Proceedings of 10th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2021 Month: October Volume: 10

Emergency Ventilation for Smoke Control in Head Race Tunnel at Solu Khola Dudh Koshi Hydro Power Project 86 MW

Amrit Bhattarai [a], Prajwal Raj Shakya [b]

[a, b] *Department of Automobile and Mechanical Engineering, Thapathali Campus, IOE, Tribhuvan University, Nepal*
Corresponding Email: [a]amritbhattarai085@gmail.com, [b]prajwalrajshakya@gmail.com

Abstract
This is a paper related to the tunnel fire which is caused due to misfire, excessive combustible gases,short circuit and other hazardous activities which takes place inside tunnel during construction of tunnel. In this paper,Solu Khola Dudh Koshi Hydro Electric Project(SKDKHEP) Head Race Tunnel size is reduced to 1:20 scale and fire scenario is created.In addition to this,comparative study is done by arranging ventilation fan at different place inside tunnel. By using Pyrosim, simulation is done and optimized solution is achieved.Also, fan load for overall tunnel ventilation is calculated.Also, Heat Release Rate(HRR) is plotted against time alongwith parametric study of Velocity and temperature profile during the fire evacuation.

Keywords
Critical Velocity, Computational Fluid Dynamics (CFD), Tunnel Ventilation, Heat Release Rate(HRR), Backlayering

1. Introduction

Solu Khola Dudh Koshi Hydroelectric Project(SKDKHEP) is a 86 MW project which is being constructed at Solududhkunda Municipality & Thulung Dudh Koshi Gaupalika.The project uses discharge of 17.05 m3/s. This project is being constructed by CE Construction Pvt. Ltd. Head Race Tunnel(HRT) of total length 4503 m is constructed to flow the water which is joined to Drop shaft-I and Drop shaft-II of height 185m and 255m respectively.During the construction of the HRT, ventilation fan is required to supply air flow inside tunnel for proper working environment.During construction and service of the tunnel different machines is run with electricity and fuels. This is a major challenge to dilute the emission like CO,methane(CH4).Also, there might take place fire inside tunnel which results destruction of the machines,human lives and eventually have negative impact in the tunnel.

The issue of fireplace and smoke control is a major challenge in tunnel. This publication addresses the threats to human health because of fire events, e.g., high temperatures, the existence of varied toxic gases, and low oxygen content. Moreover, low visibility not only can jeopardize the evacuation but can also slow down the rescuing and firefighting with high temperatures and radiation leading to increasing fire [1].

Thus, fire and smoke control are essential to:
1. Save lives by facilitating user evacuation,
2. Support rescue and fire-fighting operations,
3. Reduce risk of explosions,
4. Reduce damage to tunnel structure and equipment and to surrounding facilities

Figure 1: Damage caused by fire inside tunnel in Mont Blanc Tunnel [1]

1.1 Problem Statement

Tunnels are essential parts of the infrastructure.Mainly, it is constructed for movement of the vehicles,flow of pressurized water for production of electricity,supplying water for irrigation, etc.A safe operation of these is crucial.Tunnel ventilation plays special role in this case.Tunnel ventilation provides essential fresh air and necessary air conditions during normal tunnel operation and control smoke and other hazards.Various types of ventilation systems are used depending on its tunnel characteristics and geometry.Mainly, transverse and longitudinal ventilation is used.Transverse systems uses combined supply and extraction points for smoke control and are suitable for two-directional traffic. Whereas, longitudinal systems control smoke by providing longitudinal air flow and blowing the smoke in single direction.Critical velocity is the minimum velocity which does not allow backlayering.Ventilation fan should be placed at various location inside tunnel to avoid backlayering.However, it affects Heat release rate(HRR) and may increase the HRR.This is a challenging topic and study is going on to find optimized ventilation system.So, this topic will use Pyrosim which is a Fire Dynamic Simulator(FDS) to establish the relationship between the number of required jet fans and heat release rate of a fire for a typical uni-directional road tunnel.

1.2 Related Researches

As mentioned by [2], atmosphere of tunnel is affected by the fire that needs to be in control, which is possible by ventilation of tunnels. The conception about ventilation differs from country-to-country [2]. As for some, it means to prevent the rising of smoke (upstream) whereas others understand it as reducing the smoke propagation by maintaining low velocity of the tunnel during self-evacuation [2]. It can be debatable in some instances because the firstly mentioned option entails having moderately high velocities upstream of the fire (2.5 to 3m/s), and the second mentioned option needs comparatively low smoke propagation velocities since the walking speed inside an area filled with smoke is about 0.5m/s [3]. This is why, low velocity is applied of 1.0–1.5m/s upstream of the fire is applied as a form of negotiation [3]. It helps to control the effects of back layering of the smoke on the upstream side and also maintains lower smoke propagation downstream of the fire [3]. Furthermore, [3] also posit that while longitudinal ventilation is mostly moved from the downstream site of fire to the whole, reformed transverse ventilation systems permit withdrawal of smoke in the tunnel and thus offers large smoke-free zones to both sides of the fire. However, to restrict the withdrawal of smoke, transversely ventilated tunnels require complicated control system for ventilation (Sturm, et. al., 2017). Which is why the information on movement of smoke inside the tunnel (for monitoring purpose) should be correctly recognized by highlighting the controlled performance of the transverse ventilation as [3]. Fire Ventilation, according to [3], aids to rescue and evacuate during any emergency situation, for which, fire detection alarm system can be useful to make sure that workers and equipment are safe inside the tunnel. In this study, uses of Mono-Dimensional model, CFD (Fluent) and Multi-Dimensional Model has been compared. The multi-dimensional model by the use of Mono dimensional model for Far field and CFD for 2D/3D geometry [4]) will be mostly focused.

In a large aspect ratio tunnel, the velocity of smoke above the source of fire was produced quickly [2]. In small aspect ratio tunnel, according to Park, et. al. (2019), the buoyancy force and momentum force, smoke velocity in the longitudinal direction is bigger. The smaller the aspect ratio, the higher is the temperature inside cross-section area of the tunnel as the blaze touches the underneath the ceiling directly [2]. In addition, the vortex produced by the wall was strong when the width of the tunnel was narrow due to which securing visibility was challenging [2]. The areas with constant heat release undergo inadequate changes in temperature, while the temperature changes are seemingly higher in areas where the heat release rate is clearer [2].

2. Gas Test Result

Table 1: Gas Test Result

S.N.	Location	Activity	H2S(ppm)	O2(%)	CO(ppm)	EX(%lel)
1	Adit-2 HRT	Survey and Rib	0	19.3	63	0
2	Adit-2 HRT	Anchor	0	19	200	0
3	Adit-2 HRT	Face Drilling	0	19.3	118	0
4	Adit-2 HRT	Shotcrete	0	20.8	0	0
5	Adit-2 HRT	Mucking	0	19.6	0	0
6	Adit-2 HRT	Survey and Rib	0	20.2	0	0
7	Adit-2 HRT	Chipping	0	19.5	200	0
8	Adit-2 HRT	Survey and Rib	0	19.11	60	0
9	Adit-2 HRT	Anchor	0	18.9	159	0
10	Adit-2 HRT	Face Drilling	0	19.7	127	0
11	Adit-2 HRT	Shotcrete	0	19.2	132	0
12	Adit-2 HRT	Mucking	0	18.9	135	0
13	Adit-2 HRT	Survey and Rib	0	20.7	0	0
14	Adit-2 HRT	Chipping	0	20	123	0

3. Methodology

The power necessary to evacuate the tunnel is to be calculated for which relevant data is collected via different articles [5]. Formulas will be used to calculate the necessary power required to drive the ventilation fan. In addition to it, CFD (Fluent) analysis is done to calculate energy required to evacuate safely [5]. Solidworks will be used for creating CAD model. Pyrosim is used which is a Fire Dynamic Simulation which is a CFD model of a fire-driven fluid flow. Literature review is done to collect necessary data.

4. Design

4.1 CAD Model

The total length of HRT tunnel is 4500m.Since the limitation of Solidworks in terms of length, and time required for simulation was a challenge, total size of the tunnel was reduced to 225 m (Ratio 1:20)

Length: 225 m

Table 2: Dimensions

Dimensions	Real Scale	Reduced Scale
Length	4500 m	225 m
Width/Diameter	3.8 m	0.19 m
Height	4 m	0.2 m

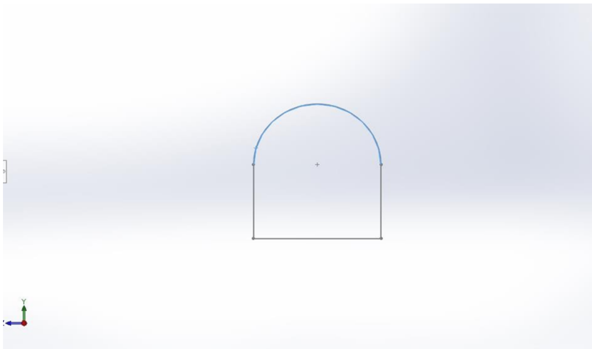

Figure 2: Tunnel Front View

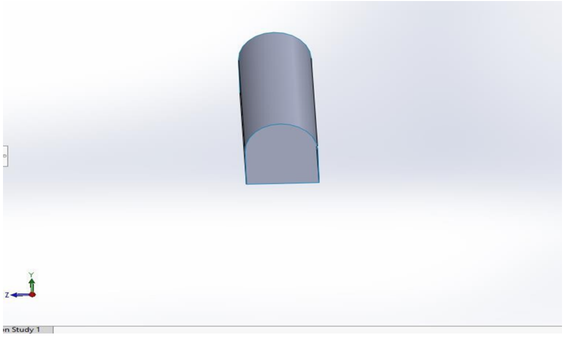

Figure 3: Tunnel Top View

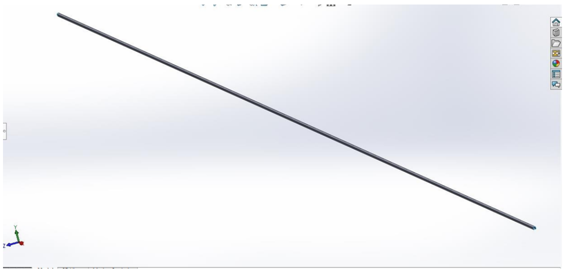

Figure 4: Overall Tunnel

5. Calculations

Froude Scaling Procedure is followed to reduce simulation time from the real scale to reduced scale so as to reduce the simulation time.

$$Fr = \frac{v}{\sqrt{gd}}$$

where, v=velocity of flow,m/s g=acceleration due to gravity,m/s^2 d=length of the tunnel,m

Initial velocity of fluid=4 m/s

Using Froude scaling, velocity reduced from 4 m/s to 0.9 m/s

In an undivided tunnel section, the total pressure change can be calculated by adding contributions from friction and area changes in the tunnel, vehicle piston effect, jet fans, and stack effect.

Following calculation shows total load of the Ventilation for safe and smooth operation of the work inside tunnel.

$$\text{Total Air to be supplied per sec}(Q) = A \times\ L \times\ ACH$$

where,

A=Area of tunnel=13.0068 m^2

L=Length of tunnel=4500 m

Q=58500 m^3

$Air\ Change\ per\ Hour\ (ACH)$= 4

$$\text{Total Air to be supplied per sec}(Q) = A \times L \times ACH$$

$$= 53.8\ m^3/s$$

$\triangle P_{tot}$ =-$\triangle P_{frict}$-$\triangle P_{area}$+$\triangle P_{veh}$+$\triangle P_{fan}$-$\triangle P_{stack}$

$$\triangle P_{frict} = \frac{\rho}{2} \sum_i \lambda \frac{l_i}{d_i} * Vair_i * |\ Vair_i\ |$$

where,

ρ= density of air (kg/m^3)

λ= friction factor

l_i= length of tunnel segment i

d_i= hydraulic diameter of tunnel segment i

$Vair_i$= mean air velocity in segment i(m/s)

$\triangle P_{frict}$= 51.82 Pa

$$\triangle P_{area} = \frac{\rho}{2} \sum_j \zeta_j * Vair_j * |\ Vair_j\ |$$

where,

ζ_j= resistance factors for area changes, depending on areas, shape of transition, and direction of flow

$\triangle P_{area}$= 1.225×4×$\frac{4}{2}$=9.8 Pa

$$\triangle P_{veh} = \frac{\rho}{2} \sum_k c_d * \frac{A_d}{A_{tun}} f_d * (Vveh_k - Vair_k) * |\ Vveh_k - Vair_k\ |$$

where,

c_d= drag coefficient, depending on vehicle type

A_d= vehicle cross section area(m^2)

A_{tun}= tunnel area(m^2)

f_d= factor>1 correcting the drag coefficient for the tunnel effects

$Vveh_k$= vehicle velocity (m/s) for vehicle k

$\triangle P_{veh}$= 1.225×0.48×0.6×1.1×(10-4)^2=7 Pa

$$\triangle P_{fan} = \rho \sum_m k_{fan} * (Vfan_m - Vair_m) * |\ Vfan_m\ |$$

k_{fan}= efficiency factor for the fan

$Vfan_m$= fan outlet velocity (m/s) for fan m

$\triangle P_{fan}$= 1.225×0.5×(84.6-4)×84.6=7146.5 Pa

$$\triangle P_{stack} = \rho * \triangle z * g$$

where,

$\triangle z$= change of altitude in segment (m)

g= acceleration of gravity (m/s^2)

$\triangle P_{stack}$= 1.225×9.8×$\frac{20}{2}$=120.05 Pa

$$\triangle P_{tot} = -\triangle P_{frict} - \triangle P_{area} + \triangle P_{veh} + \triangle P_{fan} - \triangle P_{stack}$$

$$= -51.82 - 9.8 + 7 + 4176.5 - 120.05 = 4001.83\ \text{Pa}$$

Now,

$$\text{Total Power of the Ventilation Fan} = \triangle P_{tot} * \frac{Q}{\eta f * \eta m}$$

$$= 4001.83 * \frac{53.8}{0.5 * 0.85} = 453000\ \text{W} = 453\ \text{KW}$$

6. Simulations and Results

The given length of the tunnel i.e., 4500 m is reduced to 225 m. Pyrosim is used to analyze heat release rate, temperature distribution and velocity profile. As size of the tunnel was reduced, air velocity was reduced from 4 m/s to 0.9 m/s using Froude scaling procedure. Different arrangements of ventilation fan were set for the tunnel.

6.1 Ventilation Fan at shortest Distance

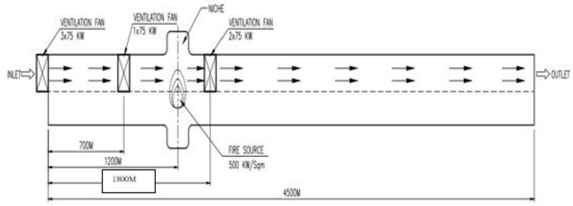

Figure 5: Arrangement of ventilation Fan at Inlet, 700 m and 1800 m

This diagram showed 2D arrangement of the ventilation fan and tunnel portal along with outlet. Fire source was at 60 m with heat release rate per unit area of 500 kW/m2. Ventilation fan position was at shortest distance and total power of 453 kW was arranged to measure characteristics HRR, velocity profile and temperature profile near fire source

In this simulation, blower fan was placed at shortest distance in order to avoid back layering. As air flows with more speed, it propagates fire. One ventilation fan is kept at distance of 35 m which in real scale was 700 m from the entrance. However, due to its placement nearer to the fire source, HRR increased. The main reason of increased HRR was the arrangement of ventilation fan closely, which significantly increases velocity of air flow inside the tunnel leading to fire that hampers the safety inside tunnel.

Boundary Conditions:

Heat Release Rate per Unit area=500 kW/m2 at 60 m
Blower Fan velocity at X=0 m=0.9 m/s
Blower Fan Velocity at X=35 m=0.4 m/s
Blower Fan Velocity at X=90 m=0.4 m/s

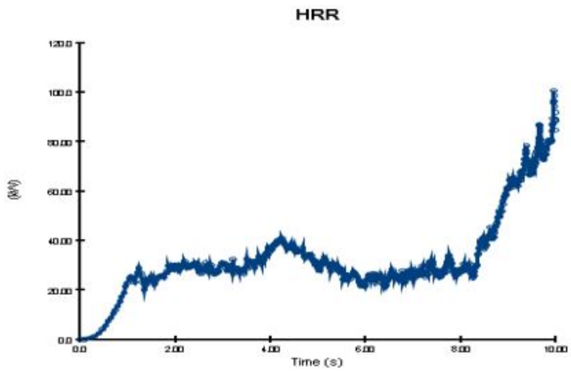

Figure 6: 10 HRR Vs Time

As a result of simulation done by keeping fire source at 60 m from the portal, heat release rate (HRR) has increased to 100 KW at 10 sec.

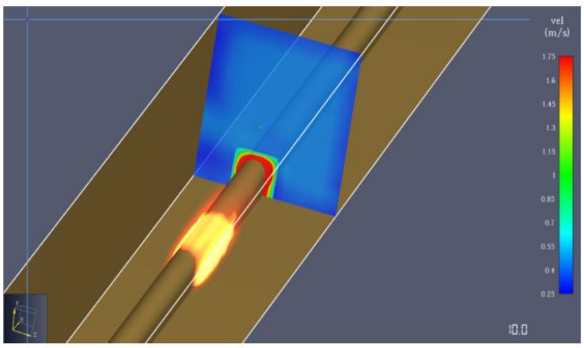

Figure 7: Velocity at X=62 m, max velocity=1.75 m/s

As fire source was at 60 m, which in real scale was 1200 m, flow of air at the nearest location from the fire source was checked. It ensured how velocity of air affects the HRR. At this location, velocity of air was 1m/s which is higher than the speed of the air of the blower.

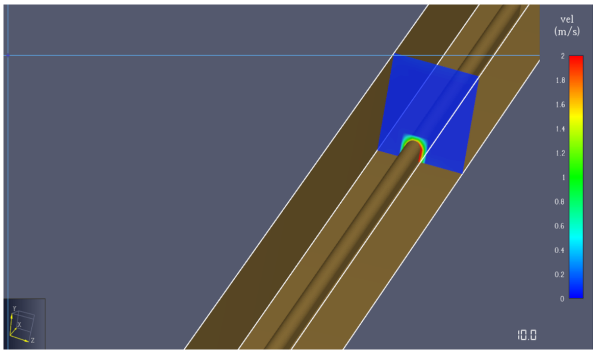

Figure 8: Velocity at X=62 m, max velocity=2 m/s

In this figure, Velocity of air at the distance 120 m i.e., 2400 m ranges from 0 m/s to 2 m/s. Average velocity of air was 1 m/s.

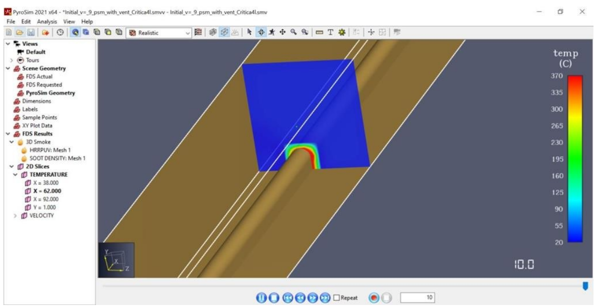

Figure 9: Temperature at X= 62 m 1

Temperature increased rapidly to 370 °C.

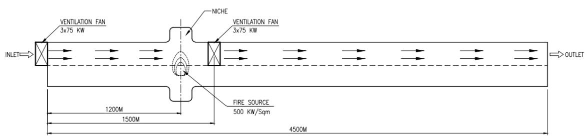

Figure 10: Arrangement of ventilation fan at Inlet and 1500 m

Boundary Conditions:

Heat Release Rate per Unit area=500 kW/m2 at 60 m
Blower Fan velocity at X=0 m=0.9 m/s
Blower Fan Velocity at X=75 m=0.9 m/s

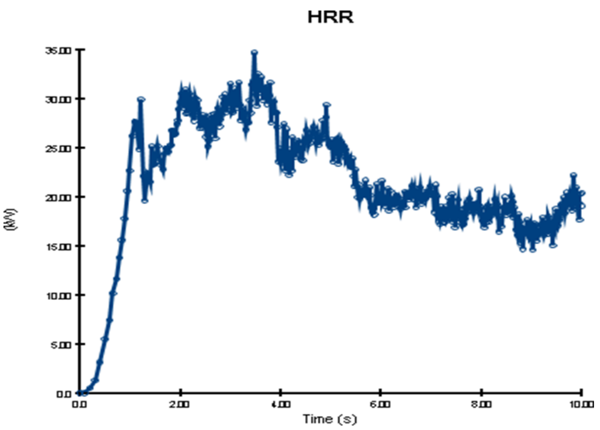

Figure 11: HRR Vs Time

This figure shows Heat Release Rate (HRR) 35 kW and is slowly reducing to 20 kW at 10 sec.

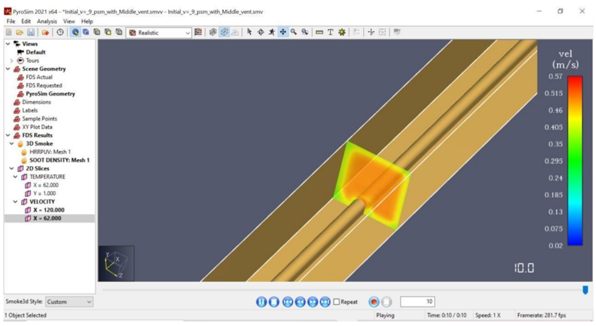

Figure 12: Velocity distribution at X=62 m, maximum velocity=0.57 m/s

It shows velocity distribution of air and was found to be 0.57 m/s maximum at 62 m i.e., 1240 m from the tunnel entrance. This shows velocity has decreased from the speed it was initially supplied as there is no ventilation fan placed in this location and near to it.

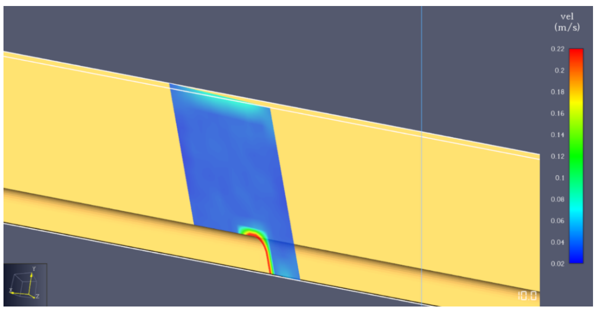

Figure 13: Velocity distribution at X=120 m, maximum velocity=0.22 m/s

In this figure,it is found that there is significant drop in the velocity of air.The main reason for this is ventilation fan is not placed in this location.

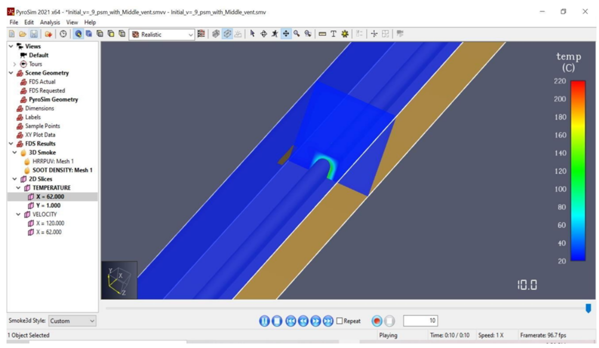

Figure 14: Temperature distribution at X=62 m, maximum Temperature=220°C

Temperature distribution in the surface of tunnel is shown which shows the maximum temperature at a distance of 1240 m is 220 °C.

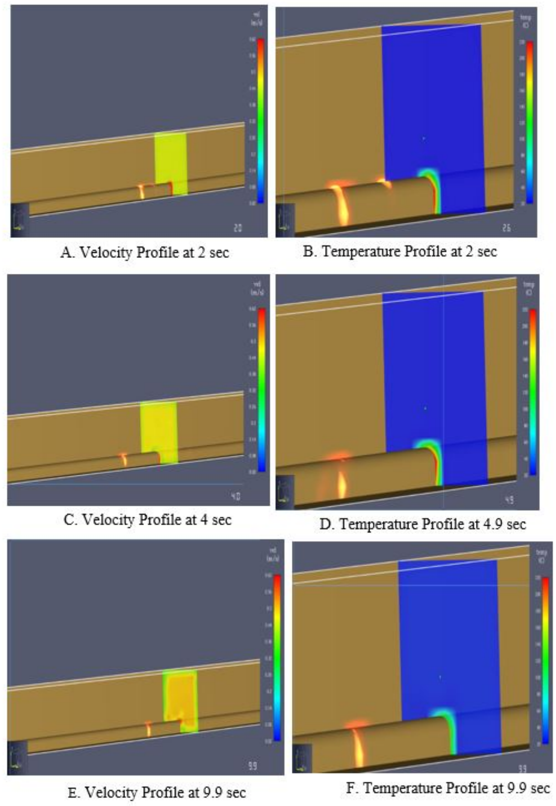

Figure 15: Velocity and Temperature Distribution at 62 m during Ventilation fan is placed at 0 m and 75 m.

Initially, when fire started at 2.0 sec as shown in Fig A, speed of air increased to 0.62 m/s and the maximum temperature recorded was 220 °C which can be seen in Fig. B. Similarly, at 4 sec as shown in Fig. C velocity of air drops to 0.46 m/s and temperature dropped down to 200 °C in this case. At 9.9 sec of fire, velocity of air is 0.40 m/s and temperature decreased to 140 °C. This clearly shows that the speed of air affects temperature as in the above figure, with decrease in the velocity of air fire propagation also drops with the temperature. Hence, fire propagation is affected by the speed of air as the physics involved in it is due to increased velocity of air, force is increased which ultimately increases momentum of flow and further increases Heat Release Rate (HRR). In addition to this, temperature also rises.

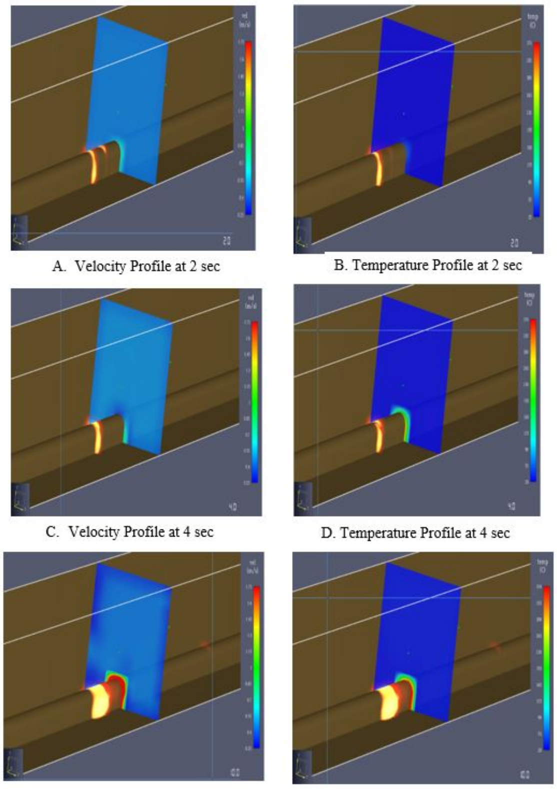

Figure 16: Velocity and Temperature Distribution at 62 m during Ventilation fan is placed at 0 m ,35 m and 90 m.

In this simulation, at 2 sec velocity of air is 0.85 m/s and temperature is 90 °C as shown in fig. At 4 sec, air speed increased to 0.95 m/s and temperature recorded was 230 °C.. Again at 10 sec, air velocity increased to 1.75 m/s and temperature had increased to 370 °C. This showed that with increase in velocity of air, temperature increases which is directly related to the HRR. So, HRR is shown in increasing trend in this simulation. The main reason for this is as temperature increases air density decreases which further induces to increase speed of air. Due to increased speed of air, it pushes fire to spread easily as momentum increases and it makes easier for propagation of fire. Fan load was calculated using formula and result obtained is 453 kW. As 4 m/s air velocity was required to get the proper air flow, same velocity was used in reduced scale; 0.9 m/s for simulation with different arrangements.

The simulation done showed increased heat release rate (HRR) when ventilation fan was placed at a distance of 0 m, 35 m and 90 m from the tunnel portal. The first simulation also showed increased speed of air flow when using ventilation fan at these distances enhances HRR. In 10 sec, HRR increased to 100 KW and velocity of the air at 120m was 0.9 m/s maximum. The highest temperature recorded was 370 °C.

Whereas, second simulation shows decreasing HRR when ventilation fan is placed at entrance of the portal and 75 m from the tunnel inlet. The result of the simulation is decreased speed of air from 0.57 m/s to 0.22 m/s at 62 m (1240 m) and 120 m (2400 m) respectively. The maximum temperature recorded in this simulation is 220°C at 2m away from the fire source.

Hence, ventilation fan at shortest distance increased heat release rate (HRR). It also increased the velocity from initial recorded velocity. Whereas, when the ventilation is placed far from the source, it will reduce the HRR and helps for emergency evacuation as the flow development retards in less time.

7. Conclusions

The objectives of the research were found and discussed as Results and Discussion via calculations and simulation method. The following are the major conclusions made through this research work:

a) In this design and analysis, tunnel was constructed in Solidworks with length, width and height of the tunnel 225 m, 0.19 m and 0.2 m respectively and simulation was done in PyroSim which is a Fire Dynamic Simulator.

b) Fan load required to obtain proper air flow was calculated mathematically and the value obtained was 453 kW. Speed of air obtained is 4 m/s and this velocity was used for simulation in PyroSim. In PyroSim, original scale was reduced using Froude Scaling Procedure and the value of velocity of air obtained was 0.9 m/s. Ventilation duct size is taken 900 mm during calculation of the ventilation fan.

c) Velocity obtained is tested by using it with blower fans. Simulation was done in PyroSim . CFD analysis was done in the 1st analysis i.e., at 0 m, 700 m and 1800 m. While in 2nd case, only in two distances i.e., 0 m and 1500 m.

d) When ventilation fan was kept at 0 m, 700 m, 1800 m and fire source was kept at 1200 m; velocity profile displays maximum velocity of air increased to 1.75 m/s due to low density of air caused by fire and the propagation increased to 100 kW and 370 °C at 10 sec of simulation.

e) When arrangement of the ventilation fan is done at 0 m and 1500 m from the portal; velocity profile shows maximum velocity of air was 0.57 m/s which ultimately decreased temperature inside to 220 °C and HRR to 20 kW in 10 sec.

f) The main reason for this was as temperature increases air density decreases which further increases speed of air. Due to increased speed of air, it pushed fire to spread easily as momentum increased and it made easier for propagation of fire.

g) As per the arrangement and result of simulation done in PyroSim, ventilation fan was set up with the best result obtained so as to control HRR and other parameters like velocity and temperature. Hence, velocity of air can be set in range of low velocity to control fire. And for SKDKHEP 86 MW, air velocity necessary to provide sufficient air is 4 m/s and it supplied with 453 kW ventilation fan. As per the result obtained, ventilation fan is arranged in the same way in SKDKHEP 86 MW and project is running continuously. Finally, no problem was encountered related to the ventilation fan arrangement as it was done as per the result of simulation.

8. Recommendations

This topic can be further investigated by considering following points mentioned below:

a) Simulation can be done by considering tunnel which is not completely breakthrough.

b) Study can be done based on ventilation duct with friction coefficient and arrangement.

c) Ventilation fan running at rotation can be arranged and simulation can be done.

References

[1] A López-de Arriba-Escribano, G Peris-Sayol, and F De-Kluijver-Benzaquén. On the operation of longitudinal ventilation systems in tunnels. *Research and Advanced Technology in Fire Safety*, page 375.

[2] Younggi Park, Youngman Lee, Junyoung Na, and Hong Sun Ryou. Numerical study on the effect of tunnel aspect ratio on evacuation with unsteady heat release rate due to fire in the case of two vehicles. *Energies*, 12(1):133, 2019.

[3] Peter Sturm, Michael Beyer, and Mehdi Rafiei. On the problem of ventilation control in case of a tunnel fire event. *Case Studies in Fire Safety*, 7:36–43, 2017.

[4] Francesco Colella, Guillermo Rein, Romano Borchiellini, R Carvel, Jose L Torero, and Vittorio Verda. Calculation and design of tunnel ventilation systems using a two-scale modelling approach. *Building and environment*, 44(12):2357–2367, 2009.

[5] Richard E Ray Jr and Abdelgbani Zigh. Cfd applications in tunnel ventilation analysis. 1999.