FLOW SIMULATION ANALYSIS OF MAJU CABLE TUNNEL VENTILATION SYSTEM

AHMAD HAFIZ BIN JAFARUL TAREK

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2019

FLOW SIMULATION ANALYSIS OF MAJU CABLE TUNNEL VENTILATION SYSTEM

AHMAD HAFIZ BIN JAFARUL TAREK

RESEARCH REPORT SUBMITTED IN FULFILMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTER OF MECHANICAL ENGINEERING

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2019

UNIVERSITY OF MALAYA

ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: Ahmad Hafiz Bin Jafarul Tarek

Matric No: KQK18

Name of Degree: Master of Mechanical Engineering

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

Flow Simulation Analysis of Maju Cable Tunnel Ventilation System

Field of Study: Mechanical Engineering – Computational Fluid Dynamics

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: 11 September 2019

Subscribed and solemnly declared before,

Witness's Signature   Date: 11 September 2019

Name: Prof Madya Ir Dr. Nik Nazri bin Nik Ghazali

Designation: Supervisor

FLOW SIMULATION ANALYSIS OF MAJU CABLE TUNNEL VENTILATION SYSTEM

ABSTRACT

One of Maju Utility Tunnel functions is to accommodate the cable buried underground. Hence, tunnel ventilation is required to prevent the cable from malfunctioning due to lack of air throughout the tunnel. This research report shows the types of tunnel ventilation (natural vs mechanical ventilation) where the velocity variation plays its role to achieve thermal comfort for workers. Natural ventilation cools down the tunnel using the surrounding air (Case 1), and mechanical ventilation setup requires auxiliary components such as inlet pump and exhaust system for cooling down (Case 2 and Case 3). High velocity shows improvement for temperature distribution. This is proven by CFD analysis as it can achieve the lowest temperature, which is 33.8˚C. At the same time, we determine Air change per hour (ACPH) of the tunnel. ACPH indicates how much air can enter and exit the tunnel, which means the higher the velocity, the higher the exchanged air is. The range between 20 - 40 of ACPH is used as a benchmark on ideal condition. The volume and CFM are also set as the parameters for ACPH calculations. The difference between Case 2 and Case 3 is the addition of the exhaust system. The capability of the mechanical ventilation pumping power to cool down the tunnel is determined via formula calculation.

Keywords: Ventilation, Air Change per Hour, Thermal comfort, CFD

ANALISIS SIMULASI PENYALURAN TEROWONG KABEL MAJU SISTEM PENYAHUDARAAN

ABSTRAK

Penggunaan terowong utiliti Maju adalah untuk menampung kabel yang tertanam di bawah tanah. Oleh yang demikian, penyahudaraan terowong diperlukan untuk mengelak kabel daripada rosak disebabkan kekurangan udara di sepanjang terowong. Laporan ini menunjukkan setiap jenis penyahudaraan terowong (penyahudaraan semulajadi vs penyahudaraan mekanikal) di mana variasi halaju udara memainkan peranan untuk memastikan keselesaan haba pekerja. Penyahudaraan semulajadi menyejukkan terowong menggunakan udara di sekeliling (Kes 1). Penyediaan penyahudaraan mekanikal memerlukan komponen tambahan seperti pam saluran masuk dan sistem saluran keluar untuk penyejukan (Kes 2 dan Kes 3). Halaju yang tinggi menunjukkan penambahbaikkan dalam penyebaran suhu. Ini dibuktikan melalui analisis CFD kerana ia mampu mencapai suhu paling rendah iaitu 33.8˚C. Pada masa yang sama, kadar perubahan udara sejam (ACPH) di dalam terowong ditentukan. ACPH menunjukkan berapa banyak udara masuk dan keluar terowong, bermaksud semakin tinggi halaju semakin tinggi perubahan udara. Dalam julat 20 - 40 ACPH digunakan sebagai batas di keadaan ideal. Isipadu dan CFM adalah pembolehubah untuk pengiraan ACPH. Perbezaan antara kes 2 dan kes 3 adalah penambahan system saluran keluar. Kebolehan kuasa pam penyahudaraan mekanikal menyejukan terowong didapati melalui pengiraan formula.

Kata Kunci: Penyahudaraan, Kadar perubahan udara sejam, keselesaan haba, CFD

ACKNOWLEDGEMENTS

A special thanks to Faculty Engineering of University Malaya for giving an opportunity to further study on Mechanical Engineering knowledge. I learned a lot regarding CFD simulation under the tutelage of Assoc. Prof. Ir. Dr Nik Nazri. At the same time, a token appreciation on my wife Nur Aina Shazlin, family and colleagues to help me discover the importance of self-discipline and consistency to perform at a higher standard. This research improves my ability to do research and develop my interest to pursue PhD near future.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS AND ABBREVIATIONS

CFD	:	Computational Fluid Dynamics
CAD	:	Computer Aided Drafting
RANS	:	Reynold's Averaged Navier-Stokes
PDE	:	Partial Differences Equation
ASHRAE	:	American Society of Heating, Refrigerating and Air-Conditioning Engineers
TEPCO	:	Tokyo Electric Power Company
HRR	:	Heat Release Rate
SMART	:	Stormwater Management and Road Tunnel

CHAPTER 1: INTRODUCTION

1.1 Introduction

1.1.1 Utility Tunnel

Utility tunnel consists of cables, fibre optics, water supply, and they are buried underground. This tunnel is useful for city planning where all cables and fibre optics are managed or serviced underground to reduce congestion above the ground. The utility tunnel underground concept aligns with rapid development that demands more space with clean environment. A renowned example of a utility tunnel is Tokyo Underground Cable Tunnel. TEPCO is an organization based in Japan responsible for planning, researching, cost analyzing and executing the construction of utility tunnel.

Figure 1.1: Tokyo Cable Tunnel Underground (Retrieved from http://www.tepco.co.jp/en/corpinfo/consultant/facilities/2-cons-t-e.html)

Meanwhile, the famous tunnel in Malaysia is SMART Tunnel where it is utilized as roadway and flash-flooding prevention tunnel. SMART Tunnel uses axial fan as a ventilation method which means it uses mechanical ventilation concept to ensure air supplied is sufficient for road user. Similar with this research report, the concept of mechanical and natural ventilation is being compared for Maju Tunnel to prove that mechanical ventilation is the best design choice.

Research for cable tunnel analysis study by Xu et al. (2019) highlighted the importance of cable tunnel current rating including its positioning and layout by using simulation of CFD. It indicates temperature distribution in cable tunnel; the severity of high temperature will affect the longevity of cable life.

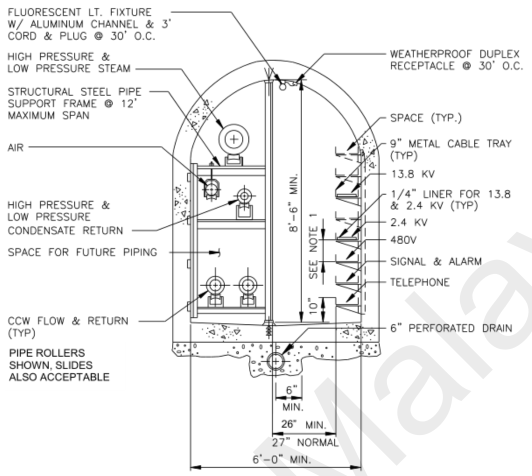


Figure 1.2 : Sample Utility tunnel layout (Retrieved from https://facilities.uw.edu/files/media/fsdg-02-u-utility-tunnels-trenches.pdf)

1.1.2 Maju Tunnel system

Located in Selangor, Maju Tunnel is 163m of length consists of 4 cables cover wall (Figure 1.3). The function of this tunnel is to secure the cables deep underground so it can provide electricity without interference. The tunnel uses mechanical ventilation where it consists of two fans and one exhaust (Figure 1.4). It is operated in a longitudinal airflow system. Case 2 is reflected as the original design of Maju Tunnel system.

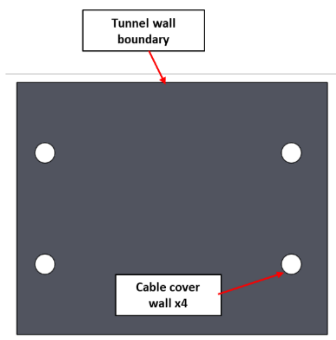


Figure 1.3 : Maju cable tunnel main component

Figure 1.4 : Maju tunnel longitudinal air flow system

1.2 Problem Statement

A tunnel requires a proper ventilation system to reduce risk, for instance, fire hazard. Besides cooling down the cable temperature, the air intake is also necessary to enable workers to perform their tasks efficiently. One of the most important aspects of ventilation that is often overlooked is thermal comfort. Thermal comfort is a condition where humans satisfied with the thermal environment. The thermal comfort affected by three factors which are airspeed, humidity, and temperature. The workers must be able to reach the optimum number of thermal comforts while working in the tunnel. A working temperature that is too high can cause exhaustion and when it is too low, it will cause low productivity as the worker unable to move around freely (Seppänen et al., 2006). Essentially, the standard temperature for the thermal comfort of the tunnel is about 34.5˚C (Duris et al., 2017).

Another design criterion of tunnel ventilation is the amount of fresh air intake. The fresh air intake will help to maintain the tunnel’s comfort as it circulates to replace old air. The fresh air intake quality evaluated based on Air Change per Hour (ACPH). ACPH should be in the range of 20-40 for fresh air intake level of the tunnel (ToolBox, 2005). Furthermore, when ACPH value is lower than the limit threshold, this will cause uneasiness of breathing, dizziness, and anxiety to workers.

Mechanical ventilation provided by air pumps. The air pumps with optimum efficiency can help to supply air required to cool the tunnel. Without sufficient power, airflow unable to circulate inside the tunnel efficiently and this will shorten the distance to cool down. Subsequently, the worker is forced to deal with more heat and reduced its productivity. Based on research on tunnel ventilation, the most appropriate power is 186.5kW for cooling (Goswami & Biseli, 1993).

1.3 Objective

1. To analyze the appropriate temperature required in the cable tunnel.

2. To determine the appropriate Air Change per Hour (ACPH) in the cable tunnel.

3. To determine the pumping power required to cool down the cable tunnel.

1.4 Scope of Project

In order to achieve an appropriate temperature in the cable tunnel, CFD simulation will determine the temperature distribution along the tunnel by manipulating the variation of velocity. The first case represents natural ventilation where inlet velocity entered the tunnel without any auxiliary mechanism, the second case is based on the existing Maju Tunnel design where mechanical ventilation will provide one auxiliary inlet and one exhaust to cool down temperature along the tunnel. Lastly, an improvement of mechanical ventilation by adding an extra auxiliary exhaust for the third case. Graphs between these cases plotted and analyzed for comparison.

Air change per hour is determined via the formula of ACPH equal to 60 times volume air flow rate divided by the volume of the tunnel for every case. The volume air flow rate is in cubic feet per minute or commonly known as CFM. The range of ACPH is influenced by changes in velocity value and volume of the tunnel. Thus, the most optimum value of ACPH examined and if it is appropriate for tunnel ventilation.

Pumping power for cooling down tunnel can be determined by applying the formula of power equal to volumetric airflow times differential pump pressure overtime for every case. The higher power pump means more airflow is able to circulate the tunnel. All these results will be discussed to determine if the air pump power required is sufficient.

CHAPTER 2: LITERATURE REVIEW

2.1.1 Computational Fluid Dynamics

CFD is a simulation of fluid analysis modeling and numerical approach to solve. Therefore, this research report utilizes air as a fluid problem where it is a fundamental issue for the CFD problem. Understand the concept of the Navier-stokes equation serves as the basis for fluid motion. In order to understand the Navier-stokes, it is important to understand the concept of continuity and momentum equation.

The continuity equation fulfills the law of mass conservation which provides the source term in governing equation. In simpler words, it deals with mass flows out of a control volume is equal to time decrease inside mass at a fixed space (Anderson, 1995). For Momentum equation, it follows the fundamental of Newton's second law of motion where F = ma which indicates moving fluid of an element. The summation of forces in a moving particle is equal to the rate of increase momentum of a fluid particle. This moving fluid particle has two forces which are body force and surface force (pressure, viscous, gravity force).

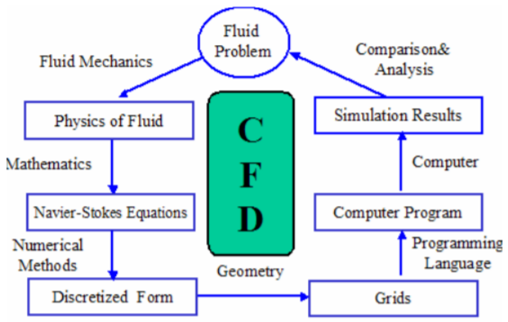


Figure 2.1 : CFD Process flowchart (Zuo, 2005)

2.1.1.1 Governing Equation

The governing equation as following:

(a) ***Continuity Equation***

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0$$

(b) ***Momentum Equation***

$$x \text{ component :} \qquad \rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

$$y \text{ component :} \qquad \rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

$$z \text{ component :} \qquad \rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$

Using non-conservation form due to assumption the control volume is not moving inside tunnel geometry with fluid (air) moving through and the air particle will remain in the control volume (Anderson, 1995).

2.1.1.2 Turbulence Modelling - K-ε Equation

Another important CFD theory is turbulence modeling. RANS based turbulence model is divided into three categories which are linear eddy viscosity, non-linear eddy viscosity, and Reynold's stress model. The purpose of RANS is to calculate Reynold's stresses found inside turbulence flow. The linear eddy viscosity is needed to evaluate the turbulence flow simulation such as k-ε modeling. Then, k represents the kinetic energy and ε is a rate of dissipation turbulence energy. Both describe transportation PDE.

Kinetic energy, k:

$$\frac{\partial(\rho k)}{\partial t} + \operatorname{div}(\rho k \mathbf{U}) = \operatorname{div}\left[\frac{\mu_t}{\sigma_k} \operatorname{grad} k\right] + 2\mu_t S_{ij} \cdot S_{ij} - \rho\varepsilon$$

Rate of dissipation turbulence energy, ε:

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \operatorname{div}(\rho \varepsilon \mathbf{U}) = \operatorname{div}\left[\frac{\mu_t}{\sigma_\varepsilon} \operatorname{grad} \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t S_{ij} \cdot S_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

This can be explained as in sentence form as Rate of change, k or ε + Transport of k or ε by convection is = Transport of k or ε by diffusion + Rate of production, k or ε - Rate of destruction, k or ε. The equation has five adjustable constants which are C_{μ} = 0.09, σ_k = 1.00, σ_{ε} = 1.30, $C_{1\varepsilon}$ = 1.44 and $C_{2\varepsilon}$ = 1.92. (Versteeg & Malalasekera, 2007).

k-ε model is common among the industry as it predicts temperature from far of wall boundary condition with small pressure gradients as compared to k-ω which predicts near the wall boundary condition. It is an important factor of the turbulence model to be used as the viscous model for this paper. CFD is used as a validation for flow physics aspects where it can compare the simulation result and experimental data of wind tunnel airflow. Further complex phenomenon such as boundary layer flow separation, vortex interaction can be simulated using CFD. It shows several errors such as convergence issue, grid refinement plays a critical role. (Favier, 2010)

2.2 Thermal Comfort

Thermal comfort is a condition where people is comfortable with the surrounding temperature. This subsequently will affect people's productivity and health. Therefore, thermal comfort is an important factor to be considered when designing an underground utility tunnel as workers spend most of their time working inside. Temperature, humidity, air movement are several factors affecting thermal comfort. We can relate the ventilation with thermal comfort as it deals with temperature, air velocity and humidity.

2.2.1 Past Study in Thermal Comfort

An indoor thermal comfort has been discussed on the optimal temperature using CFD theory and Airpak simulation. This paper determined the influence of wind frequency, wind velocity and temperature inside the house as a boundary condition. (Fengying et al., 2011)

Another study on thermal comfort has been done on a 2-storey residential house in Malaysia where the main parameter is air velocity, air temperature, and air humidity by using CFD software and to be compared with ASHRAE standard. The result showed it is poor thermal comfort by natural ventilation and does not meet ASHRAE standard. However, the addition fan as mechanical ventilation improving the condition inside the house especially during the critical thermal comfort period which is afternoon. (Malek et al., 2015)

2.3 Ventilation Tunnel System

Ventilation is a necessity for a system to react with heat incoming and outgoing. Tunnel construction is affected by the factor of heat, compressed air and dust trapped in the underground, and this will lead to safety and comfort factor for workers. Proper ventilation will provide better thermal comfort and will contribute to energy saving. For example, adequate tunnel ventilation design for a specific area by the addition of airflow needed to reach the best thermal comfort for the worker.

2.3.1 Ventilation of tunnel construction

During the construction of the tunnel, 200 to 500 cubic feet of fresh air supplied to workers. However, this fresh air is compressed and likely to be contaminated with oil and dust. It is necessary to remove dirt and fumes after the explosion to prevent hazard upon health. In summary, the ventilation of tunnel construction is affected by three factors (Arjun, 2017) :

i Length of tunnel and size

As the tunnel distance increased, more fresh air needed to travel throughout tunnel

ii Condition, temperature differences and humidity

Humidity, temperature differences and condition inside the tunnel are the factors for the workers to able to perform the task efficiently.

iii Explosives and the frequency of blasting

When explosives and frequency of blasting used, its radius changed the landscape of tunnel, and subsequently, dirt and moisture are the byproducts of blasts.

According to Sadokierski and Thiffeault (2007), they conducted a heat transfer analysis of turbulent air towards open tunnel wall as air moving concentric core by an analytical approach. The train used as a concentric core. Heat transfer coefficient increased as the train pass through the tunnel; high shear stress introduced.

2.3.2 Ventilation airflow system

The air intake or outgoing to the tunnel must follow a specific pathway to ensure the ventilation functioned well. There are 2 types of airflow system inside the tunnel, which are:

(a) ***Longitudinal Airflow system***

For longitudinal airflow, the tunnel airflow moves the pollutants and dust first before fresh air coming in at the beginning of the tunnel. At the end of the tunnel, the pollutants and dust discharged.

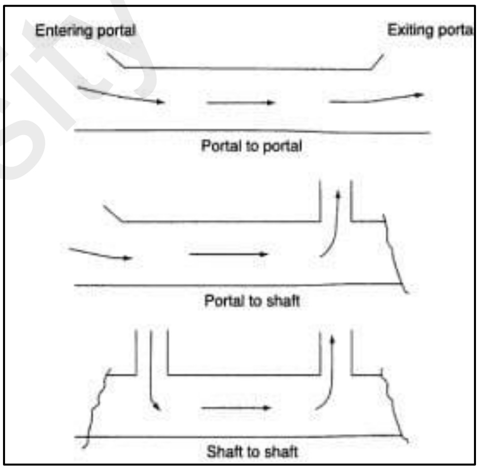


Figure 2.2 : Longitudinal Ventilation airflow movement(Arjun, 2017)

One of the researches on longitudinal ventilation where ambient pressure on smoke back layering, the ventilation velocity, and HRR are the focus. The effect of smoke and fire control inside tunnel posed a threat to human life when smoke trapped inside the tunnel. The airflow gathered shows the movement of smoke back layering from fire source at different critical ventilation velocity (Guo et al., 2019)

(b) ***Transverse Airflow system***

Transverse airflow divided into 2 subsystems which is full transverse airflow or semi-transverse airflow. It uses a separate air duct to introduce incoming fresh air and removing exhaust flue. This system usually implemented in road tunnel ventilation as it is inexpensive due to additional works and requires more substantial space. However, in terms of safety, the transverse airflow system is better than the longitudinal airflow system.

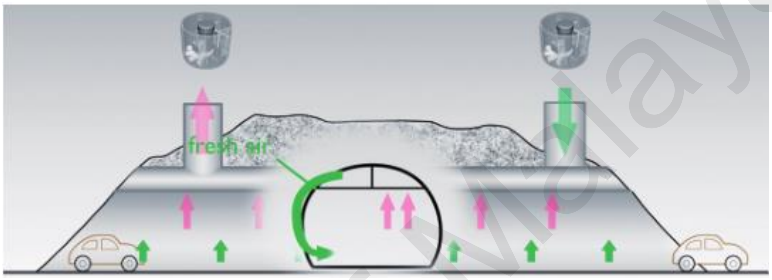


Figure 2.3 : Full transverse airflow in road tunnel (Retrieve from https://www.systemair.com/si/resitve/predori/road-tunnel-ventilation/)

Regardless of the longitudinal or transverse airflow system, cable tunnel ventilation relied on using natural or mechanical ventilation depending on the project's requirement. However, the Maju tunnel design concept is using the longitudinal airflow system instead of the transverse airflow system.

2.3.3 Natural Ventilation

The natural ventilation relies on wind energy where its direction is crucial for incoming or outgoing airflow. Pressure differences used in the natural ventilation where the incoming airflow to the contained space. The airflow inside cable tunnel changed continuously due to wind and ambient temperature. The 2 types of natural ventilation which are:

(a) ***Wind driven ventilation***

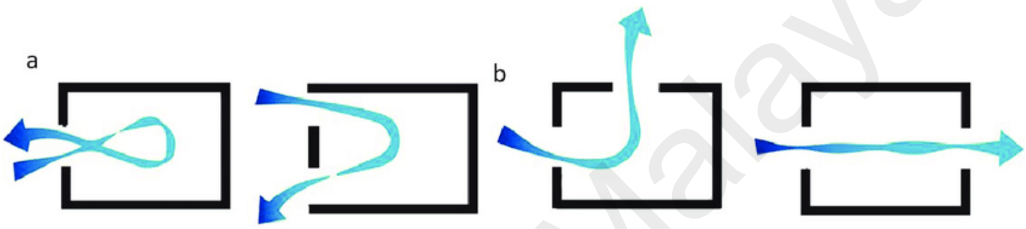


Figure 2.4 : Single sided ventilation (a) vs Cross ventilation (b) (Marzban et al., 2017)

Wind-driven ventilation system divided into the cross ventilation and the single-sided ventilation. The external wind to cool down building needed with a different strategy of removing exhaust air (Figure 2.4).

i $\quad$ *Cross ventilation airflow rate calculation*(Linden, 1999):

$$Q = U_{\mathrm{wind}} \sqrt{\frac{C_{\mathrm{p1}} - C_{\mathrm{p2}}}{1/\left(A_1^2 C_1^2\right) + 1/\left(A_2^2 C_2^2\right)}}$$

Q = Airflow rate

U_{wind} = Field wind speed

Cp_1 = Local pressure drag coefficient at upstream opening

Cp_2 = Local pressure drag coefficient at downstream opening

A_1 = Cross section area at upstream opening

A_2 = Cross section area at downstream opening

C_1 = Discharge coefficient at upstream opening

C_2 = Discharge coefficient at downstream opening

ii *Single sided ventilation airflow rate calculation* (Linden, 1999):

$$\bar{Q} = \frac{C_d\ l\ \sqrt{Cp}\ \int_{z_0}^{h} \sqrt{-\frac{2\ \Delta\ P(z)}{\rho}}\, \mathrm{d}z}{z_{ref}^{1/7}}\ \bar{U}$$

$\bar{Q}$ = mean air flow rate

l = width of the window

h = elevation of top edge of window

z_0 = elevation of neural level (Pressure balance inside and outside)

z_{ref} = reference elevation where wind velocity is measured

$\bar{U}$ = mean velocity at the reference elevation

For this report, we considered the cross ventilation as a parameter of natural ventilation (case 1) due to tunnel construction as it followed the Maju tunnel airflow system.

(b) ***Buoyancy driven ventilation***

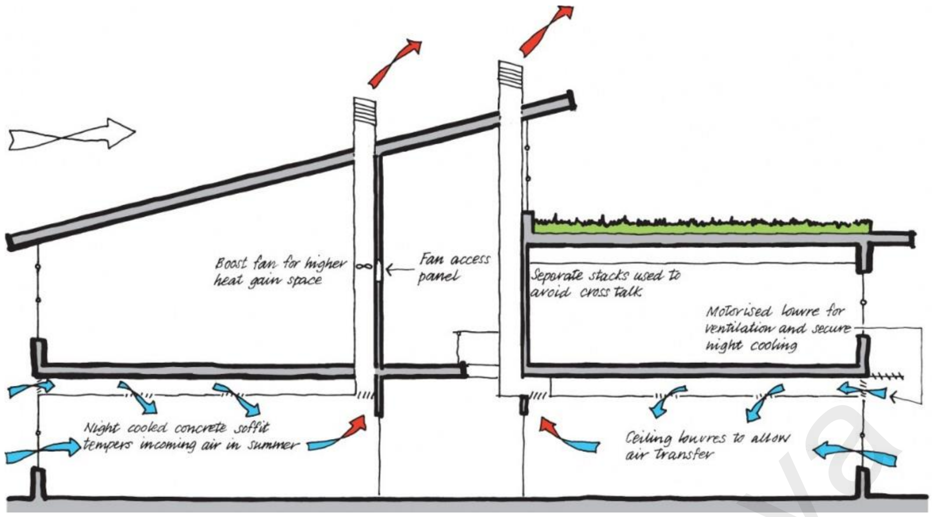


Figure 2.5 : Passive Stack ventilation

Another type of natural ventilation is buoyancy-driven ventilation. It influenced by density and temperature difference of a building. When the air temperature becomes warmer, the density decreased, and therefore, the air becomes more buoyant and rise above denser air or cold air. It created an upward air stream.

One of the standard applications of buoyancy-driven ventilation is the passive stack ventilation where usage of cross ventilation, buoyancy and venturi effect (wind passing over the terminal to crease the suction towards the end) for airflow system. Buoyancy driven ventilation airflow rate calculation(Linden, 1999):

$$Q_S = C_d\ A\ \sqrt{2\ g\ H_d\ \frac{T_I - T_O}{T_I}}$$

Q_s = buoyancy driven ventilation airflow rate

A = cross sectional area of opening

C_d = Discharge coefficient of opening

g = gravitational acceleration

H_d = height differences at midpoint of upper and lower opening

T_I = average indoor temperature between inlet and outlet

T_O = Outlet temperature

2.3.3.2 Past Study on Natural Ventilation

Natural ventilation research by experimental and simulation validation for underground construction done where it considered both thermal and airflow system. (Porras-Amores et al., 2019). Air loss study inside the tunnel compared between simulation and experimental. The study shows that CFD results comparable with experimental results within a 17% error. For more complex geometries, further meshing strategies to be adopted (Diego et al., 2011). Both papers show a similar methodology although on different parameter.

2.3.4 Mechanical Ventilation

Figure 2.6 : Mechanical ventilation in road tunnel (Retrieve from https://www.clarage.com/products/tunnel-ventilation-fans)

Mechanical ventilation is a mechanism to provide air intake or exhaust of a system by using a mechanical component such as air duct, fan and exhaust. It is beneficial for fire safety risk, thermal comfort by using mechanical ventilation. Since mechanical ventilation stands in either the longitudinal or transverse airflow system, below is an example of design consideration for ventilation.

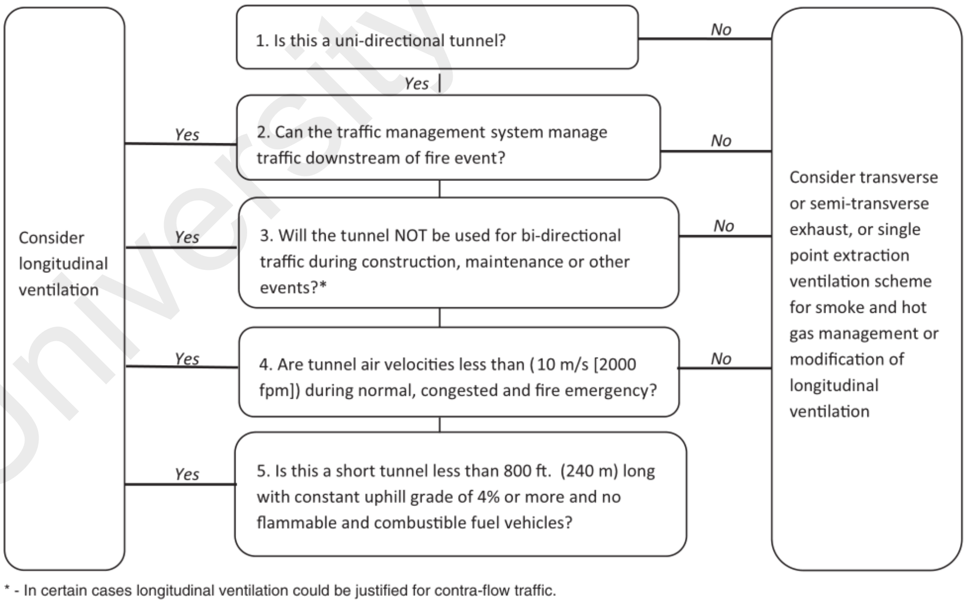


Figure 2.7 : Design consideration for ventilation system for road tunnel system (Maevski, 2017)

The Maju tunnel followed longitudinal ventilation design consideration which is uni-directional, less velocity than 10m with less than 240m length as per Figure 2.7.

2.3.4.1 Past Study on Mechanical Ventilation

Hasheminasab et al. (2019) conducted mechanical ventilation of expulsion of methane gas flow distribution inside underground coal mine study using CFD. It highlighted the mechanical ventilation with fan usage must be carefully designed to maximize the effectiveness of methane gas distribution. Without proper ventilation, it could do more harm to the workers, and the spread of methane gas unable to contain.

2.3.5 Air Change Per Hour (ACPH)

ACPH represented as the measurement volume of air incoming or removed from specific space and divided by space volume. In short, the number of times of air intake and removal from its space. Thus, ACPH in the ventilation of the tunnel utilized as the rate of changes of airflow. As the tunnel buried underground, insufficient air incoming and out has an impact on the workers.

Building/Room	**ACPH**
Assembly halls	4-6
Bars	20-30
Department Stores	6-10
Foundries	15 - 20
Garages Repair	20-30
Museums	12-15
Precision Manufacturing	10-50
Supermarkets	4-10
Swimming Pools	20-30
Transformers	10-30

Figure 2.8 : Common ACH type of room (Retrieved from https://www.engineeringtoolbox.com/air-change-rate-room-d_867.html)

In regards of Figure 2.8, although there is no mention of underground tunnel ACPH, the value of ACPH should be higher than bars (>20) due to factor it buried inside the tunnel which is enclosed area like garages. The other factors such as the temperature, humidity and air incoming flow rate have an interference impact on ACPH value.

CHAPTER 3: METHODOLOGY

3.1 Introduction

An assumption made before starting the simulation:

a. The humidity of the air factor ignored.

b. Ambient temperature retrieved from meteorological data for the Maju tunnel located in Selangor. The highest temperature is 33°C (Figure 3.1).

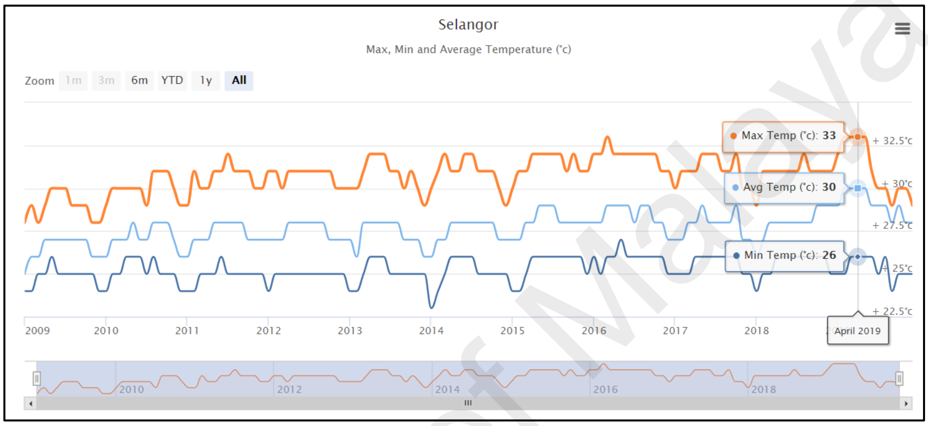


Figure 3.1 : Temperature in Selangor by yearly (Retrieved from https://www.worldweatheronline.com/selangor-weather-averages/selangor/my.aspx)

c. Soil surrounding tunnel wall temperature not considered.

d. For mechanical ventilation, air supplied by each inlet has the same pressure differences with 100% efficiency.

There are 3 simulations with a variation of velocity investigated. We divided each case as follows:

a. Case 1 – Natural Ventilation. The air inlet supplied from outside and enter the tunnel without ventilation mechanism to cool down the temperature.
b. Case 2 – Mechanical Ventilation with 2 external inlets and 1 exhaust. This design from the actual design of the Maju tunnel. The external inlet supplied the fresh air from the pump and exhaust removed the flue to reduce the temperature.
c. Case 3 – Mechanical Ventilation with 2 external inlets and 2 exhausts. Similar to Case 2 but with additional exhaust to check if the cooling temperature improved.

3.2 Simulation Parameter

Setup for this simulation parameter is used as following

Table 3.1 : Parameter for each Maju Tunnel case

Specifications	Case 1	Case 2	Case 3
Ambient Temperature (°C)	33	33	33
Cable wall Temperature (°C)	60	60	60
Tunnel Length (m)	163	163	163
*Tunnel Height (m)	3	3	3
*Tunnel Width (m)	2.45	2.45	2.45
Tunnel Volume (m^3)	1156	1213	1236
**Cable Diameter (m)	0.193	0.193	0.193

* Ambient temperature from meteorological data and cable wall temperature from Maju data.

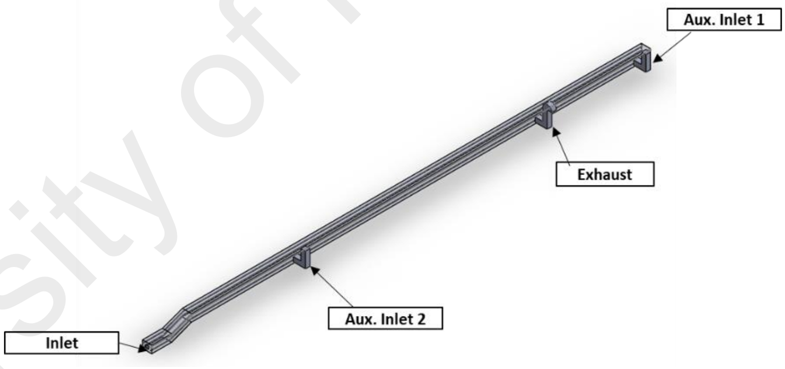
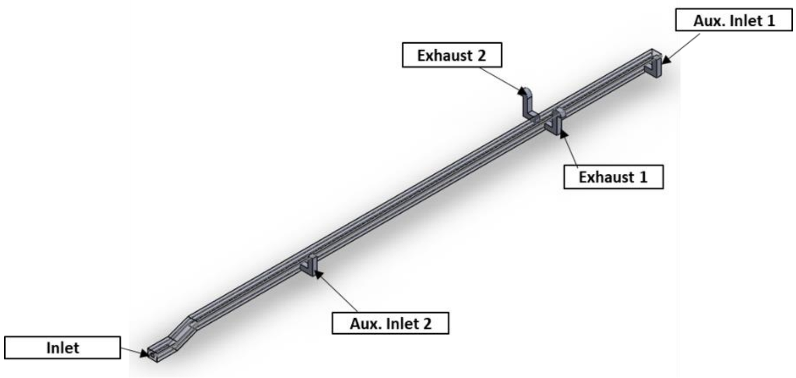
*Tunnel Height and Tunnel Height took from intake cross-section view which excludes the mechanical ventilation equipment airflow height.

**Cable diameter only for individual cable. Since there are 4 cables used in Maju tunnel. Total cable diameters are 0.772m.

3.3 Design Setup

CAD modelling for Maju Tunnel based on drawing provided by Maju tunnel plan layout by using Solidworks software.

Table 3.2 : Models for each Maju Tunnel case

Case Study	Modelling
Case 1 – Natural Ventilation	
Case 2 – Mechanical Ventilation with 2 external inlet and 1 exhaust	
Case 3 – Mechanical Ventilation with 2 external inlet and 2 exhaust	

3.4 ANSYS FLUENT Setup

To start simulation for this research project, ANSYS FLUENT 18.1 selected. There is a need to do meshing before we can run the simulation.

3.4.1 Meshing Setup

All 3 geometries are using the cut-cell method. For mechanical ventilation, the auxiliary exhaust and inlet have their meshing individually. The denser of meshing, more accurate result is achievable. Each geometry is set as fluid to replicate actual airflow inside the tunnel. Table 3.3 shows the geometry views and statistics of meshing of each case by Table 3.4.

Table 3.3 : Meshing model for each Maju Tunnel case

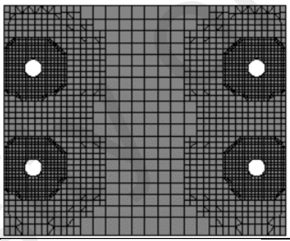
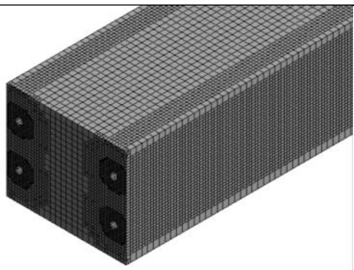
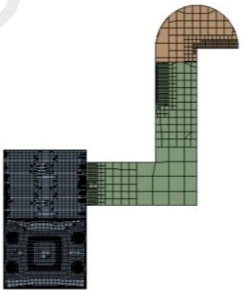
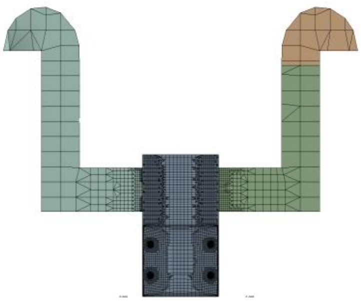
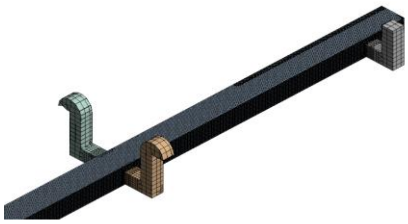
Case Study	Front View	Isometric View
Case 1 – Natural Ventilation		
Case 2 – Mechanical Ventilation with 2 external inlet and 1 exhaust		
Case 3 – Mechanical Ventilation with 2 external inlet and 2 exhaust		

Table 3.4 : Meshing statistic for each Maju Tunnel case

Statistic	**Case 1**	**Case 2**	**Case 3**
No of Nodes	6021759	6104420	8804787
No of Elements	5301334	5598896	8259013
Average Skewness	0.0019	0.16	0.13
Max	0.25	0.45	0.50
Min	1.31 x 10^{-10}	3.6 x 10^{-9}	8.6 x 10^{-7}

Both tables above indicate meshing types of each case. As geometry grew complex, the meshing takes longer time to complete, and the number of nodes also increase; Case 3 has 8,804,787 number of nodes as compared to Case 1 which as 6,021,759 number of nodes. For meshing quality, we opted to use skewness as measurement guideline where maximum skewness must be less than 0.95. Skewness selected due to meshing for the geometries under cartesian the cut-cell method. The advantage of the cut-cell method it is accurate where the mesh is stationary without stretching and compressing the cell when geometry moves.

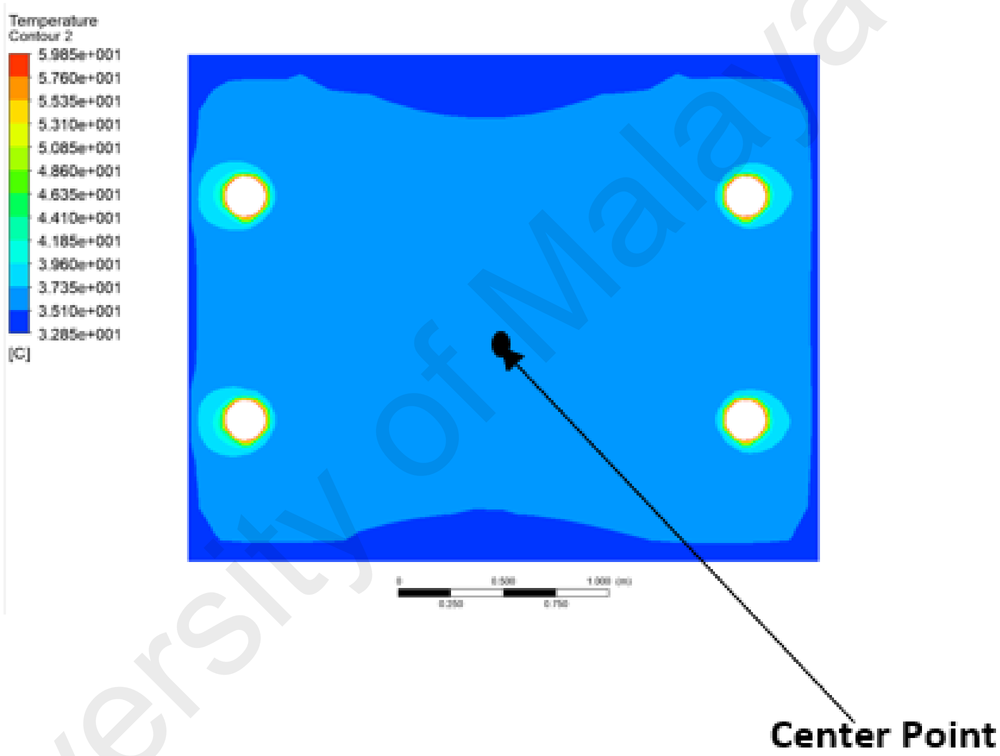
After meshing completed, the solution process setup began. At this stage, we selected the turbulence model, which is the standard k-ε model, the SIMPLE method as a CFD solver for this modelling. Next, we set the boundary condition for each case. Every parameter follows as per Table 3.1. For example, the thermal boundary condition for cable wall temperature set at 333K (60°C) and wall temperature based on ambient temperature, which is 306K (33°C). At the same time, we set inlet velocity at 0.5m/s, 1.0m/s and 2.0m/s respectively on each case.

Iteration taken for each case is 600 steps to observe whether these steps are capable of reaching convergence. In the residual graph, as the iteration began, the simulation stops at once if reached convergence.

3.5 Result Analysis

3.5.1 Temperature distribution

Temperature contour of the airflow in tunnel measured at the centre point of each cut section of the tunnel. For more details on the cut section, the explanation is in the next chapter. Then, the temperature of each section compiled and plotted the graph to compare the effect of different velocity inlet.



5

Figure 3.2 : Temperature contour of tunnel

3.5.2 Air Change per Hour calculation (ACPH)

By applying the formula below, we can calculate and tabulate ACPH needed for each case. The higher number of ACPH means more airflow incoming or removal (Retrieved from https://www.engineeringtoolbox.com/air-change-rate-d_882.html).

$$ACPH = \frac{60Q}{Vol}$$

$ACPH$ = Air change per hour, the rate of changeable airflow inside a space

Q= Volume air flow rate in CFM

Vol = Volume of the tunnel. (Width x Height x Length)

3.5.3 Pumping Power calculation

Pumping power for mechanical ventilation as per below equation (Retrieved from https://neutrium.net/equipment/pump-power-calculation/) :

$$P = \frac{Q\ x\ dP}{3600}$$

Q= Volume air flow rate of pump's fluid in m^3/hr

dP = Differential pressure across pump in kPa

Static Pressure of each auxiliary inlet is taken

CHAPTER 4: RESULT AND DISCUSSION

4.1 Case 1: Natural Ventilation Simulation

For Natural ventilation simulation, we observed the temperature distribution at various sections to understand the effect of velocity. From Figure 4.1, cross-section defined; each section displayed its temperature contour for further analysis on next the subsection.

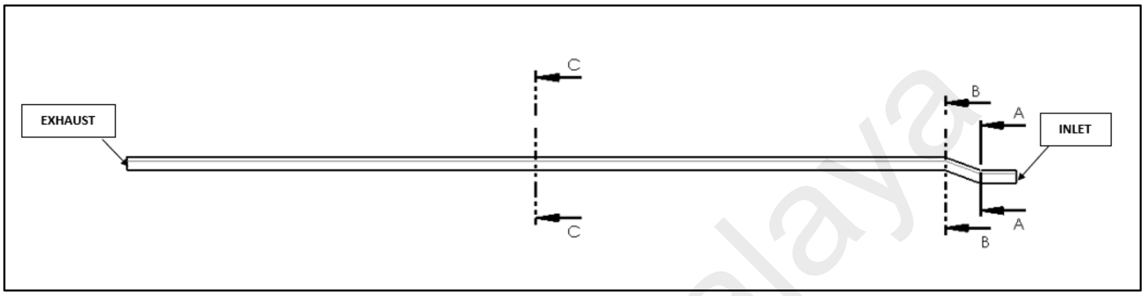


Figure 4.1: Tunnel cross section for parameter observation (Natural ventilation)

4.1.1 Air Change Per Hour (ACPH)

Table 4.1 : ACPH calculation for Case 1

Inlet Area (m^2)	Velocity Inlet (m/s)	Volume (ft^3)	CFM (ft^3/min)	ACPH
7.23	0.5	40817.03	7662.91	11.26
7.23	1.0	40817.03	15325.81	22.53
7.23	2.0	40817.03	30651.63	45.06

Table 4.1 indicated the higher velocity provided the better Air Change per Hour (ACPH). For example, ACPH for velocity 0.5 m/s produced 11.26 meanwhile velocity of 2.0m/s produced 45.06. The higher value of ACPH means more air exchangeable throughout the tunnel. Since the benchmark used is in range 20-40; the velocity of 1.0m/s with 22.53 ACPH suited in this case but 2.0m/s with 45.06 ACPH not much difference.

4.1.2 Temperature distribution across plane

Table 4.2: Temperature distribution for Natural Ventilation

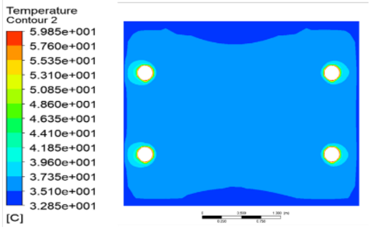
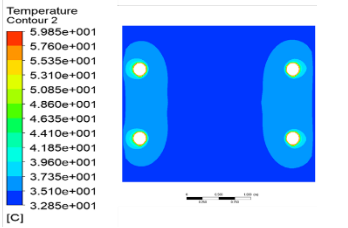
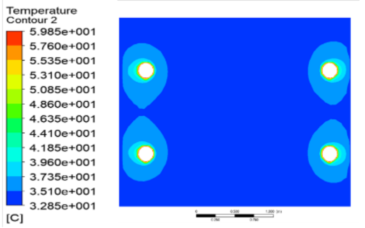
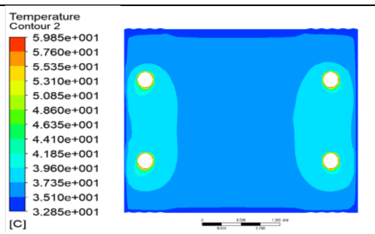
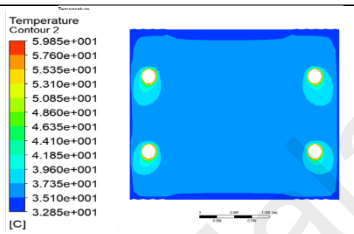
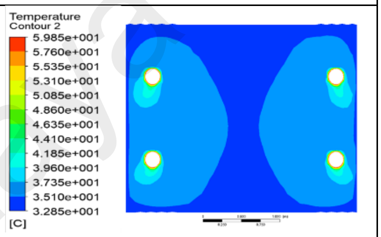
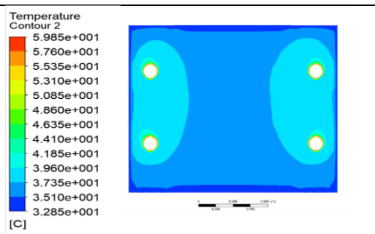
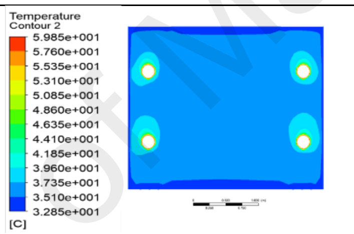
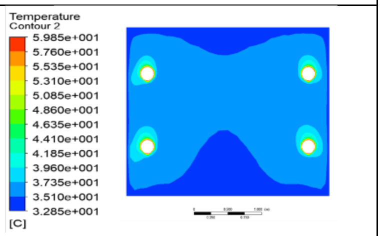
Section	**V= 0.5 m/s**	**V= 1.0 m/s**	**V = 2.0 m/s**
Section A-A			
Section B-B			
Section C-C			

Table 4.2 shows the contour of temperature changes around cables wall at a different velocity. An observation of the contour pattern changed at the highest temperature is around 4 cables wall which is 60°C. The temperature slowly increases as it travels along the length. The velocity of 0.5m/s shows in section A-A, the centre point temperature is at 38.5°C as compared to the velocity of 2.0 m/s where the centre point temperature is at 33°C. It means at the early stage of the tunnel; the air incoming is unable to cool down for the velocity of 0.5m/s as compared to the velocity of 2.0m/s.

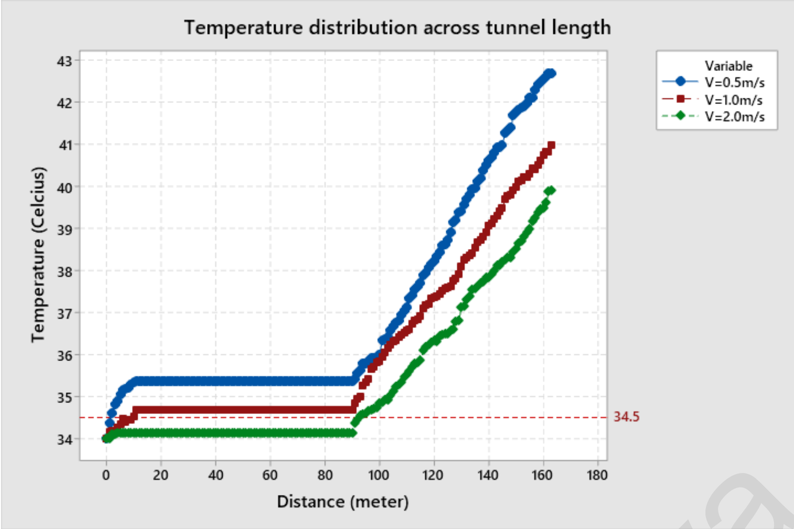


Figure 4.2 : Temperature changes across tunnel in different velocity (Case 1)

Based on Figure 4.2, the pattern of the graph shows it is linear at the beginning inlet and linearly increase after 90m from the entrance. The comparison between velocities of inlet observed from this point where it shows 0.5 m/s can reach 42.8°C where the workers are likely to feel uncomfortable, and exhaustion due to heat increase as this temperature surpassed the benchmark of 34.5˚C. Therefore, thermal comfort failed if we insisted on using natural ventilation. Even with the highest velocity, which is 2.0 m/s, reached 40°C at the end of the tunnel. This temperature not ideal for workers as the work carried out or the amount of energy used increased while inside the tunnel. In summary, natural ventilation is not ideal for Maju Tunnel as the temperature is increasing towards the end. This data does not include external factor such as debris that affected users experience inside the tunnel.

4.2 Case 2: Mechanical Ventilation (2 auxiliary inlet and 1 exhaust)

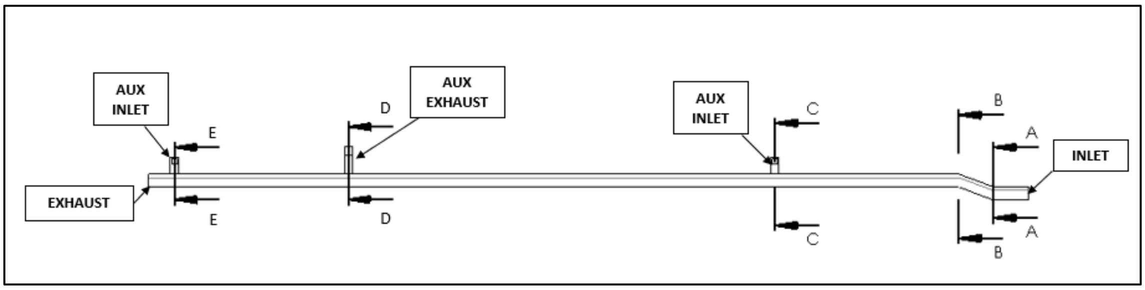


Figure 4.3: Tunnel cross section for parameter observation (Mechanical Ventilation)

Figure 4.3 shows the cross-section of mechanical ventilation where there is double auxiliary inlet pump in the tunnel and single exhaust for the airflow system. This design is the original design of Maju tunnel team.

4.2.1 Air Change Per Hour (ACPH)

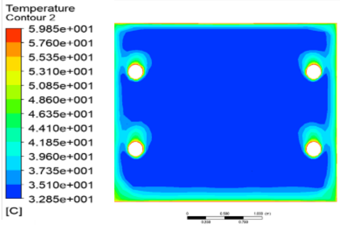
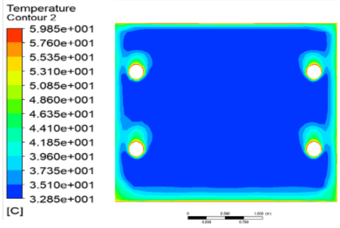
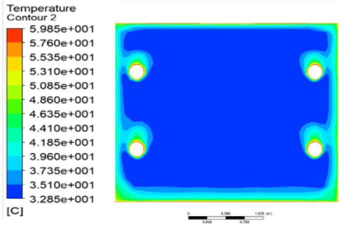
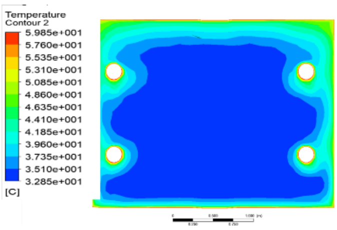
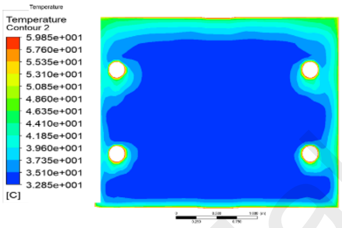
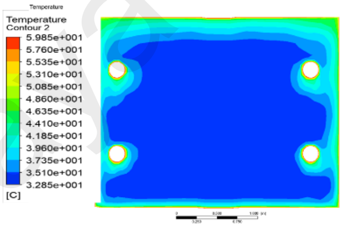
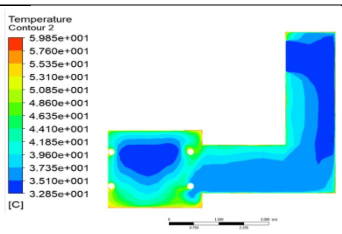
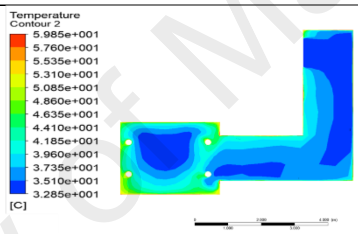
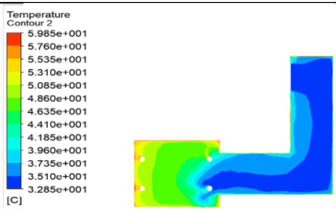
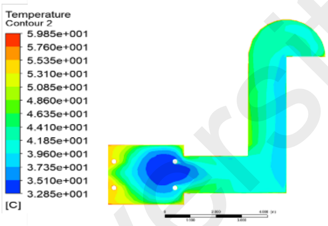
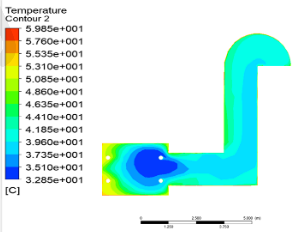
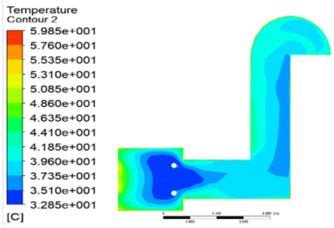
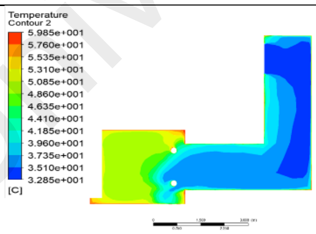
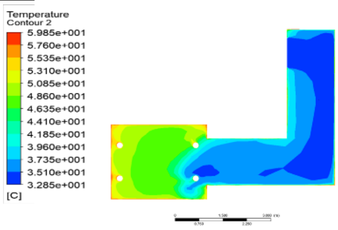
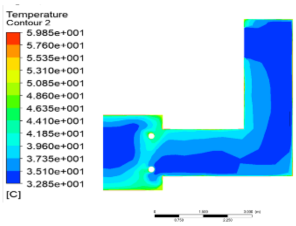
Table 4.3 : ACPH calculation for Case 2

Inlet Area (m^2)	Velocity Inlet (m/s)	Volume (ft^3)	CFM (ft^3/min)	ACPH
7.23	0.5	42830.85	7662.91	10.73
7.23	1.0	42830.85	15325.81	21.47
7.23	2.0	42830.85	30651.63	42.94

Comparably with section 4.1.1; the higher ACPH influenced by higher CFM or velocity. For example, ACPH for the velocity 0.5m/s produced 10.73 meanwhile the velocity of 2.0m/s produced 42.94. Rate of air incoming and outgoing defined by this value of ACH in a confined space. The velocity of 1.0m/s is ideal for case 2, but the velocity of 2.0m/s is not much different than benchmark range of 20-40; this is due unknown quantities inside the tunnel might affect the value.

4.2.2 Temperature distribution

Table 4.4: Temperature distribution for Mech. Ventilation (2 aux. inlet & 1 exhaust)

Section	**V= 0.5 m/s**	**V= 1.0 m/s**	**V = 2.0 m/s**
Section A-A			
Section B-B			
Section C-C			
Section D-D			
Section E-E			

The temperature changes across tunnel as shown by temperature contour at a velocity of 0.5, 1.0 and 2.0m/s in Table 4.4. The temperature increased as the distance grew towards the end of the tunnel. The difference temperature contour between Case 1 and Case 2 at the section C-C, D-D and E-E are due to the addition of auxiliary components. For example, at section C-C, we observed there is a low-temperature contour reached 34°C at the inlet component because the inlet received air supply from the auxiliary pump. The average temperature is taken for each case as follows: the velocity 0.5m/s with 36.9±2.4°C, the velocity of 1.0m/s with 36±2°C, and the velocity of 2.0m/s with 35.3±1.7°C.

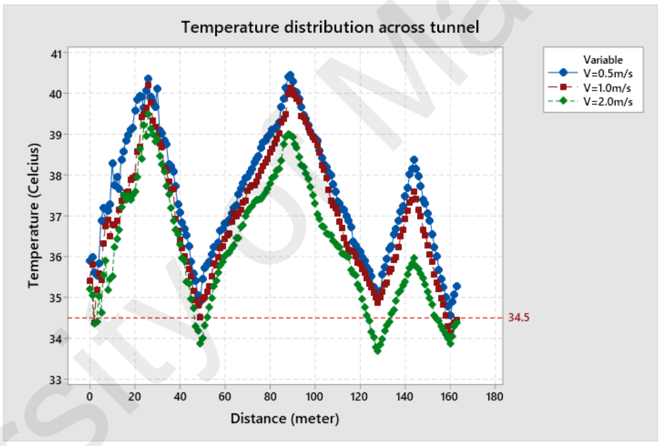


Figure 4.4 : Temperature changes across tunnel in different velocity (Case 2)

As illustrated by Figure 4.4, the graph curve indicated a similar pattern for all velocity variations. For example, the temperature is at the highest peak, which is at 20m than fall to the minimum temperature at 45m and increased as it travelled further from the auxiliary components. This pattern repeated every time the auxiliary component added. The temperature at 0.5m/s can reach maximum 40.3°C at peak and 34.6°C on minimum temperature. This temperature improved by velocity increased at 2.0m/s where the highest temperature reached 39.3°C and the lowest temperature at 33.8°C. This mechanical ventilation result is a significant improvement as compared to natural ventilation as workers can reach better thermal comfort.

4.3 Case 3: Mechanical Ventilation (2 auxiliary inlet and 2 exhaust)

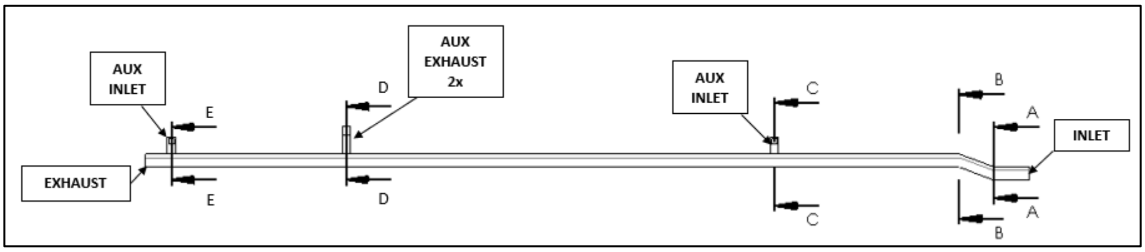


Figure 4.5 : Tunnel cross section for parameter observation (Mechanical Ventilation)

For Case 3, we investigated if the additional exhaust auxiliary component has a significant improvement in temperature distribution as per Figure 4.5. We do not consider additional auxiliary as the addition of pump lead to cost increased than additional exhaust only.

4.3.1 Air Change Per Hour (ACPH)

Table 4.5 : ACPH calculation for Case 3

Inlet Area (m^2)	Velocity Inlet (m/s)	Volume (ft^3)	CFM (ft^3/min)	ACPH
7.23	0.5	43653.46	7662.91	10.53
7.23	1.0	43653.46	15325.81	21.06
7.23	2.0	43653.46	30651.63	42.13

Table 4.5 proves the higher CFM determined the higher ACH which same as per Case 1 and Case 2. Airflow CFM value influenced by velocity; thus, the higher velocity has higher CFM. Similarly, to Case 1 and Case 2, the velocity of 1.0m/s achieved the optimum benchmark of ACPH in range of 20-40, but the velocity of 2.0m/s should not be ignored, as its value slightly above standard range. The external factors may have caused a slight change of ACPH values as the above value determined in ideal condition.

4.3.2 Temperature distribution

Table 4.6: Temperature distribution for Mech. Ventilation (2 aux. inlet & 2 exhaust)

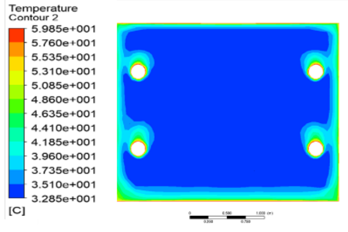
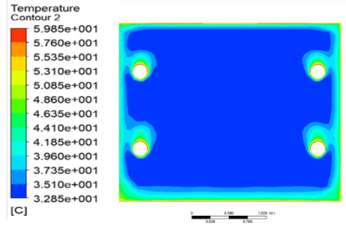
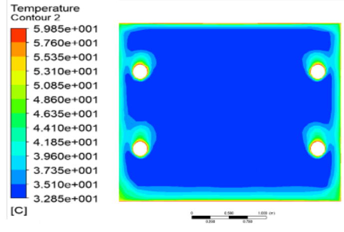
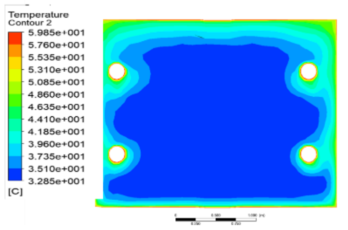
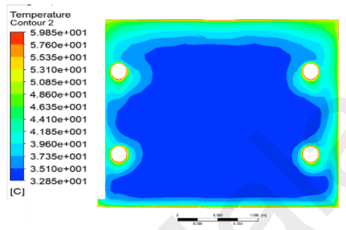
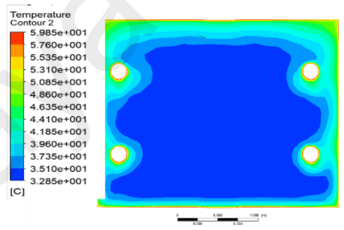
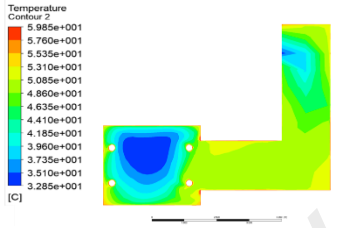
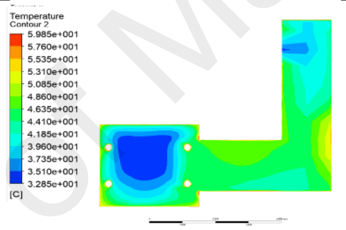
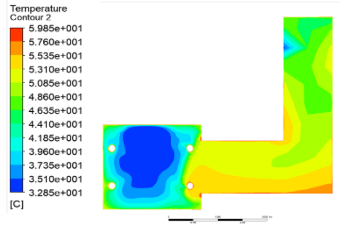
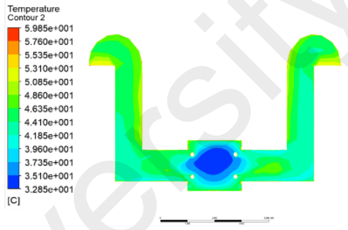
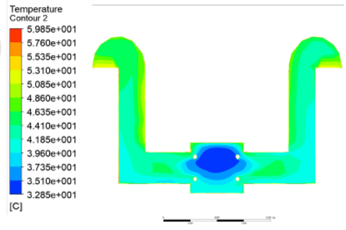
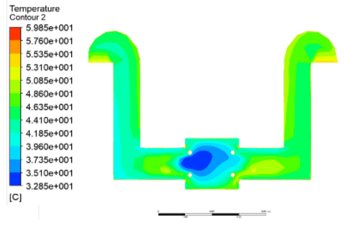
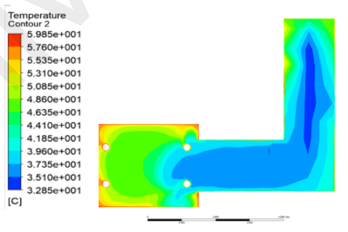
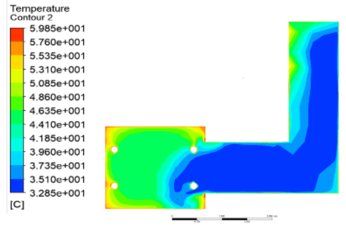
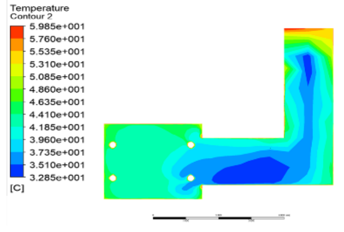
<table>
<tr><th>Section</th><th>V= 0.5 m/s</th><th>V= 1.0 m/s</th><th>V = 2.0 m/s</th></tr>
<tr><td>Section A-A</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td>Section B-B</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td>Section C-C</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td>Section D-D</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td>Section E-E</td><td>
</td><td>
</td><td>
</td></tr>
</table>

Table 4.6 shows the comparison of the temperature contour at individual sections at the various velocity of 0.5, 1.0 and 2.0m/s. The resemblance with Case 2, the inclusion of the auxiliary component shows temperature contour have variation temperature range as compared to case 1. Due to turbulence nature, the temperature variation more noticeable at section C-C, D-D and E-E as incoming and outgoing flow. The average temperature is taken for each case as follows: the velocity 0.5m/s with 37.1±1.5°C, the velocity of 1.0m/s with 36.5±1.5°C, and the velocity of 2.0m/s with 35.7±1.5°C. There are minimal differences in average temperature between Case 2 and Case 3, this due to the auxiliary components are located at the same place despite the addition of another exhaust.

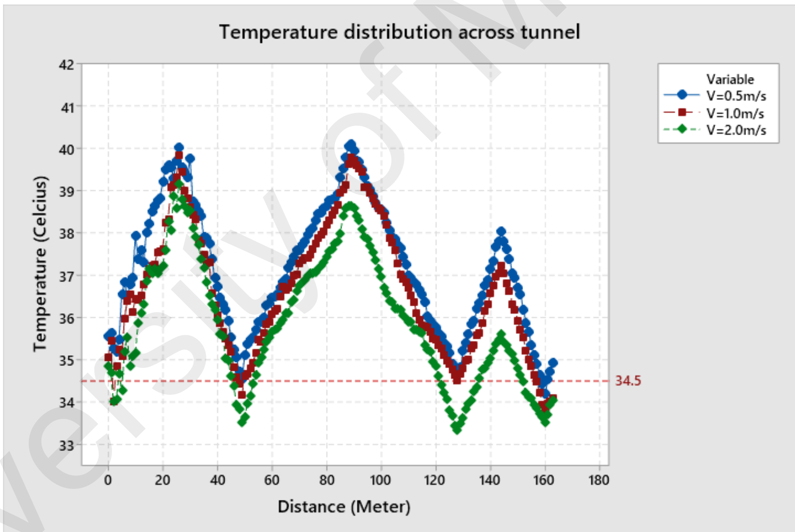


Figure 4.6 : Temperature changes across tunnel in different velocity (Case 3)

Figure 4.6 indicated the temperature changes across the tunnel, where it is comparable with Case 2. Further comparison referred in section 4.4. Even with additional auxiliary exhaust, there is no contrasting value exceeding 40°C. For example, at velocity 0.5m/s; the temperature reached the maximum temperature of 40°C and the minimum temperature of 33.9°C. Thus, this temperature is ideal for the worker to gain the best thermal comfort as the peak temperature located at 25m and 90m.

4.4 Comparison of Mechanical Ventilation (Case 2 vs Case 3)

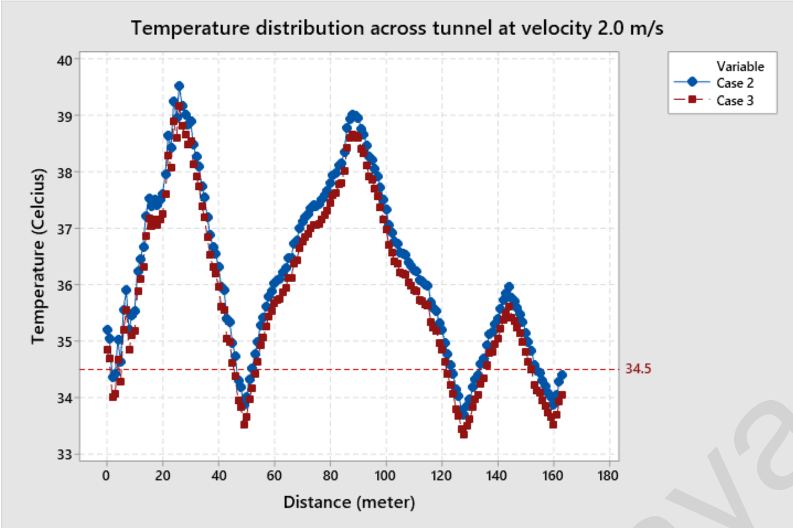


Figure 4.7 : Comparison of Case 2 & 3 in term of velocity on temperature changes

In this section, we compared both mechanical ventilation (Case 2 & Case 3) and determine whether it is enough to fulfil the ventilation required of Maju tunnel. The temperature distributions are identical as the inlets are the same for both cases.

The velocity of 2.0m/s selected as the best outcome for both cases, as the temperature drops to 33.8˚C (Case 2) and 33.9˚C (Case 3), which are the ideal temperature for worker's productivity. Thus, case 3 does not show significant changes in temperature.

In term of pumping power calculation with 100% efficiency, Case 2 and Case 3 have similar power which is 148kW (Case 2) and 149.9kW (Case 3) as per below:

Case 2: $\left(\frac{14.466*36.9099}{3600}\right) * 1000 = 148.3kW$

Case 3: $\left(\frac{14.466*37.322}{3600}\right) * 1000 = 149.9kW$

The percentage error between Case 2 and benchmark and Case 3 and benchmark:

Case 2: $\left(\frac{186.5-148.3}{186.5}\right) * 100 = 20.48\%$

Case 3: $\left(\frac{186.5-149.9}{186.5}\right) * 1000 = 19.62\%$

Even though the error of both cases is around 20.48% and 19.62%, the design for the benchmark is different such as inlet pump quantities, pressure drops and others.

CHAPTER 5: CONCLUSION AND RECOMMENDATION

As for the conclusion, we have achieved the object of this research report to determine the ideal temperature met the lowest threshold, which is Case 2 at 33.8˚C. This result achieved by CFD simulation. This temperature defined for workers to achieve the best thermal comfort and the cable life has higher lifespan due to low temperature throughout the tunnel.

Concerning on ACPH, the air exchange inside and out of the tunnel is similar for all cases. It means the design sufficient for workers to do tasks without health risks at either the velocity of 1.0m/s or 2.0m/s, but the external factors such as humidity affected the ACPH values.

Pumping power is similar for both Case 2 and Case 3, where there is no significant change. However, for further study, we should focus on efficiency pump as well as power loss relationship with nozzle pump location. Furthermore, when we want to determine ventilation in the tunnel, we can continue to study air compression, particle size and the soil height effect relationship with temperature to be taken into consideration for tunnel design.

REFERENCES

Anderson, J. D. (1995). *Computational Fluid Dynamics : The Basics with Applications*.

Arjun, N. (2017). Ventilation in tunnels - types of ventilation systems in tunnel construction. Retrieved from https://theconstructor.org/transportation/ventilation-systems-in-tunnel-construction/15998/

Diego, I., Torno, S., Toraño, J., Menéndez, M., & Gent, M. (2011). A practical use of CFD for ventilation of underground works. *Tunnelling and Underground Space Technology, 26*(1), 189-200. doi:https://doi.org/10.1016/j.tust.2010.08.002

Duris, L., Aldorf, J., & Geryk, J. (2017). Thermal load of the final lining of traffic tunnels. *IOP Conference Series: Materials Science and Engineering, 236*, 012073. doi:10.1088/1757-899x/236/1/012073

Favier, D. (2010). The Role of Wind Tunnel Experiments in CFD Validation. In *Encyclopedia of Aerospace Engineering*. doi:10.1002/9780470686652.eae034

Fengying, Y., Huaigang, L., Long, W., & Haixia, Z. (2011). *Simulation of Indoor Dynamic Thermal Comfort Based on CFD.* Paper presented at the 2011 Fourth International Conference on Intelligent Computation Technology and Automation.

Goswami, D. Y., & Biseli, K. M. (1993). Use of Underground Air Tunnels for Heating and Cooling Agricultural and Residential Buildings. *Fact Sheet EES 78: A Series of the Florida Energy Extension Service, Florida Cooperative Extension Service, Institute of Food and Agricultural Sciences, University of Florida*.

Guo, F., Gao, Z., Wan, H., Ji, J., Yu, L., & Ding, L. (2019). Influence of ambient pressure on critical ventilation velocity and backlayering distance of thermal driven smoke in tunnels with longitudinal ventilation. *International Journal of Thermal Sciences, 145*, 105989. doi:https://doi.org/10.1016/j.ijthermalsci.2019.105989

Hasheminasab, F., Bagherpour, R., & Aminossadati, S. M. (2019). Numerical simulation of methane distribution in development zones of underground coal mines equipped with auxiliary ventilation. *Tunnelling and Underground Space Technology, 89*, 68-77. doi:https://doi.org/10.1016/j.tust.2019.03.022

Linden, P. F. (1999). The fluid mechanics of natural ventilation. *Annual Review of Fluid Mechanics, 31*(1), 201-238. doi:10.1146/annurev.fluid.31.1.201

Maevski, I. (2017). *Guidelines for Emergency Ventilation Smoke Control in Roadway Tunnels*.

Malek, N. A., Khairuddin, M. H., & Rosli, M. F. (2015). Thermal comfort investigation on a naturally ventilated two- storey residential house in Malaysia. *IOP Conference Series: Materials Science and Engineering, 88*, 012014. doi:10.1088/1757-899x/88/1/012014

Marzban, S., Ding, L., & Fiorito, F. (2017). An Evolutionary Approach to Single-sided Ventilated Façade Design. *Procedia Engineering, 180*, 582-590. doi:10.1016/j.proeng.2017.04.217

Porras-Amores, C., Mazarrón, F. R., Cañas, I., & Villoría Sáez, P. (2019). Natural ventilation analysis in an underground construction: CFD simulation and experimental validation. *Tunnelling and Underground Space Technology, 90*, 162-173. doi:https://doi.org/10.1016/j.tust.2019.04.023

Sadokierski, S., & Thiffeault, J.-L. (2007). Heat Transfer in Underground Rail Tunnels. Retrieved from https://arxiv.org/abs/0709.1748

Seppänen, O., Fisk, W. J., & Lei, Q. H. (2006). Ventilation and performance in office work. *Indoor Air, 16*(1), 28-36. doi:10.1111/j.1600-0668.2005.00394.x

ToolBox, E. (2005). Air Change Rates in typical Rooms and Buildings. Retrieved from https://www.engineeringtoolbox.com/air-change-rate-room-d_867.html

Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics 2nd Edition*.

Xu, X., Yuan, Q., Sun, X., Hu, D., & Wang, J. (2019). Simulation analysis of carrying capacity of tunnel cable in different laying ways. *International Journal of Heat and Mass Transfer, 130*, 455-459. doi:https://doi.org/10.1016/j.ijheatmasstransfer.2018.10.059

Zuo, W. (2005). *Introduction of Computational Fluid Dynamics*. FAU Erlangen-Nürnberg. JASS 05, St. Petersburg.