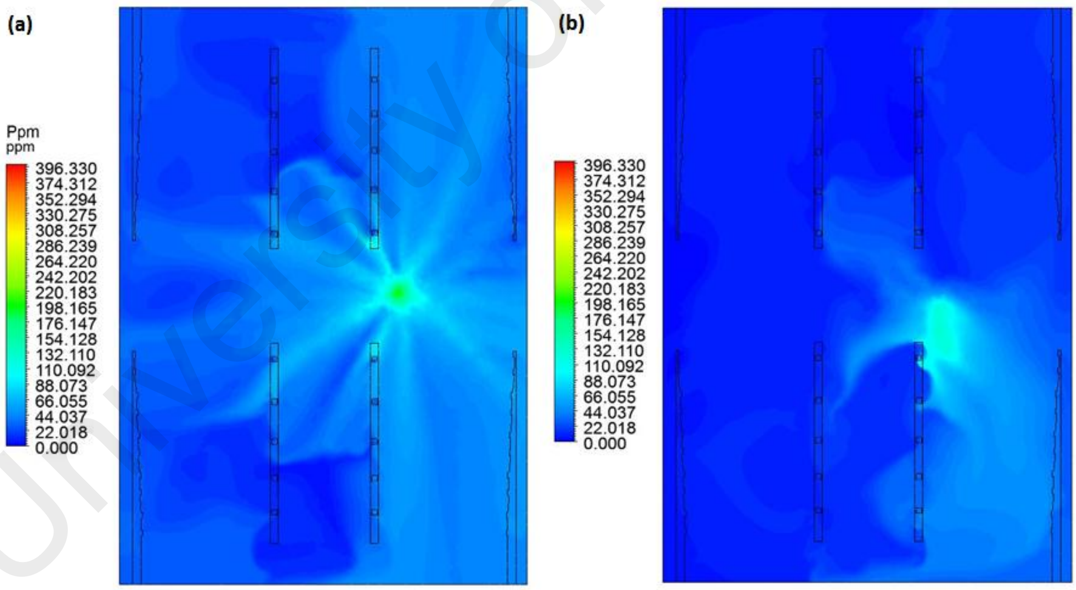


Figure 4.40: PPM Counter Z=13m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

Figure 4.29 to Figure 4.40 show that the smoke concentration increase along the Velodrome in the airflow direction. For the seating area, the head level is at the height 6 m above the floor. A plane being 2 m above the floor should be investigated; thus, the concentration contours are presented on jet fans. As expected, the peak smoke concentration occurred two zone as seen in Figure 4.30 and Figure 4.31 (between two exhaust ducts) where CO concentration was high and the airflow velocity was low. While the amount of smoke increases along the velodrome in flow direction, it is important to know that there was one-to-one correlation between the regional airflow velocity and CO dispersion. Thus, low airflow velocity in the upper zone and lower zone led to an increase in CO concentration. As illustrated in the Figure 4.29 to Figure 4.40, the maximum CO concentration in height at Z = 2 m for the case of jet fans are 40 PPM in subsequent time. This difference of about 65% between the peak concentrations is because of the jet fans are “ON” or “OFF”. As seen in the Figure 4.29 to Figure 4.40, the maximum CO concentration in the height at Z = 4 m for the case of jet fans are 50 PPM in subsequence time. This create a difference of about 75% between the peak concentrations was with and without applying Jet fans. Figure 4.29 to Figure 4.40 further emphasized how the low velocity regions correlated with high CO concentration areas. The permissible level of CO concentration for condition of safe respiratory was 70 PPM. While level of CO concentration in the Velodrome breathing zone is slightly lower than that. Hence, the number of jet fans provided a condition of safe respiratory for the people inside the Velodrome. While evacuating global concentration of smoke has a very close relation to average air velocity in the Velodrome, hence, convection was more important than diffusion in CO distribution. Diffusion coefficient was a function of turbulent viscosity, and it depended on the flow field. Investigating on smoke concentration in the whole domain of the velodrome as well as in the horizontal plane 2, 4, 10, 13 m above the floor, it has showed that when the jet fans blew, the highest amount of CO was about 35 PPM, while the corresponding value with using the jet fans was about 70 PPM.

4.4.2 Velocity Analysis of Case C

Figure 4.41 to Figure 4.52 show the top view of the velocity profile when Jet fans turned “OFF” (a) and Jet fans turned “ON” (b) after subsequent time from 0-90 minutes in height of Z=2 to Z= 13 meter in Velodrome.

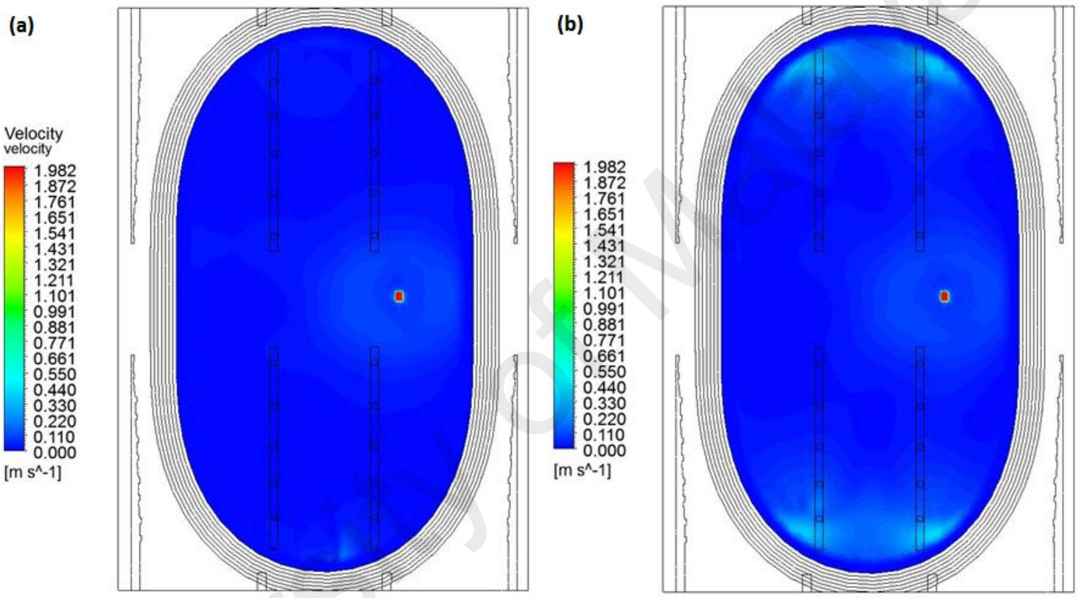


Figure4.41: Velocity Counter Z=2m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

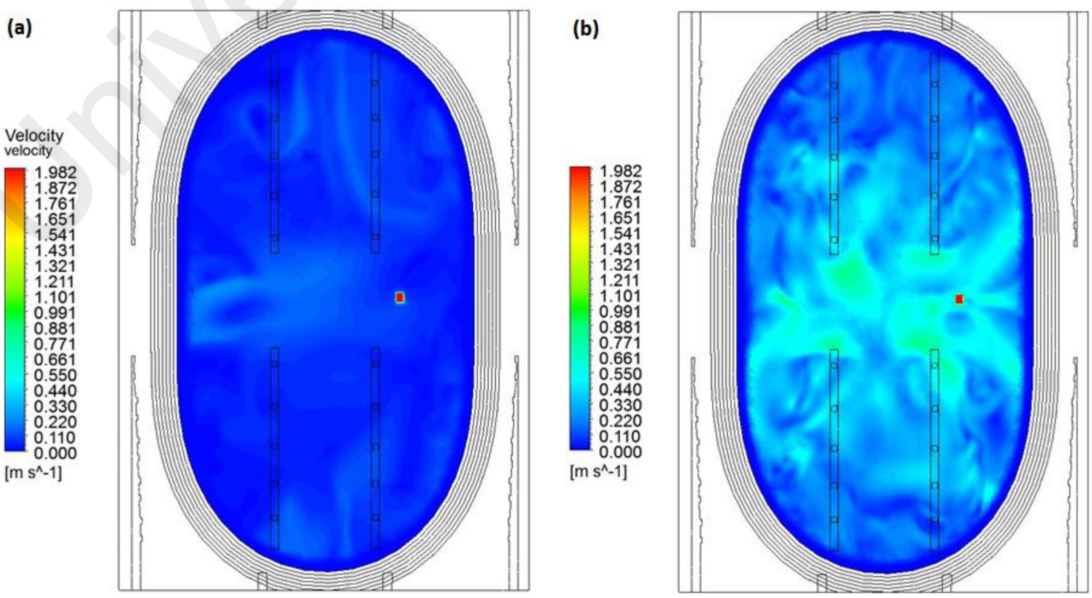


Figure 4.42: Velocity Counter Z=2m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

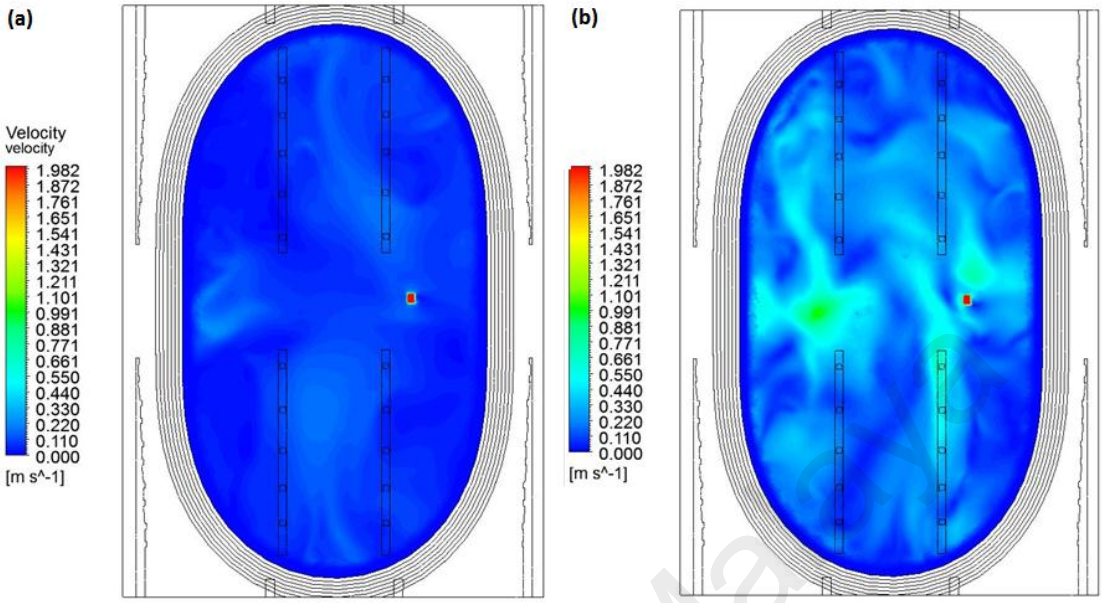


Figure 4.43 Velocity Counter Z=2m, t=90 min, Jet fan "OFF" (a) Jet fan "ON" (b)

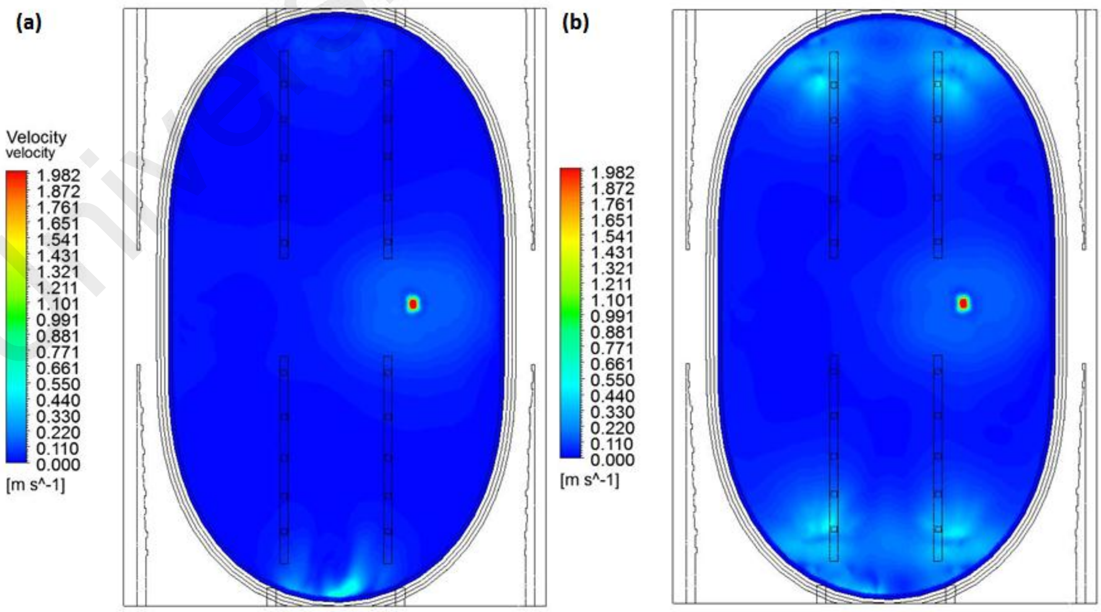


Figure 4.44: Velocity Counter Z=4m, t=10 min, Jet fan "OFF" (a) Jet fan "ON" (b)

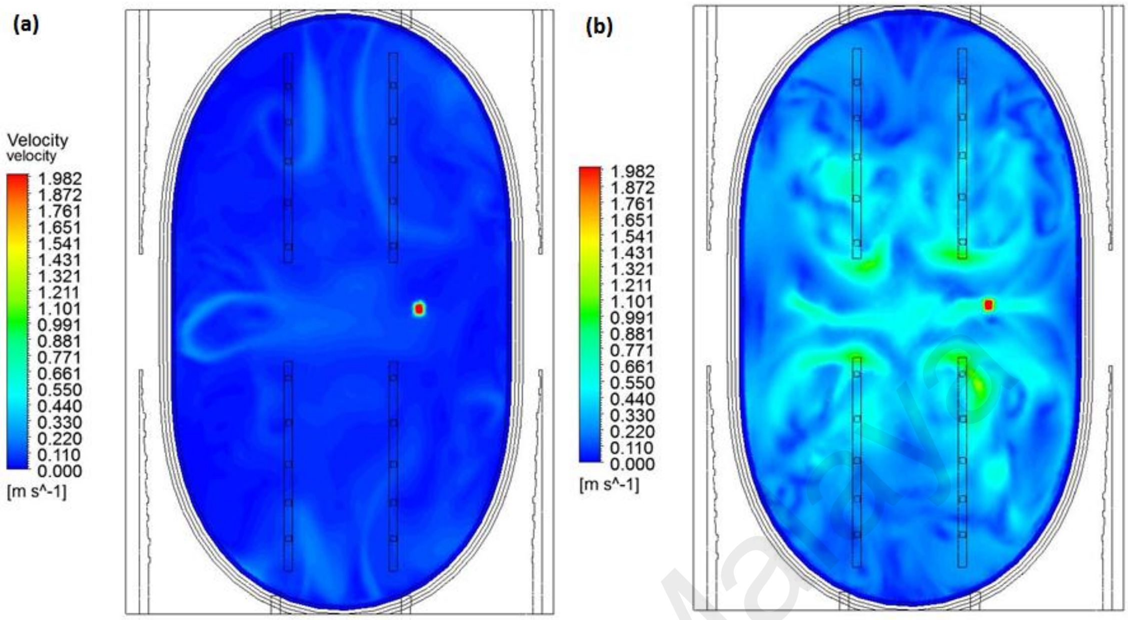


Figure 4.45: Velocity Counter Z=4m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

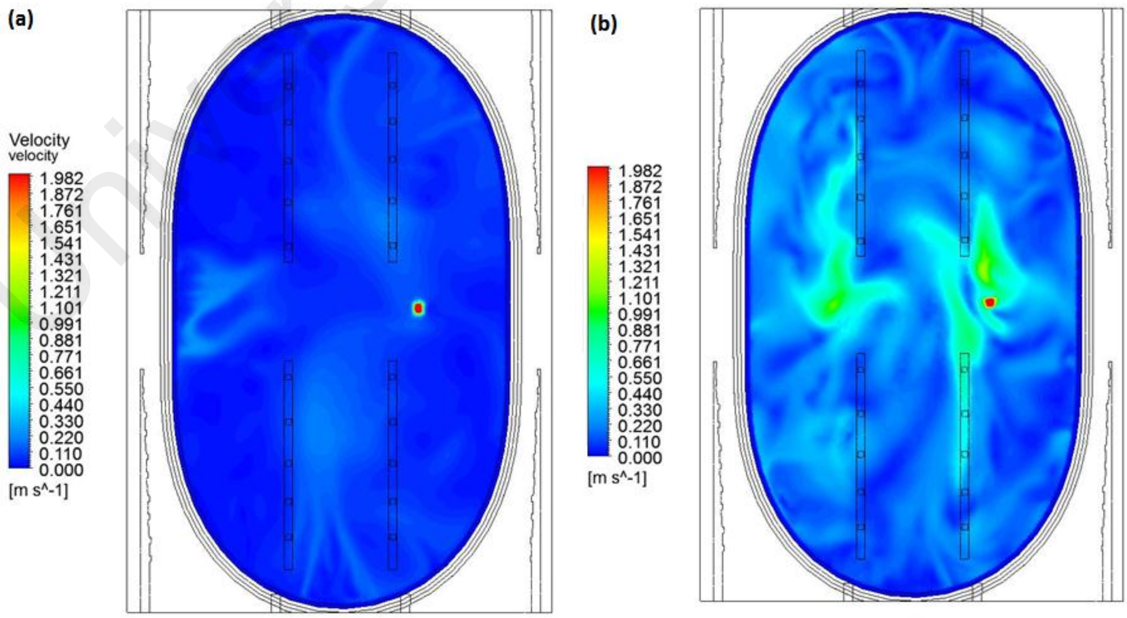


Figure 4.46: Velocity Counter Z=4m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

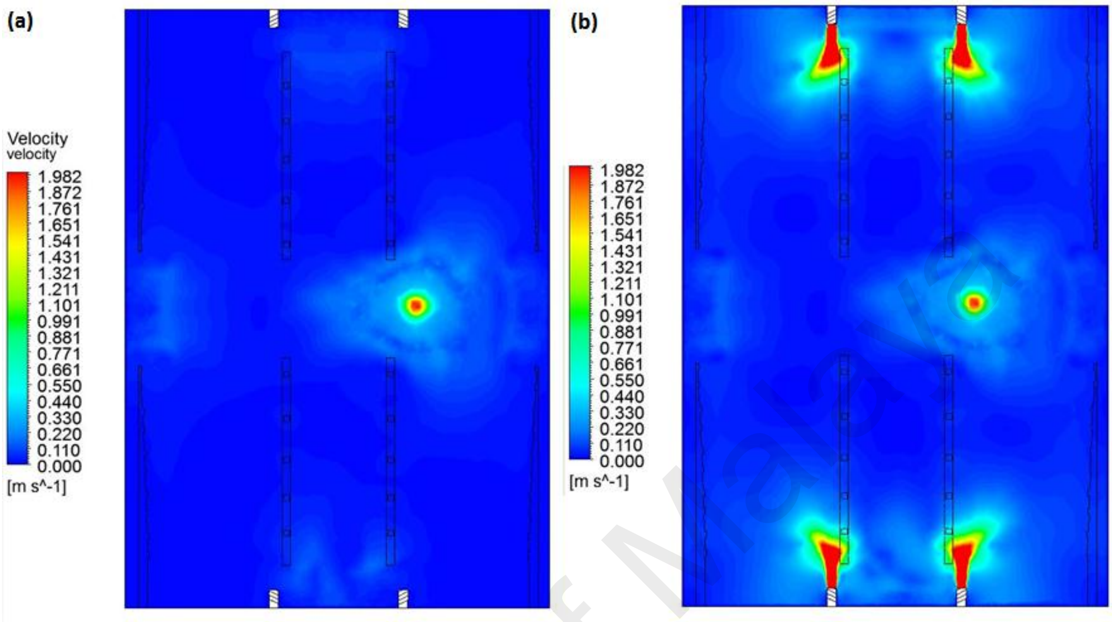


Figure 4.47: Velocity Counter Z=10m, t=10 min, Jet fan "OFF" (a) Jet fan "ON" (b)

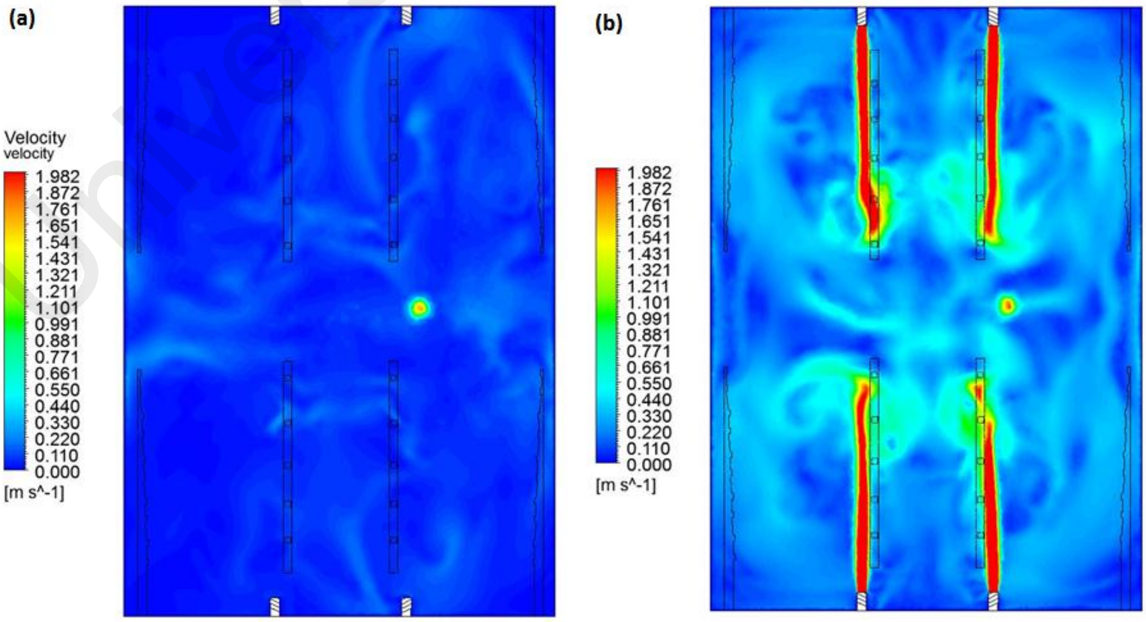


Figure 4.48: Velocity Counter Z=10m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b)

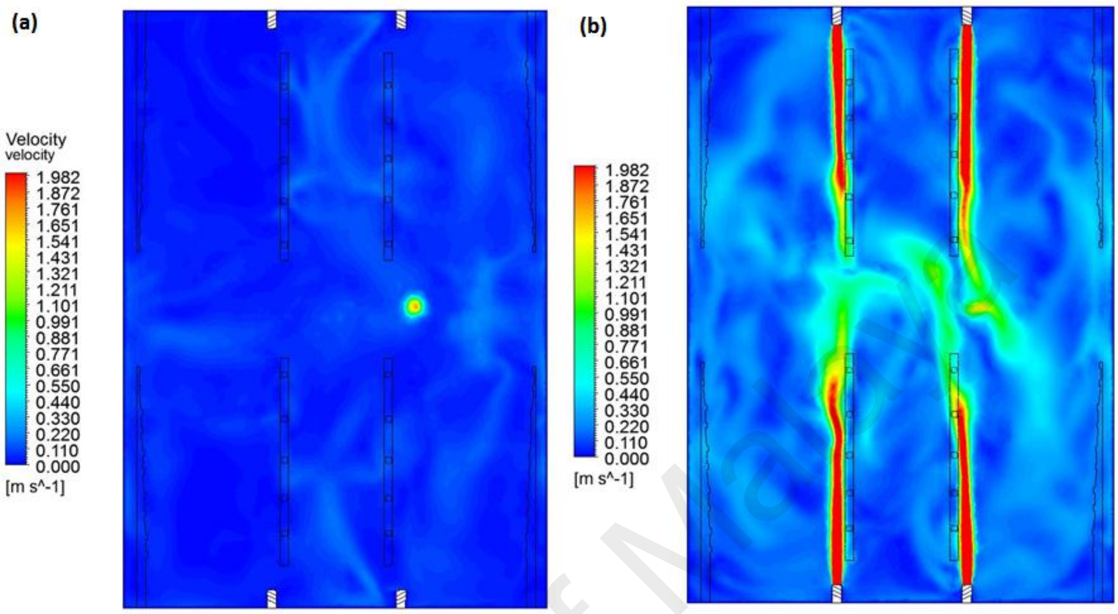


Figure 4.49: Velocity Counter Z=10m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

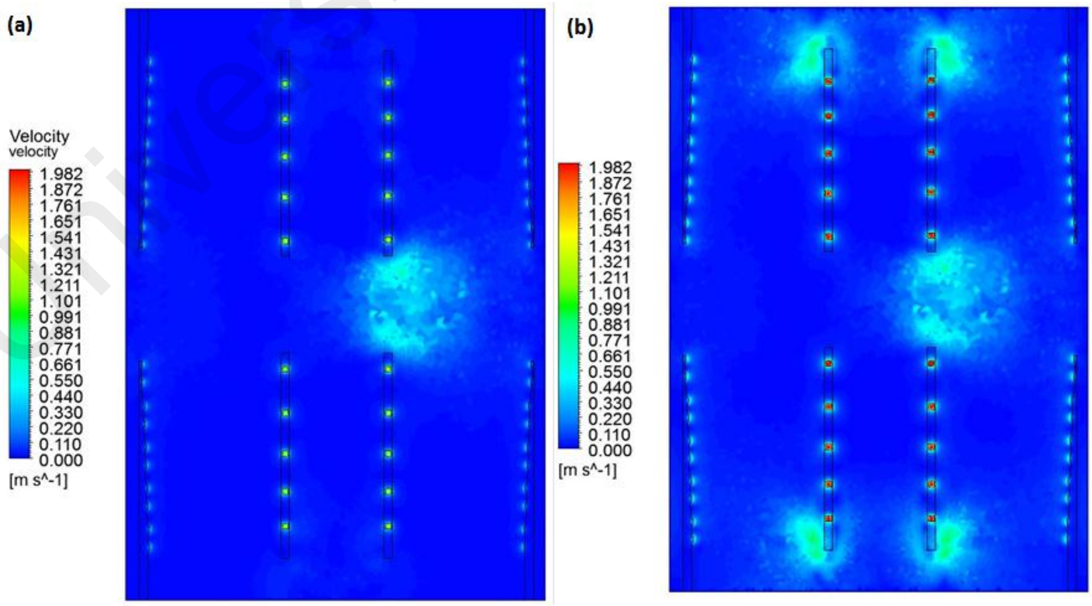


Figure 4.50: Velocity Counter Z=13m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

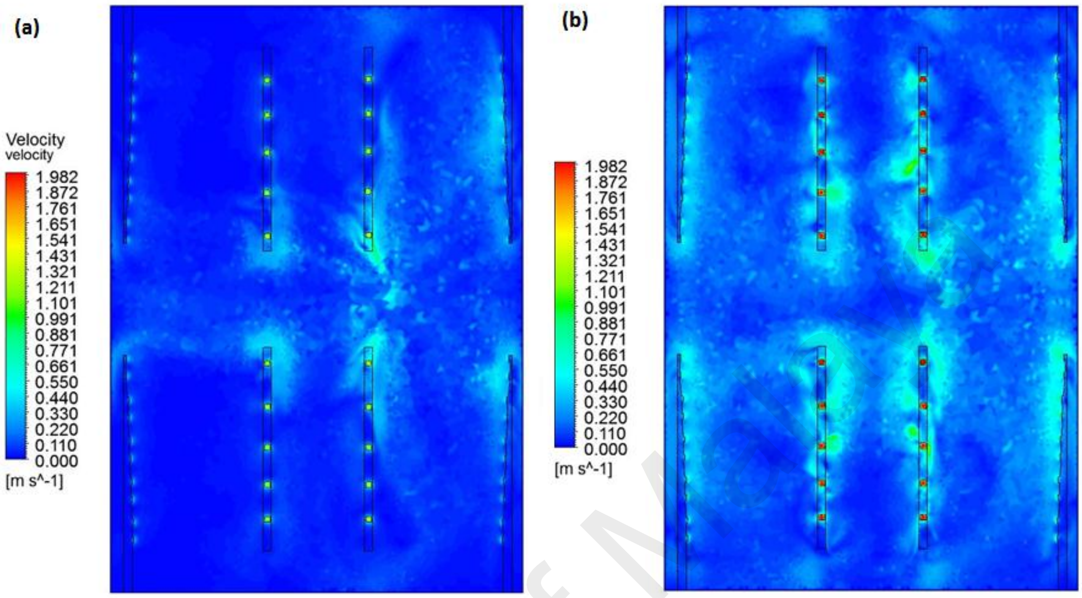


Figure 4.51: Velocity Counter Z=13m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

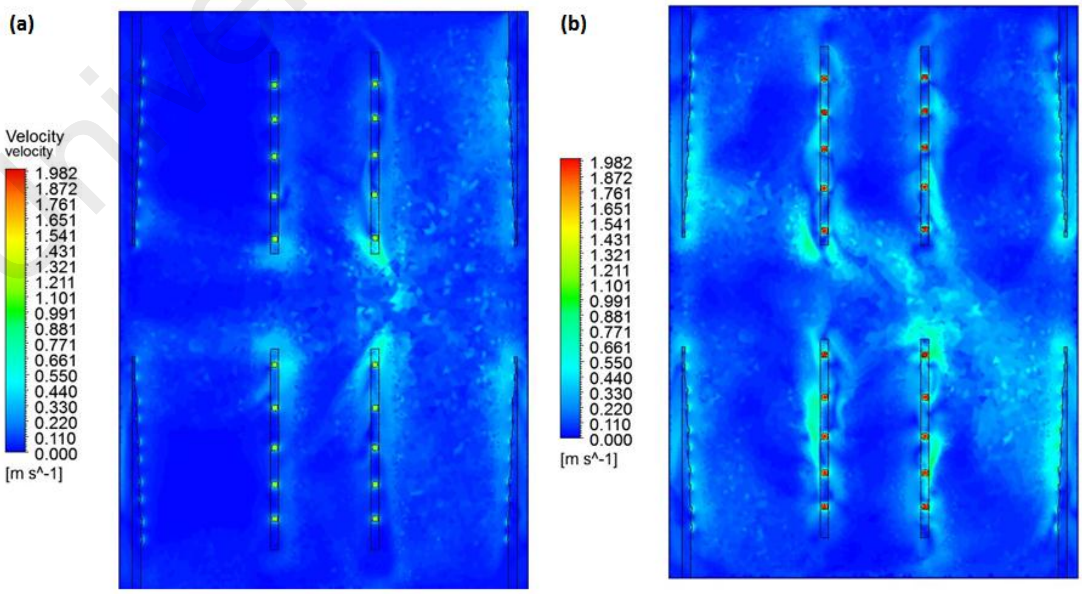


Figure 4.52: Velocity Counter Z=13m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

Figure 4.41 to Figure 4.52 illustrate the airflow interactions with higher height in the Velodrome. The flow path lines in the Velodrome with the jet fans ventilation system are shown in these illustrations. As show that the airflow exiting the jet fans can flow on its straight path and is not deflected by anything in the Velodrome. Figure 4.41 to Figure 4.52 show that interaction between air flows is provided by jet fans in the Velodrome. The velocity contours at various sections of the Velodrome was compared with and without applying jet fans. Figure 4.45 to Figure 4.46 illustrate that the jet fans generate a higher air velocity in region closer to the center of Velodrome. Hence, the jet fans are more effective in eliminating smoke from the lower and central part of the Velodrome compared to when it was without jet fans. As shown in Figure 4.47 to Figure 4.49, jet fans generate high-velocity air- flow along the direction of flow in Velodrome; while for jet fans the high velocity region was put away from the ceiling. Amazingly, even at distances further away from the fan outlet, the peak velocity region provided by the jet fans remained in the central region while for the jet fans was confined at the height of 10 meter from floor. Therefore, there was a noticeable difference between the velocity field generated with and without jet fans. The jet fans removed the smoke such as CO, and provide the people inside Velodrome with fresh air. The performance of these jet fans in preventing the increase of CO concentration is highly importance in Velodrome ventilation. A higher pressure-rise coefficient of jet fans can significantly decrease the number of fans and therefore reduce the investment and operating expense of the Velodrome ventilation system.

4.4.3 Temperature Analysis of Case C

Figure 4.53 to Figure 4.64 show the temperature contour toggling between Jet fans turned “OFF” (a) and Jet fans turned “ON” (b) after subsequent time from 0-90 minutes in height of Z=2-13 meters in Velodrome.

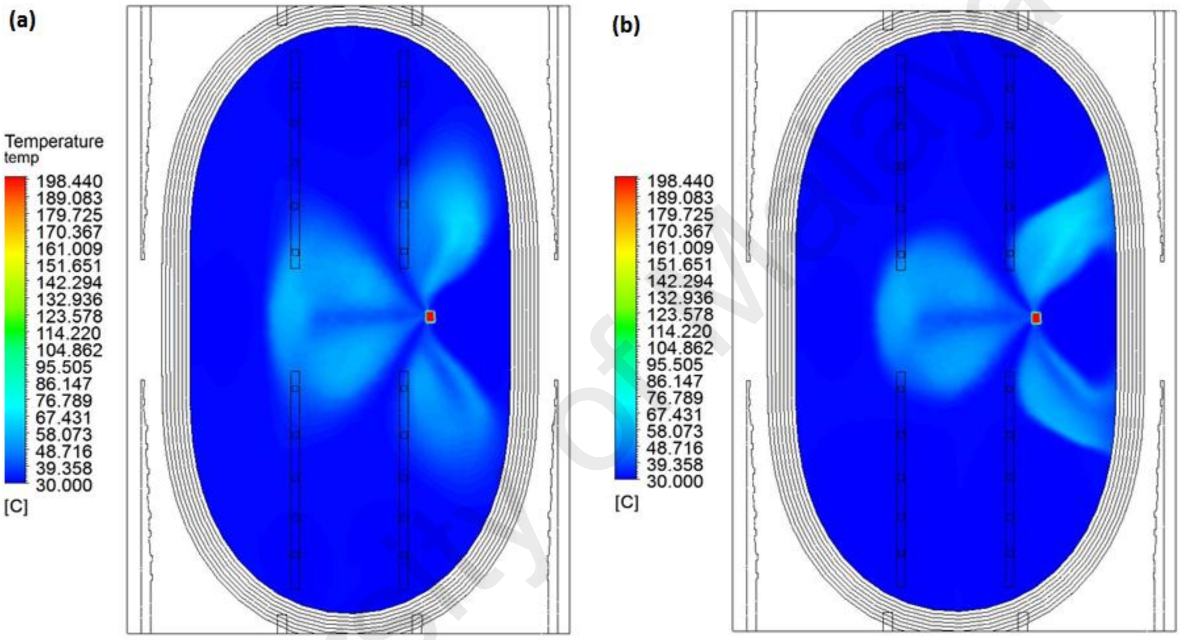


Figure 4.53: Temperature Counter Z=2m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

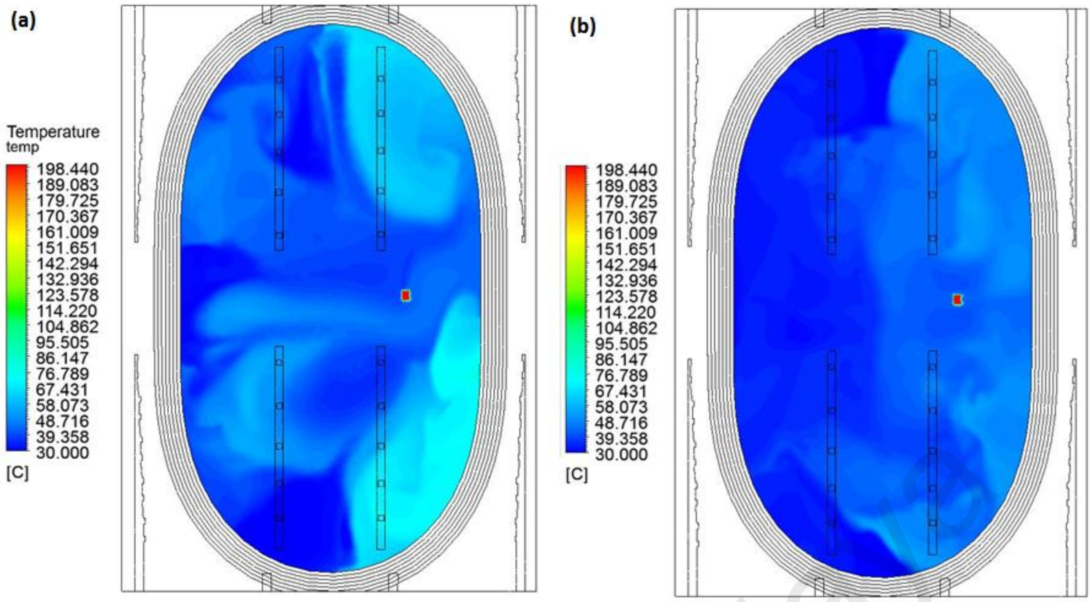


Figure 4.54: Temperature Counter Z=2m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b)

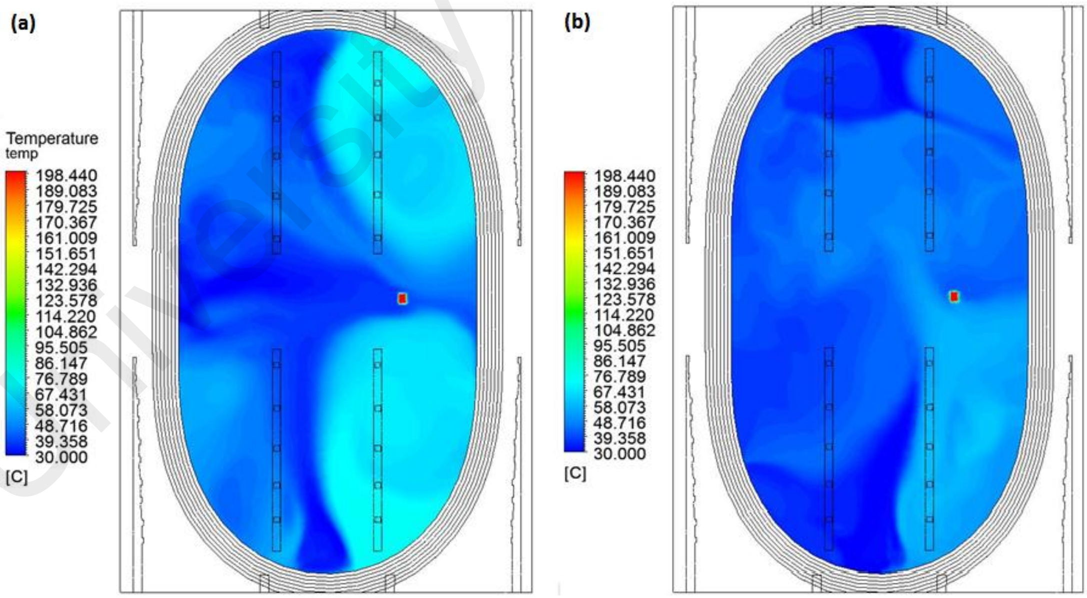


Figure 4.55: Temperature Counter Z=2m, t=90 min, Jet fan "OFF" (a) Jet fan "ON" (b)

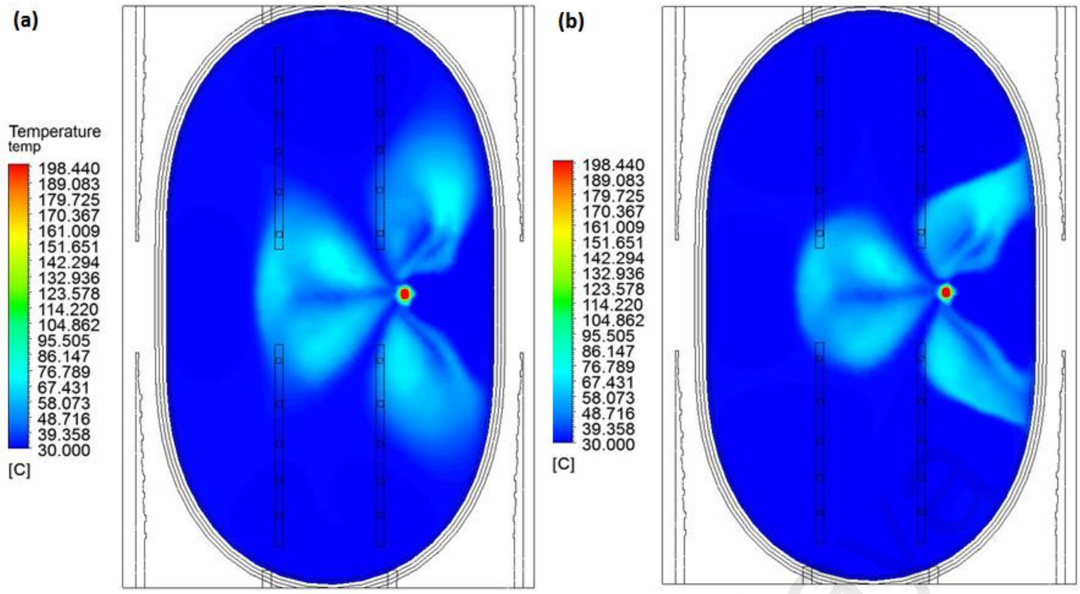


Figure 4.56: Temperature Counter Z=4m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

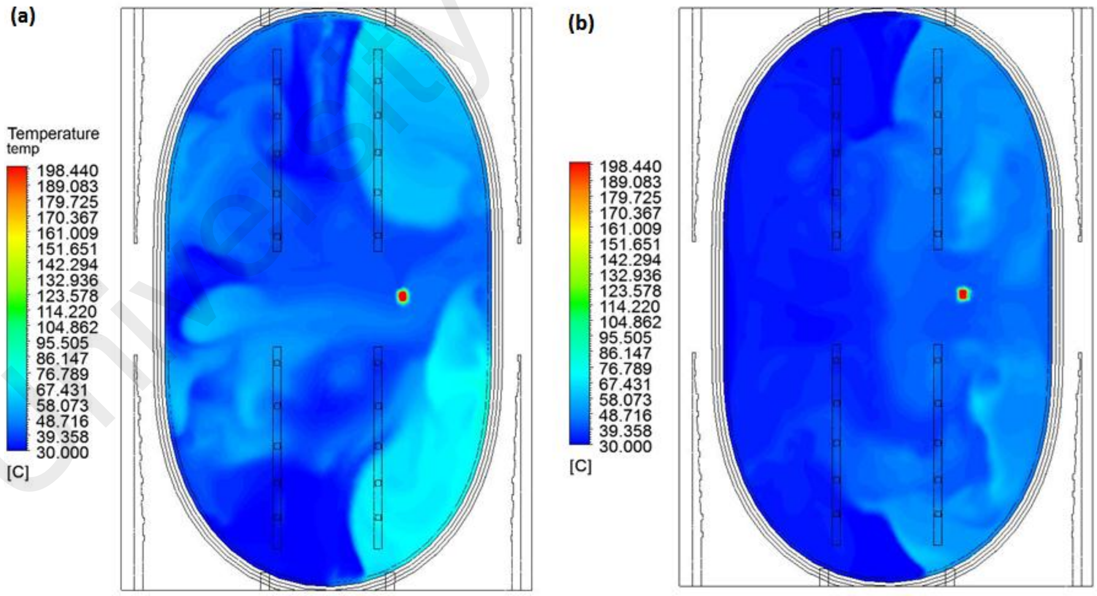


Figure 4.57: Temperature Counter Z=4m, t=50 min, Jet fan “OFF” (a) Jet fan “ON” (b)

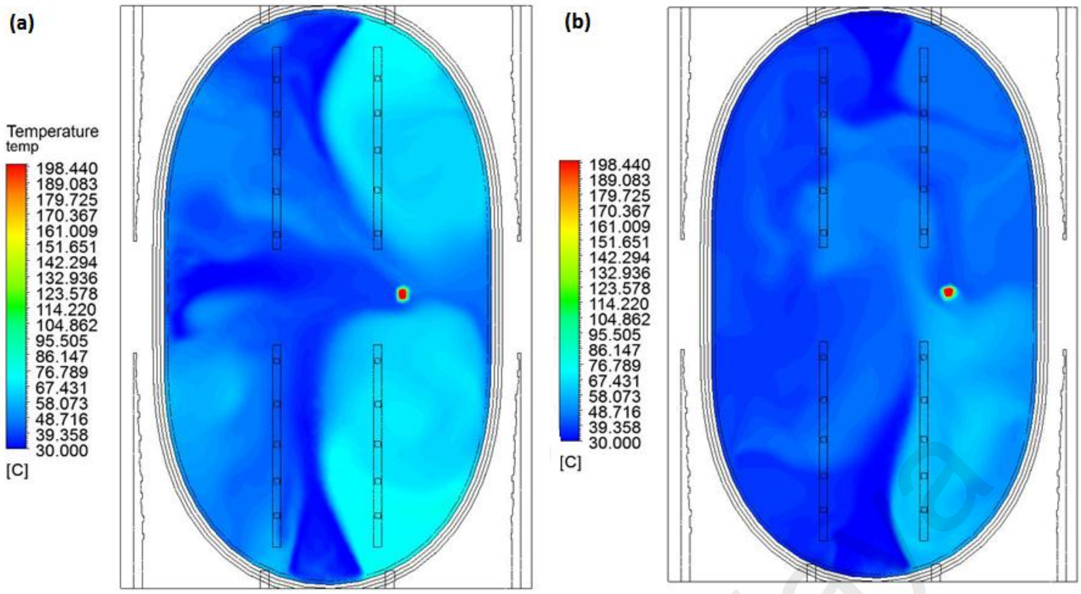


Figure 4.58: Temperature Counter Z=4m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

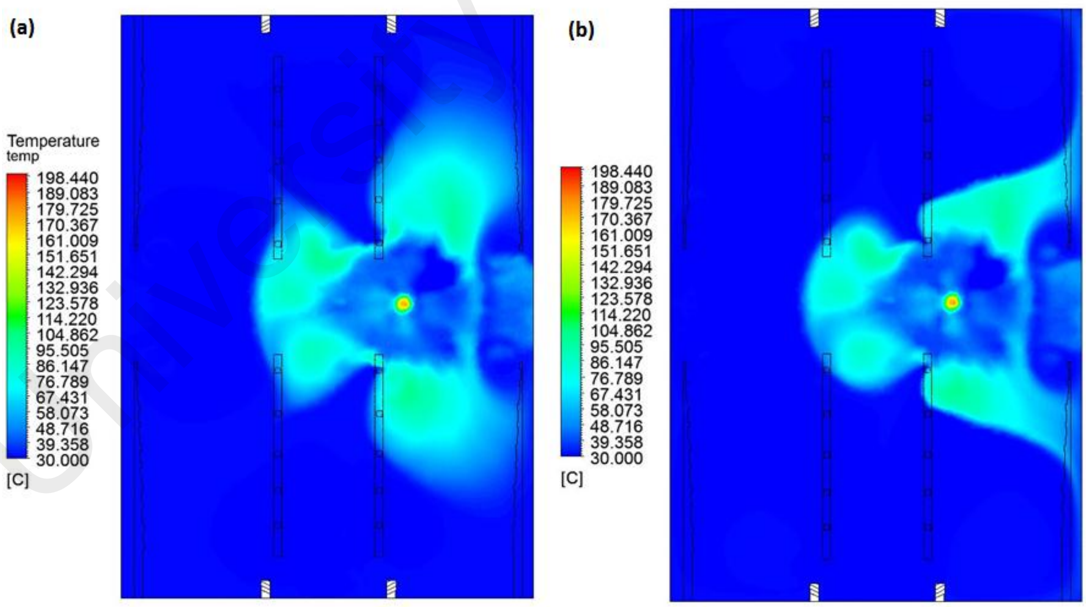


Figure 4.59: Temperature Counter Z=10m, t=10 min, Jet fan “OFF” (a) Jet fan “ON” (b)

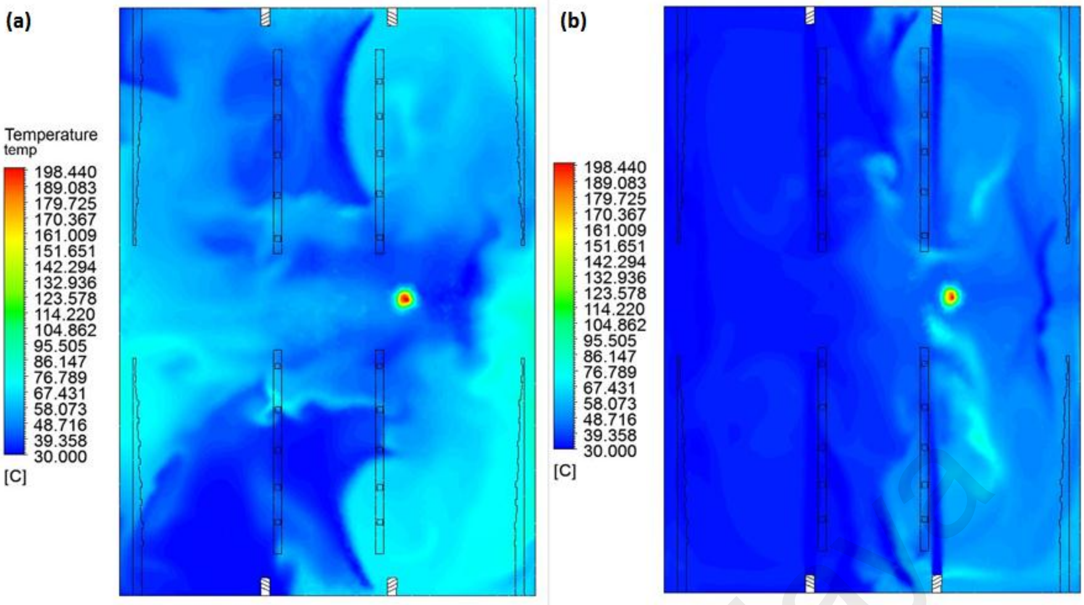


Figure 4.60: Temperature Counter Z=10m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b)

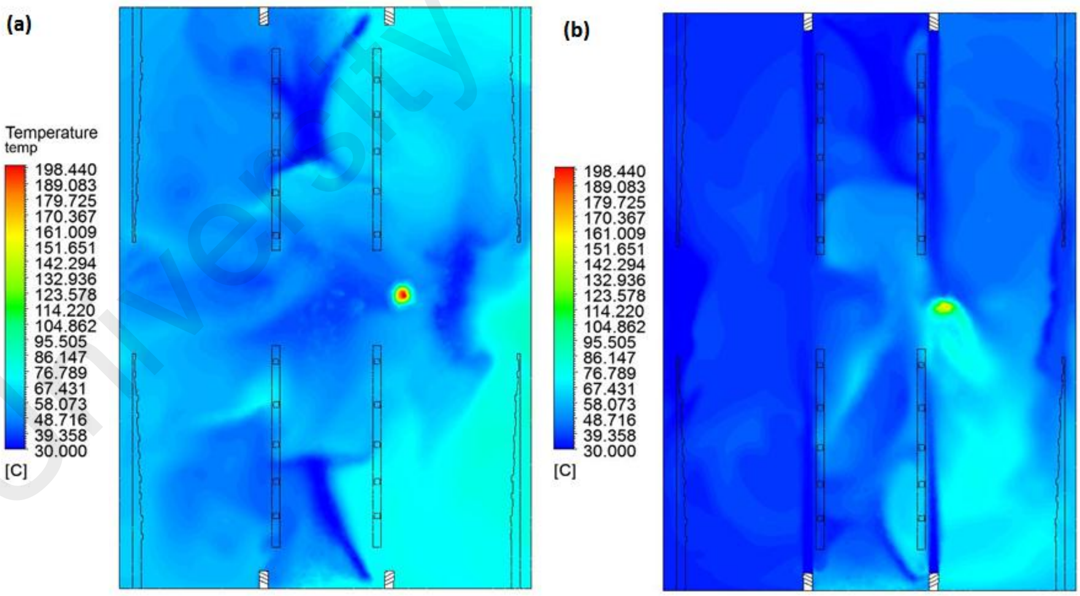


Figure 4.61: Temperature Counter Z=10m, t=90 min, Jet fan "OFF" (a) Jet fan "ON" (b)

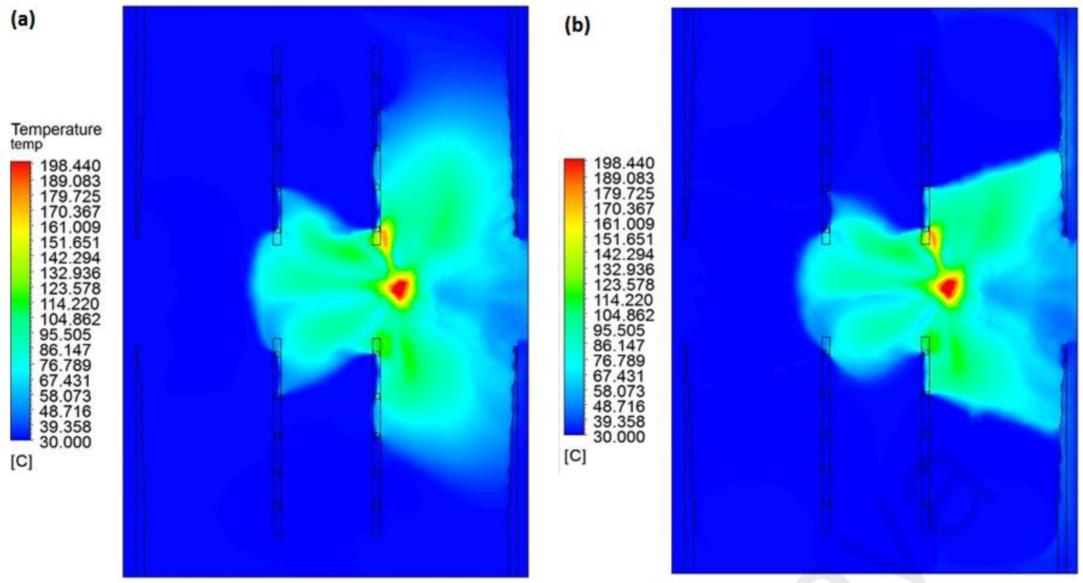


Figure 4.62: Temperature Counter Z=13m, t=10 min, Jet fan "OFF" (a) Jet fan "ON" (b)

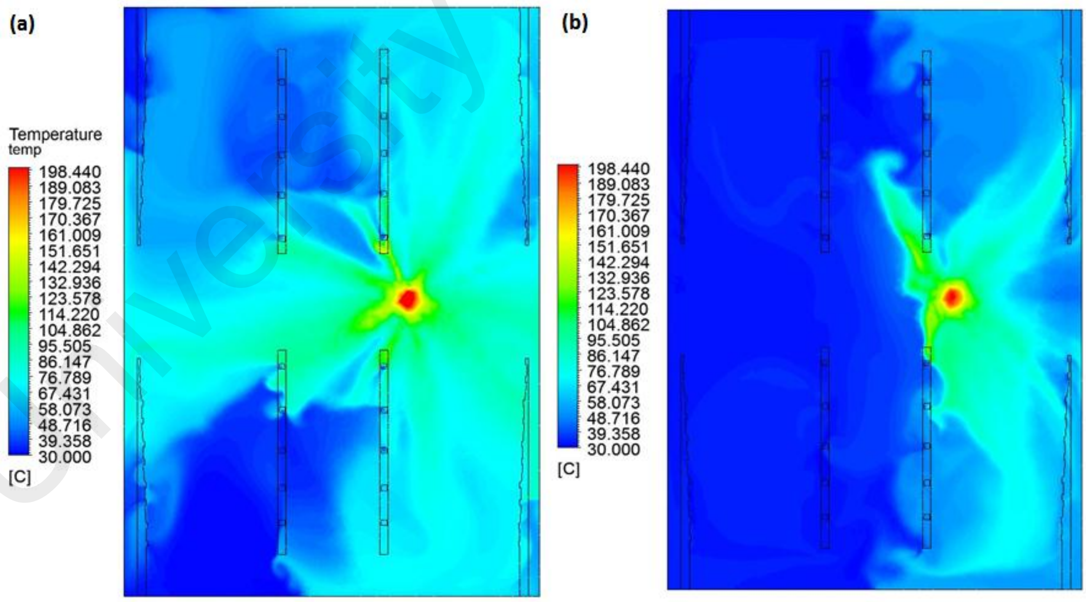


Figure 4.63: Temperature Counter Z=13m, t=50 min, Jet fan "OFF" (a) Jet fan "ON" (b**)**

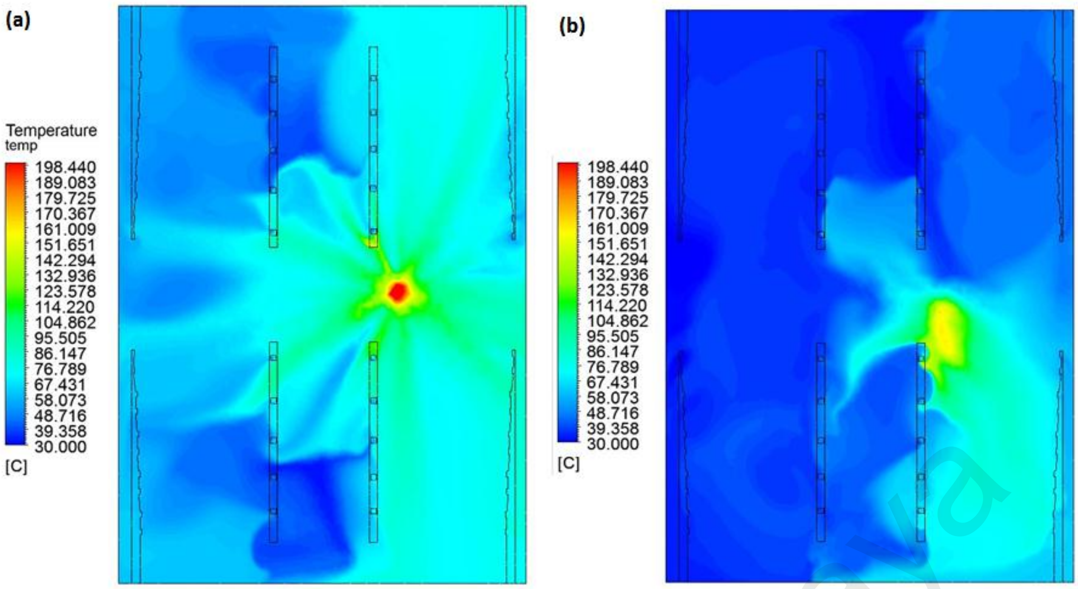


Figure 4.64: Temperature Counter Z=13m, t=90 min, Jet fan “OFF” (a) Jet fan “ON” (b)

Figure 4.53 to Figure 4.64 show simulation result of temperature field at a height of Z=2m to Z=13m. Temperature is dramatically increased at 2 m above the floor and it is depend on time. However, temperature obviously rises downwind of fire source, because jet fans push hot smoke to the west side of fire source. The temperature on the upper level of fan is higher than lower levels, as can be seen the induce effect of jet fans in Figure4.63 and Figure 4.64 is greater. The maximum temperature on the height of Velodrome Z=13 of fire source is between 100^{o}C to 140^{o}C. As jet fan is far from fire source, it only makes a little effect on temperature distribution. As a result, temperature field in Figure 4.53 to Figure 4.64 is similar in general. The ability of a ventilation system to evacuate smoke is limited by its own maximum service temperature.

4.5 Compression and Contrast of Cases

The key element in the fire engineering strategy is the use of a "zoned" smoke management system to protect the Velodrome. This involves the operation of smoke exhaust ventilation and Jet Fan. Increasing the airflow rate in the Velodrome means that the airborne concentration of potentially harmful emission can be decreased. The decrease in concentration is beneficial to people exposed to smoke. However, a situation can arise in which the source is completely removed and smoke put no threat of exposure to people; actuating any fans can draw the existing smoke to the evacuation velodrome. In a Velodrome, smoke management necessitates either direct extraction at the fire location or the generation of a velocity in the Velodrome that is capable of transporting heated gases and the smoke in the desired direction to a point of extraction or discharge from the Velodrome. Analysis with constant mass airflow through the fan results in an overestimate because under constant mass air flow conditions velocity doubles and density is halved when temperature doubles. One of the ways to improve the function of smoke vents is to ensure that pressure in a space is positive. This could be achieved by running the supply ventilation system by displacement only. In the case with the mixing supply ventilation, smoke may possibly spread down to the floor. The layout of the ventilation system could be adjusted. The resulting parameters from simulations are airflow, airflow direction through a smoke vent. This provides greater opportunities for further and more detailed analysis of airflow conditions in a certain studied space.

4.6 Concluding Remarks

In this chapter, the effects of various kinds of fans, and the ventilation airflow field as well as CO concentration inside Velodrome have been analyzed and presented. This findings and result is helpful in the design of ventilation system using natural ventilation such as wind. Simulations were also executed for CO concentration, Velocity and Temperature, to simulate of growing environment in Velodrome is done to design and propose an improved air circulation system that provides a desired average air current speed. The design of air circulation system with perforated air tubes is able to improve the air movement at Velodrome. The four units of air jets on the perforated air tube have helped to expand the coverage of air flow at Velodrome. In next chapter (chapter 5) will discuss about conclusions and lessons learnt and recommendation for future work.

CHAPTER 5: CONCLUSIONS

This chapter discusses on the lesson learnt during this study, overview of the study, and challenges that the researcher had to deal with during the conducting this thesis. This study focuses on the smoke propagation during fire event in Velodrome using the Ducting and Fan in developing fire ventilation system. Research methodology is the major part of fire simulation process. In this work, CFD results have been presented for Velodrome fire configurations.

5.1 Lessons Learnt

The focus of this thesis was on personnel safety in fire events to prevent any casualties that normally is caused by fire emissions specially Carbon monoxide. The smoke will hinder safe evacuation of people and hamper firefighters extinguishing the fire. So, the purpose of this work was to identify effectiveness of ventilation systems in fire safety and to use computational fluid dynamics (CFD) simulation results for analysis of smoke spill with actual experimental results. This work applied skills on study about CFD simulation of smoke to identify a list of requirements for developing a ventilation system. So, studies about CFD simulation of smoke on related journals were done to gather information and the knowledge in requirements of developing a ventilation system and smoke simulation.

Velodrome ventilation systems determine the required air flow for preventing the smoke propagation. These systems determine the required smoke exhaust capacity when a dedicated smoke extraction duct is being considered. The required smoke exhaust flow rate will be determined based on the total air supply through the available makeup airflow openings of the Velodrome (i.e. ducts and inlets) in order to mitigate fire hazards from a fire incident in a Velodrome. The supply air from these openings, which can be calculated based on the mechanical flows along the Velodrome, will be mixed with the fire generated smoke and therefore increase the overall smoke volume that is required to be extracted.

5.2 Conclusion

A case study had been conducted at Velodrome to determined the smoke propagation and air flow pattern during fire condition. The real size model was developed in the CFD open source model developer. Total number of the elements in mesh generation is reached up to four million. For boundary condition, velocity inlet was adopted for fresh air inlets form doors and fresh air ducts. Outflow was selected for exhaust ducting system. About 4 MW fire at tw different positions were simulated at the voladrome, and the time based simulation is done for 90 minutes considering maximum evacuation time. The simulation also investigated the effect of the smoke spill system by simulating with smoke spill system swithed “ON” and “OFF”. The results are presented in 3-dimensional and 2-dimensional planes. The results for velocity, smoke, and temperature distributions are presented at 2m, 4m, 10m, and 13m height from floor level. The results were showed for two fire position senarios to examine the performance of the exhaust system during the fire. Results showed that diffusion of the fire smoke in area of the Velodrome by using the proposed exhaust system is completely under control as reducing amount of CO concentration around 70% when exhaust system is ON or Jet fans are ON in the system. The following results from fire simulation process using CFD model can be drawn:

The use of smoke management system in Velodrome has allowed safe exit by ensuring sufficient separation between the escaping victims or occupants and the smoky gasses released from fires. By providing the improved and effective exhaust system in fire protection measure, the efficient protection of properties can be achieved. The improved exhaust system includes limiting the spread of smoke and reducing its temperature. This study has contributed to the knowledge of exhaust ventilation system. Based on the analysis conducted, it has been discovered that an ordinary ventilation system which operates along with optimum airflow through the roof mounted smoke fan or the optimized area can determine the smoke vents. Generally, efficient smoke evacuation occurs when the exhaust ventilation is working and is able to be shutdown. Hence, the combination of working supply fan along with different combinations of smoke evacuating measures can be considered as an effective measure. Supply ventilation outlets are supposed to be placed in ceilings.

5.3 Recommendations for Future Work

For future work, the key challenges posed to the engineering of systems and environmental performance of the Velodrome are to come out with a more efficient design of the fastest track in the world, to provide a more stable and optimized temperature for cyclists and keep the spectators comfortable throughout.

REFERENCES

Ai, Z. T., & Mak, C. M. (2016). Short-term mechanical ventilation of air-conditioned residential buildings: A general design framework and guidelines. Building and Environment, 108, 12-22.

Beard, A. N. (2016). Major fire spread in a tunnel with water mist: A theoretical model. Tunnelling and Underground Space Technology, 53, 22-32.

Cao, G., Awbi, H., Yao, R., Fan, Y., Sirén, K., Kosonen, R., & Zhang, J. J. (2014). A review of the performance of different ventilation and airflow distribution systems in buildings. Building and Environment, 73, 171-186.

Cao, Z., Wang, Y., Duan, M., & Zhu, H. (2017). Study of the vortex principle for improving the efficiency of an exhaust ventilation system. Energy and Buildings, 142, 39-48.

Chen, Y., Zhou, X., Fu, Z., Zhang, T., Cao, B., & Yang, L. (2016). Vertical temperature distributions in ventilation shafts during a fire. Experimental Thermal and Fluid Science, 79, 118-125.

Chow, W. K. (2015). Performance-based approach to determining fire safety provisions for buildings in the Asia-Oceania regions. Building and Environment, 91, 127-137.

Deckers, X., Haga, S., Sette, B., & Merci, B. (2013). Smoke control in case of fire in a large car park: Full-scale experiments. Fire Safety Journal, 57, 11-21.

Deckers, X., Haga, S., Tilley, N., & Merci, B. (2013). Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations. Fire Safety Journal, 57, 22-34.

Du, T., Yang, D., Peng, S., Liu, Y., & Xiao, Y. (2016). Performance evaluation of longitudinal and transverse ventilation for thermal and smoke control in a looped urban traffic link tunnel. Applied Thermal Engineering, 96, 490-500.

Fan, C. G., Li, X. Y., Mu, Y., Guo, F. Y., & Ji, J. (2017). Smoke movement characteristics under stack effect in a mine laneway fire. Applied Thermal Engineering, 110, 70-79.

Fan, C. G., Ji, J., Wang, W., & Sun, J. H. (2014). Effects of vertical shaft arrangement on natural ventilation performance during tunnel fires. International Journal of Heat and Mass Transfer, 73, 158-169.

Gao, R., Li, A., Hao, X., Lei, W., Zhao, Y., & Deng, B. (2012). Fire-induced smoke control via hybrid ventilation in a huge transit terminal subway station. Energy and Buildings, 45, 280-289.

Heidarinejad, G., Mapar, M., & Pasdarshahri, H. (2016). A comprehensive study of two fire sources in a road tunnel: Considering different arrangement of obstacles. Tunnelling and Underground Space Technology, 59, 91-99.

Hidalgo, J. P., Welch, S., & Torero, J. L. (2015). Performance criteria for the fire safe use of thermal insulation in buildings. Construction and Building Materials, 100, 285-297.

Huang, H., Ooka, R., Chen, H., & Kato, S. (2009). Optimum design for smoke-control system in buildings considering robustness using CFD and Genetic Algorithms. Building and Environment, 44(11), 2218-2227.

Hull, T. R., Brein, D., & Stec, A. A. (2016). Quantification of toxic hazard from fires in buildings. Journal of Building Engineering, 8, 313-318.

Ji, J., Wan, H., Gao, Z., Fu, Y., Sun, J., Zhang, Y., ... & Hostikka, S. (2016). Experimental study on flame merging behaviors from two pool fires along the longitudinal centerline of model tunnel with natural ventilation. Combustion and Flame, 173, 307-318.

Kashef, A., Yuan, Z., & Lei, B. (2013). Ceiling temperature distribution and smoke diffusion in tunnel fires with natural ventilation. Fire Safety Journal, 62, 249-255.

Li, S. C., Huang, D. F., Meng, N., Chen, L. F., & Hu, L. H. (2017). Smoke spread velocity along a corridor induced by an adjacent compartment fire with outdoor wind. Applied Thermal Engineering, 111, 420-430.

Li, L. J., Tang, F., Dong, M. S., & Tao, C. F. (2016). Effect of ceiling extraction system on the smoke thermal stratification in the longitudinal ventilation tunnel. Applied Thermal Engineering, 109, 312-317.

Li, L., Li, S., Wang, X., & Zhang, H. (2012). Fire-induced flow temperature along tunnels with longitudinal ventilation. Tunnelling and Underground Space Technology, 32, 44-51.

Liu, Y., & Cassady, S. (2014). A modified critical velocity for road tunnel fire smoke management with dedicated smoke extraction configuration. Case Studies in Fire Safety, 2, 16-27.

Mei, F., Tang, F., Ling, X., & Yu, J. (2017). Evolution characteristics of fire smoke layer thickness in a mechanical ventilation tunnel with multiple point extraction. Applied Thermal Engineering, 111, 248-256.

Merci, B., & Shipp, M. (2013). Smoke and heat control for fires in large car parks: Lessons learnt from research?. Fire safety journal, 57, 3-10.

Meroney, R. N., Hill, D. W., Derickson, R., Stroup, J., Weber, K., & Garrett, P. (2015). CFD Simulation of ventilation and smoke movement in a large military firing range. Journal of Wind Engineering and Industrial Aerodynamics, 136, 12-22.

Mouilleau, Y., & Champassith, A. (2009). CFD simulations of atmospheric gas dispersion using the Fire Dynamics Simulator (FDS). Journal of Loss Prevention in the Process Industries, 22(3), 316-323.

Nomura, M., & Hiyama, K. (2017). A review: Natural ventilation performance of office buildings in Japan. Renewable and Sustainable Energy Reviews, 74, 746-754.

NFPA, NFPA 92 Standard for Smoke Control Systems 2015 Edition, 2015.

Seike, M., Kawabata, N., & Hasegawa, M. (2017). Quantitative assessment method for road tunnel fire safety: Development of an evacuation simulation method using CFD-derived smoke behavior. Safety science, 94, 116-127.

Su, C., & Yao, C. (2016). Performance measurement of a smoke extraction system for buildings in full-scale hot smoke test. Measurement, 93, 340-350.

Tang, F., Cao, Z. L., Chen, Q., Meng, N., Wang, Q., & Fan, C. G. (2017). Effect of blockage-heat source distance on maximum temperature of buoyancy-induced smoke flow beneath ceiling in a longitudinal ventilated tunnel. International Journal of Heat and Mass Transfer, 109, 683-688.

Tilley, N., Rauwoens, P., & Merci, B. (2011). Verification of the accuracy of CFD simulations in small-scale tunnel and atrium fire configurations. Fire Safety Journal, 46(4), 186-193.

Tominaga, Y., Akabayashi, S. I., Kitahara, T., & Arinami, Y. (2015). Air flow around isolated gable-roof buildings with different roof pitches: Wind tunnel experiments and CFD simulations. Building and Environment, 84, 204-213.

Ura, F., Kawabata, N., & Tanaka, F. (2014). Characteristics of smoke extraction by natural ventilation during a fire in a shallow urban road tunnel with roof openings. Fire Safety Journal, 67, 96-106.

Vaux, S., & Prétrel, H. (2013). Relative effects of inertia and buoyancy on smoke propagation in confined and forced ventilated enclosure fire scenarios. Fire Safety Journal, 62, 206-220.

Wang, H. Y. (2009). Prediction of soot and carbon monoxide production in a ventilated tunnel fire by using a computer simulation. Fire safety journal, 44(3), 394-406.

Wang, K., Jiang, S., Ma, X., Wu, Z., Shao, H., Zhang, W., & Cui, C. (2016). Information fusion of plume control and personnel escape during the emergency rescue of external-caused fire in a coal mine. Process Safety and Environmental Protection, 103, 46-59.

Wang, S., Li, X., & Wang, D. (2016). Mining-induced void distribution and application in the hydro-thermal investigation and control of an underground coal fire: A case study. Process Safety and Environmental Protection, 102, 734-756.

Wang, F., & Wang, M. (2016). A computational study on effects of fire location on smoke movement in a road tunnel. Tunnelling and Underground Space Technology, 51, 405-413.

Węgrzyński, W., & Krajewski, G. (2017). Influence of wind on natural smoke and heat exhaust system performance in fire conditions. Journal of Wind Engineering and Industrial Aerodynamics, 164, 44-53.

Weng, M. C., Yu, L. X., Liu, F., & Nielsen, P. V. (2014). Full-scale experiment and CFD simulation on smoke movement and smoke control in a metro tunnel with one opening portal. Tunnelling and Underground Space Technology, 42, 96-104.

Willstrand, O., Brandt, J., & Svensson, R. (2016). Detection of fires in the toilet compartment and driver sleeping compartment of buses and coaches—Installation considerations based on full scale tests. Case Studies in Fire Safety, 5, 1-10.

Yang, D., & Li, P. (2015). Dimensionless design approach, applicability and energy performance of stack-based hybrid ventilation for multi-story buildings. Energy, 93, 128-140.

Yao, Y., Cheng, X., Zhang, S., Zhu, K., Shi, L., & Zhang, H. (2016). Smoke back-layering flow length in longitudinal ventilated tunnel fires with vertical shaft in the upstream. Applied Thermal Engineering, 107, 738-746.

Yi, L., Wei, R., Peng, J., Ni, T., Xu, Z., & Wu, D. (2015). Experimental study on heat exhaust coefficient of transversal smoke extraction system in tunnel under fire. Tunnelling and Underground Space Technology, 49, 268-278.

Yuan, L., Zhou, L., & Smith, A. C. (2016). Modeling carbon monoxide spread in underground mine fires. Applied thermal engineering, 100, 1319-1326.Zhang, S., Cheng, X., Yao, Y., Zhu, K., Li, K., Lu, S. & Zhang, H. (2016). An experimental investigation on blockage effect of metro train on the smoke back-layering in subway tunnel fires. Applied Thermal Engineering, 99, 214-223.

Zhang, S., Wu, Z., Zhang, R., & Kang, J. (2012). Dynamic numerical simulation of coal mine fire for escape capsule installation. Safety science, 50(4), 600-606.

Zhang, Y., Kacira, M., & An, L. (2016). A CFD study on improving air flow uniformity in indoor plant factory system. biosystems engineering, 147, 193-205.

Zhang, S., Yao, Y., Zhu, K., Li, K., Zhang, R., Lu, S., & Cheng, X. (2016). Prediction of smoke back-layering length under different longitudinal ventilations in the subway tunnel with metro train. Tunnelling and Underground Space Technology, 53, 13-21.

Zhong, W., Lv, J., Li, Z., & Liang, T. (2013). A study of bifurcation flow of fire smoke in tunnel with longitudinal ventilation. International Journal of Heat and Mass Transfer, 67, 829-835.