[1] Yousif Naif Almutai

[2] Yajue Wu

[3] Ahmed H Hafez

CFD Simulations to Study the Effect of Ambient Temperature on The Ventilation in a Metro Tunnel under Different Heat Release Rate

Abstract: - Urban and inter-urban rail transport provide the most favoured commuting option for millions of individuals globally. In spite of massive investments of public money to develop rail technologies and related infrastructure such as tunnel systems, there remain challenges to ensuring safety in this area. Railway tunnels present specific and serious fire safety issues, based on their length and restricted width. Smoke and other emissions from fires are a central factor to be considered in relation to both structural safety and survival where tunnel fires occur. Patterns of smoke flow can impact upon how efficient exhaust systems are, and for naturally ventilated systems, establishing shaft smoke flow patterns in tunnel fires is a significant requirement. While earlier work has considered this issue, it has been based on standard atmospheric conditions, while external temperatures can influence smoke movement properties, heat distribution, and the pattern of smoke flows within vertically constructed shafts. This study takes the case study of an underground metro tunnel, applying computational fluid dynamics through ANSYS Fluent to simulate and examine the effects of more extreme external temperatures for fires of 5 and 15 mw occurring in this environment.

Keywords: Fire Size , Subway Tunnel, CFD, Ventilation, Smoke Concentration, Harsh Weather.

I. INTRODUCTION

Tunnels are complex infrastructural components used to connect one area of a metro network to another, and are generally built in the form of a semi-circle or rectangle when considered in cross-section. In response to an increasing list of tunnel fires and other fire incidents in enclosed spaces with high movement of people lead to a desire to examine the conditions which accompany these fires and develop the most effective evacuation plans to maximize survival rates [1].

The effects of tunnel fires can have more severity than fires occurring outdoors due to the denser concentration of smoke and restricted room to move [2,3]. Fires and the disturbance of hazardous materials should be examined in various contexts and scales. While there are clear dangers in the immediate environment around a fire, in addition, toxic emissions move outward from this zone in accordance with airflows [4]. When a fire occurs, it causes extreme temperatures as well as emitting hazardous substances, and this places passengers who cannot exit the tunnel in danger. For example, A 2003 metro tunnel fire at Daegu in South Korea, cause by arson, killed 198 people [5].

Poorly considered evacuation plans may cause passengers to move towards fire or smoke and experience harm from poisonous emissions, radiation and high heat levels [6] [7].

Fires in tunnels lead to the accumulation of heated smoke under the ceiling which is challenging to exhaust because of the length and narrowness of typical tunnels: this heated smoke spreads symmetrically where no ventilation is present. Smoke is demonstrated to be the most significant danger to life where individuals cannot evacuate from a fire site, with 85% of deaths occurring due to toxic, high-temperature smoke [8,9].

When fires break out within the restricted space of tunnels, the tunnel's specific geometric features, slope, length and ramp strongly influence the fire' burn and smoke distribution properties [10]. The smoke movement and temperature distribution in horizontal tunnels have been widely investigated in the last decades [11], and it is clearly recognized that fires pose a severe challenge structurally and to human safety in tunnels. Improved understandings of temperature distribution and smoke propagation are a significant aim in the bid to improve structural fire protection, evacuation safety and smoke controls [12].

Numerous ventilation systems have been applied to lower the extent to which people are exposed to smoke while evacuating from a tunnel fire, either by causing smoke to exit the tunnel or maintaining it at a safe height above the people evacuating [13]. Such systems are divided into mechanical and natural ventilation [14], Forced ventilation systems use fans to extract or push smoke out of the tunnel. Natural ventilation with vertical wells in the roof of the tunnel take advantage of the buoyancy of hot smoke, which escapes through the shafts due to the stack effect and piston effect due to the movement of trains inside the tunnel.

Push-pull ventilation as an overall approach is used for ventilation of the tunnel section impacted by fire to control back-layering of smoke to the downstream side of the fire, which allowed for safe passenger evacuation to the upstream direction from the fire [15].

[1,2] Department of Chemical and Biological Engineering, University of Sheffield, UK.
[3] Aerospace Engineering Department, Cairo University, Egypt.
[1] ynalmutairi1@sheffield.ac.uk, [2] y.wu@sheffield.ac.uk, [3] Ahmed_7afez@eng.cu.edu.eg

Improved knowledge regarding smoke propagation and distribution of temperature has a central role in informing effective evacuation strategies and structural protection measures [16], leading to considerable research interest in this topic in earlier work.

Previous research has investigated the influence of tunnel shape, ventilation conditions, and ambient pressure on tunnel fire dynamics. However, with climate change, extreme weather, like heavy rainfall, occurs with increasing intensity and frequency, which has become a new norm [17]. Consequently, the probability of tunnel fires occurring under extreme weather conditions is increasing. Such dual disaster events could cause incalculable consequences. Therefore, it is necessary to understand the characteristics of tunnel fire during extreme weather to improve the capability of fire prevention, safety analysis, firefighting operation and emergency rescue

Parameters of climate, including harsh surrounding temperatures, are frequently not included in fire safety assessment. With most of the research being conducted in the Western world which considers a pleasant ambient temperature without taking in consideration the countries that are classified as desert weather as as climate change and its effect on global temperature increases therefore, Based on the above, this study investigates the impacts of extreme climate as presented in Saudi Arabia on tunnel ventilation, modelling this using ANSYS Fluent.

Therefore, the motivation for this research is to Explore this issue and study tunnel ventilation for different fire sizes 5 MW & 20 MW in harsh weather countries in Models such as Saudi Arabia utilizing Ansys fluent author hopes this work would improve knowledge and contribution which will reflect positively on fire safety management, fire safety codes and prescriptives alike.

.

II. METHODS

- **Theoretical Modeling**

Three Dimensional CFD simulation was done by ANSYS Fluent Finite volume package, using Reynolds Average Navier Stokes (RANS), turbulence model (k-ϵ) standard wall function to model the turbulence effects, and coupling energy equation with buoyancy-driven flow which was modeled by Boussinesq Approximation to predict the temperature distribution inside the tunnel.

Some assumptions were added to our model to simplify it as:

- Incompressible flow because the compressibility effect can be neglected in the ventilation application.
- Using the Boussinesq approximation to model the buoyancy-driven flow [18]
- Fire Source modeling by Volumetric Heat Source (VHS) with certain fire size, heat flux, and smoke generation rate [18]
- Neglect the air leakage from the tunnel, and the viscous heating at the tunnel walls [18]

By using the previous assumptions, a closed system of equations for the mathematical model [18] is shown as follows:

- Continuity Equation (Conservation of mass)

$$\nabla . \vec{u} = 0 \quad \textbf{(1)}$$

- Momentum Equation (Conservation of momentum)

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + \vec{u}.\nabla\vec{u}\right) = -\nabla \mathrm{p} + \nabla.\left((\mu + \mu_t)(\nabla\vec{u} + (\nabla\vec{u})^T)\right) + \rho g(1 - \beta(T - T_o)) \quad \textbf{(2)}$$

- Energy Equation (Conservation of energy)

$$\rho\left(\frac{\partial T}{\partial t} + (\vec{u}.\nabla)T\right) = \nabla.\left(\left(\frac{k_{th}}{c_p} + \frac{\mu_t}{\sigma_t}\right)\nabla T\right) \quad \textbf{(3)}$$

- Turbulence Model (k-ϵ)

$$\rho\left(\frac{\partial k}{\partial t} + \nabla.(\vec{u}k)\right) = \nabla.\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + P_k + G_b - \rho\epsilon \quad \textbf{(4)}$$

$$\rho\left(\frac{\partial \epsilon}{\partial t} + \nabla.(\vec{u}\epsilon)\right) = \nabla.\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\nabla \epsilon\right] + C_{1\epsilon}\frac{\epsilon}{k}(P_k + G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \quad \textbf{(5)}$$

Where:

- μ_t is the eddy or turbulent viscosity
- k_{th} is the thermal conductivity of air
- $\sigma_k, \sigma_\epsilon, C_{1\epsilon}, C_{2\epsilon}, C_\mu$ are the constant coefficients [18]

- **Buoyancy Driven Flow (Boussinesq Approximation) [16]**

The Boussinesq approximation is a way to solve thermal flow with small change of temperature, such as natural convection problems, without having to solve for the full compressible formulation of the Navier-Stokes equations.

$$\rho g = (\rho_0 + \Delta\rho)g = \rho_0 g - \rho_0 \beta (T - T_0) g \quad \textbf{(6)}$$

where $\beta = -\frac{1}{\rho_o}\left(\frac{\partial \rho}{\partial T}\right)_P \approx -\frac{1}{\rho_o}\left(\frac{\rho-\rho_o}{T-T_o}\right)$ is thermal expansion coefficient; ρ_0 is reference density = 1.225 kg/m^3; T_0 is reference temperature = 25 °C.

- **Numerical Modeling**

\- Geometry Modelling

The subway tunnel distance to be analyzed was 20 m long between two station buildings with height and width as H = 3 m, W = 3 m, with a heat source in the tunnel ground with dimensions 1 m X 2 m for 5 MW model and 1 m X 6 m for 15 MW; this layout is shown in Fig. 1, which shows an overall diagram of the tunnel model and boundary conditions.

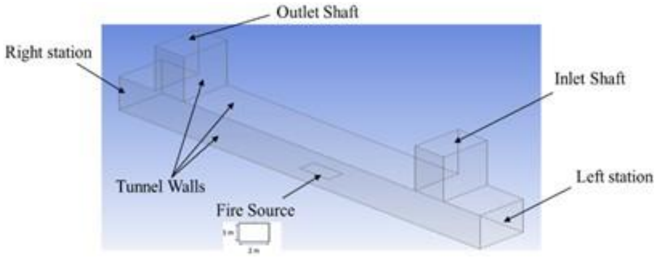

Figure 1: Tunnel Geometry Model with Boundary Conditions

– *Boundary Conditions:*

Boundary Conditions (BCs) for the tunnel are:

- **Inlet:** velocity inlet with a normal direction and magnitude equals velocity critical to avoid the back-layering with weather conditions (summer or winter) ambient temperature.
- **Outlet:** pressure outlet with atmospheric pressure with weather conditions (summer or winter) ambient temperature.
- **Tunnel walls:** are no-slip, stationary, and adiabatic walls.
- **Stations walls:** are modeled as symmetry BCs.
- **Fire Heat source:** with 5 MW & 15 MW.

– *Initial Conditions:*

- No smoke or zero smoke concentration inside the tunnel.
- Initial temperature 293 K or 20 °C.
- Initial zero velocities inside the tunnel.
- Initial zero-gauge pressure.
- Initial turbulent kinetic energy (k=0.016 m^2/s^2)
- Initial turbulent dissipation rate (ϵ=0.362 m^2/s^3).

– *Mesh Model & Mesh Independence Study:*

ANSYS Fluent meshing tool was used to generate the mesh for all cases with clustering mesh near walls to obtain the accurate values near walls due to the boundary layer effect. The tetrahedron element was used to create the mesh for the complex geometry.

The different mesh models by linear tetrahedron element as shown in the following figure with different mesh scale sizes as shown in the following table:

Table 1: Mesh Models for Mesh Sensitivity Study

	Max Element Size (m)	No#BL Layers	Total BL Layers Thickness (m)	Total Number of Mesh Elements
Case#1	0.5	10	0.05	43,387
Case#2	0.3	12	0.05	158,380
Case#3	0.2	15	0.05	501,745
Case#4	0.1	17	0.05	3,018,182
Case#5	0.075	20	0.05	8,124,105

In each model, the element size was changed and the number of boundary layer inflation was changed also to get the best mesh model to use in further cases. The following figures show the models #1 & #5

- Mesh #1 (43,387 Elements)

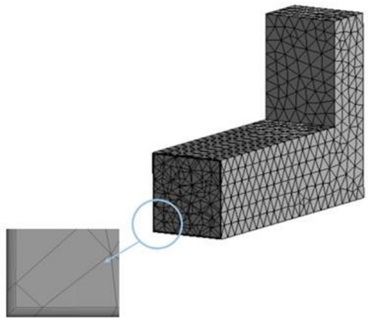

Figure 2: Mesh Model #1 (43,387 Elements)

- Mesh #5 (8,124,105 Elements)

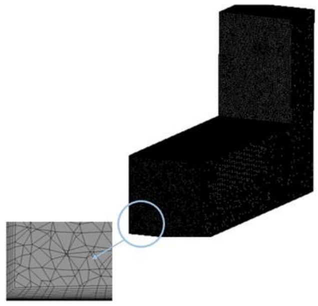

Figure 3:Mesh Model #5 (8,124,105) Elements

As shown from the previous Figs (2, 3), more refinement was done in the last model #5 than the first model to predict more accurate results and catch the velocity gradient near the walls.

- Mesh model #4 with 3,018,182 element was chosen as the best model:

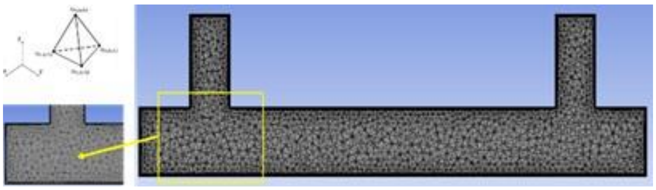

Figure 1: Mesh Model (3,018,182) elements with ANSYS meshing

- **Critical Velocity Effect CFD Check**

In first a different inlet fresh air velocities to check if the back layering will vanish and get sufficient space for evacuation from the tunnel in the inlet fresh air direction by ANSYS Fluent as CFD method:
Velocity & Temperature contours were checked at inlet velocity lower than critical velocity to make sure that ANSYS can predict the correct back-layering length as shown in the following Figs:

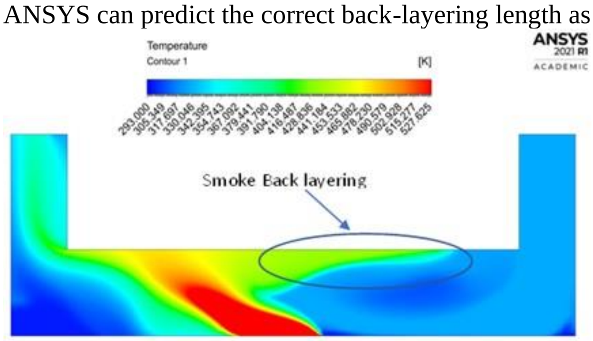

Figure 5: Temperature Contour Distribution at V < V_c

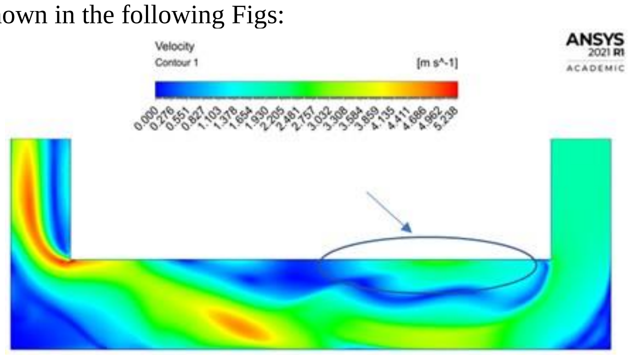

Figure 6: Velocity Contour Distribution at V < V_c

As shown in the previous Figs (5,6) the tunnel has a smoke bricklayer because a lower velocity than critical velocity was checked.

- **Critical Velocity Estimation by NFPA 502**

Critical velocity estimation was done by MATLAB code & algorithm regarding to National Fire Protection Association (NFPA 502) 2017 [3], [16]
MATLAB code is created to iterate on the following equations to get the best value of inlet fresh air velocity to prevent the back layering as follows:

$$V_c = K_1 K_g \left(\frac{gHQ}{\rho C_p A T_f}\right)^{\frac{1}{3}} \quad \textbf{(7)}$$

$$T_f = \left(\frac{Q}{\rho C_p A V_c}\right) + T \quad \textbf{(8)}$$

Where:

- V_c is the critical velocity (m/s) – K_1 is the froude number – K_g is the Grade Factor
- g is the gravitional accleration – H is the tunnel height – Q if the fire heat rate
- *ρ is the average density* – C_p is the specific heat of air – A is the inlet duct area
- T_f is the fire tempertaure – T is ambient tempertaure

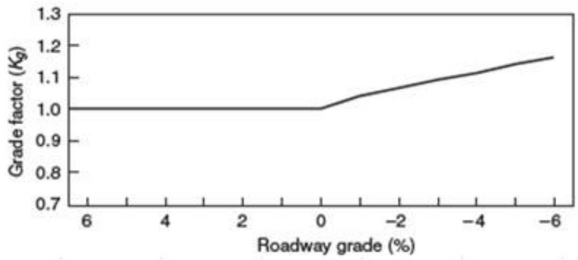

Figure 7: Grade factor Kg change with roadway grade (%) [16]

Grade factor (K_g) depends on roadway grade % so we can get it from the following figure [16]:
Froude Number (K_1) depends on Heat Rate from the fire so we can get it from the following table [16]:

Table 2:Froude Number K1 Change with Heat Rate from fire [16]

Q (MW)	K_1
>100	0.606
90	0.62
70	0.64
50	0.68
30	0.74
<10	0.87

MATLAB was used to generate a code to estimate the critical velocity of which prevent the smoke back layering in this tunnel in both cases, summer with ambient temperature (55 ℃) and winter with ambient temperature (5 ℃), and for 5 MW & 15 MW as shown in the following table (3). From the previous methods NFPA 502

Table 3: Critical Velocity & Average Fire Tempertaure Results

	Winter	Summer
5 MW		
$v_c (m/s)$	2.6511	2.6526
T_f (℃)	163.3428	242.1497
15 MW		
$v_c (m/s)$	3.1025	3.1038
T_f (℃)	410.9030	534.8322

Summer and winter effect on the critical velocity and average fire temperature inside the tunnel before a CFD study was done, so it so high motivation to make a coupled study on the effect of ambient temperature with the fire size change.

I. RESULTS

Different cases were done by ANSYS Fluent and all results are shown after 4 mins (240 secs) in the tunnel as:

1. Closed Right & Left Stations & All Fans are off
2. Open Right & Left Stations & All Fans are off
3. Open Right & Left Stations & Suction Outlet Fan is running **(More Realistic one)**

The more realistic model is the last one so the results for it will be shown here as the effect of the ambient temperature for winter and summer on the smoke concentration, Temperature distribution, and Velocity distribution.

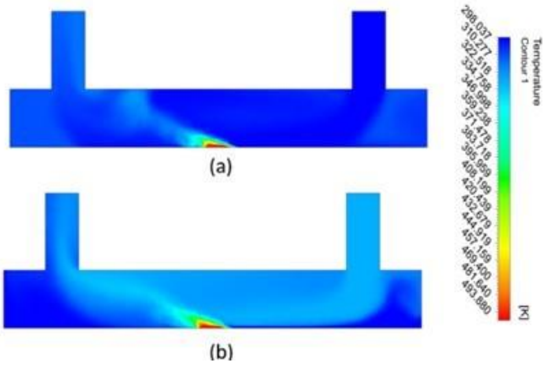

Figure 8: Smoke Concentration for fire size with HRR =5 MW (Visibility) inside the tunnel (a) Winter, (b) Summer

In Fig.8, the smoke concentration inside the tunnel was shown with almost the same inlet velocity for the winter and the summer as in Fig.(8a) the winter ambient temperature (5 ℃) makes a huge fluctuation in the smoke path because of the high buoyancy-driven flow effect with the high-temperature difference between the average fire temperature (163.428 ℃) and the inlet temperature (5 ℃).

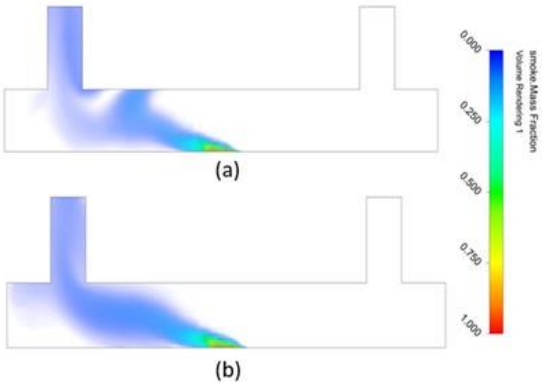

Figure 9: Temperature Distribution for fire size with HRR =5 MW inside the tunnel (a) Winter, (b) Summer

But on the other side in Fig.(8b), the fluctuation has vanished because the temperature difference is less than in the winter case with average summer fire temperature (242.1497 ℃) and the inlet temperature (55 ℃), but another issue is the smoke stored in the tunnel and didn't get out smoothly.

In Fig.9, the Temperature contour distribution inside the tunnel was shown with almost the same inlet velocity for the winter and the summer as in Fig.(9a) the winter ambient temperature (5 ℃) makes a huge fluctuation in the temperature distribution far away the fire source because of the high buoyancy-driven flow effect with the high-temperature difference between the average fire temperature (163.428 ℃) and the inlet temperature (5 ℃). Also, the winter ambient temperature decreases the temperature for the whole evacuation path from the inlet to the fire source center so it's great for evacuation.

But on the other side in Fig.(9b), the fluctuation has vanished because the temperature difference is less than in the winter case with average summer fire temperature (242.1497 ℃) and the inlet temperature (55 ℃), but the bigger issue is the average fire temperature is larger so it's worser than the winter case for evacuation.

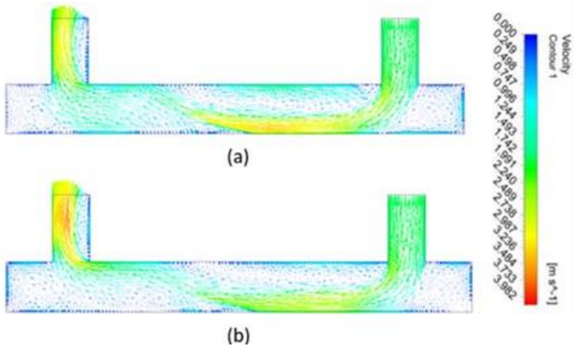

Figure 10: Velocity Vectors Distribution for fire size with HRR =5 MW inside the tunnel (a) Winter, (b) Summer

In Fig.10, the velocity vectors distribution inside the tunnel was shown with almost the same inlet velocity for the winter and the summer, the velocity vectors prove what were shown previously in Figs. (8,9).

- **The effect of different Heat Flux from the Heat Source on the tunnel airflow & smoke movement.**

The previous analysis section for the ambient temperature effect on the airflow and smoke inside the tunnel with 5 MW heat from the fire.

The summer case is the worst one and has more challenges for evacuation so the comparison between different fire HRR 5 MW & 15 MW will be done for the summer case only as it is worse than the winter one.

The heat flux was increased from the fire source to 15 MW rather than the standard case with 5 MW, and a comparison between them was done to study the effect on velocity, temperature, and smoke concertation (visibility) for the summer case.

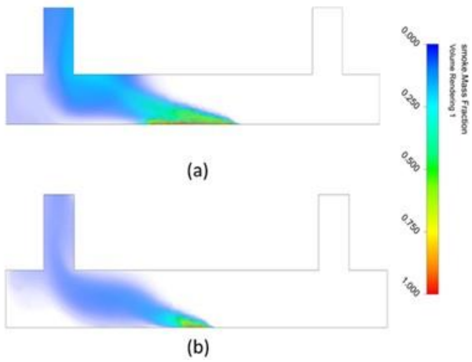

Figure 11: Smoke Concentration (Visibility) for the summer ambient temperature (55 °C) at (a) Heat Release from Heat source 15 MW, (b) Heat Release from Heat source 5 MW

As shown in the previous Fig.11, (a) is the smoke for summer with 15 MW heat flux but the suction fan can't extract all smoke from the tunnel, but a lot of smoke moves to the next station, however in (b) the smoke from heat source with heat flux 5 MW the suction fan can extract more than 15 MW case and a small amount of the smoke

So, the effect of the increasing heat source heat flux will decrease the effect of the suction fan to extract the smoke from the tunnel.

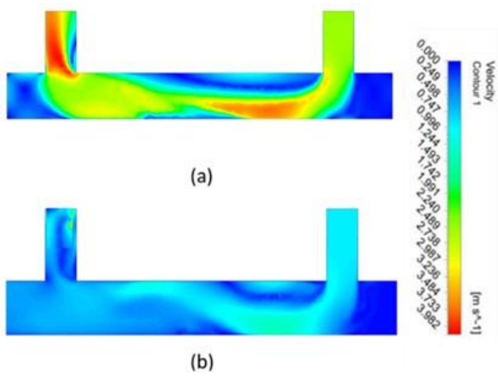

Figure 12: Velocity Contours Distribution for the summer ambient temperature (55 °C) at (a) Heat Release from Heat source 15 MW, (b) Heat Release from Heat source 5 MW

For Fig.12 the inlet velocity for the 15 MW case has larger inlet velocity (3.1038 m/s) than the 5 MW case with (2.6526 m/s) so the effect to push the smoke and exit the tunnel is large and it helps to remove more smoke. But it also has a negative effect if the amount of air increases the oxygen also increases so the fire growth rate will increase.

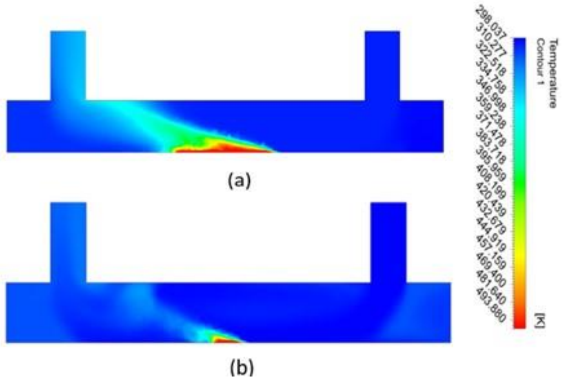

Figure 13: Temperature Contours Distribution for the summer ambient temperature (55 ℃) at (a) Heat Release from Heat source 15 MW, (b) Heat Release from Heat source 5 MW

As we said in the analysis of velocity contours the increases of inlet velocity will lead to more smoke exit the tunnel and improve the ventilation but also it has a negative side as shown in Fig.(13a), the case of 15 MW has more fire growth rate and fire area with high fire average temperature inside the tunnel (534.8322 ℃) rather than the other case 5 MW with average fire temperature (242.1497 ℃)
So, if the HRR increases it will lead to better ventilation but with high fire size and average temperature.

II. CONCLUSION & DISSCUSION

Nowadays, the most challenging study in the ventilation field is the ventilation is the closed system as underground subways tunnels; as mentioned before, the main aim of this research is to study the effect of harsh weather in this field in the Middle East with different ambient temperature as in the summer and winter seasons and get the worser one, especially in Saudi Arabia. And the effect of different of the fire size and HRR for 5 MW & 15 MW.
The first conclusion from the CFD study by ANSYS Fluent in this paper is the Critical velocity has a significant effect on smoke back layering in the tunnel with the effect of weather (summer and winter), smoke concentration is smoother in summer than in winter with less fluctuation due to low temperature difference compared to the winter case with the same inlet velocity and the same fire size 5 MW.
But also, the winter case has an advantage as a lower average temperature in the tunnel than summer so the evacuation in the winter will be more easier and with more thermal comfort for the passengers to exit the tunnel. So. the summer case is worse than the winter for evacuation so it was chosen to study the effect of fire size change from 5 MW to 15 MW, if the HRR is increased more smoke amount will be in the tunnel, and go to the next station.
The effect of the increasing heat source heat flux will decrease the effect of the suction fan to extract the smoke from the tunnel, but it also has a negative effect if the amount of air increases the oxygen also increases so the fire growth rate will increase.
Therefore, if the HRR increases it will lead to better ventilation but with high fire size and average temperature.

REFERENCES

[1] Ronchi, E. Evacuation Modelling in Road Tunnel Fires. Ph.D. Thesis, Polytechnic University of Bari, Bari, Italy, 2012.
[2] Amundsen, F.H. Studies of driver behaviour in Norwegian road tunnels. Tunn. Undergr. Space Technol. 1994, 9, 9–15.
[3] Amundsen, F.H.; Ranes, G. Studies on traffic accidents in Norwegian road tunnels. Tunn. Undergr. Space Technol. 2000, 15, 3–11.
[4] Vasilopoulos, K.; Lekakis, I.; Sarris, I.E.; Tsoutsanis, P. Large eddy simulation of dispersion of hazardous materials released from a fire accident around a cubical building. Environ. Sci. Pollut. Res. 2021, 28, 50363–50377.
[5] Zhang, T.; Zhang, Y.; Zhu, H.; Yan, Z. Experimental investigation and multi-level modeling of the effective thermal conductivity of hybrid micro-fiber reinforced cementitious composites at elevated temperatures. Compos. Struct. 2021, 256, 112988Xxxxxx
[6] Yan, Z.; Zhang, Y.; Guo, Q.; Zhu, H.; Shen, Y.; Guo, Q. Numerical study on the smoke control using point extraction strategy in a large cross-section tunnel in fire. Tunn. Undergr. Space Technol. 2018, 82, 455–467.
[7] Król, M.; Król, A.; Koper, P.; Wrona, P. Full scale measurements of the operation of fire ventilation in a road tunnel. Tunn. Undergr. Space Technol. 2017, 70, 204–213.

[8] J. Ji, H.X. Wan, Y.F. Li, K.Y. Li, J.H. Sun, Influence of relative location of two openings on fire and smoke behaviors in stairwell with a compartment, Int J Therm Sci 89 (2015) 23–33.
[9] Y. Alarie, Toxicity of fire smoke, Crit Rev Toxicol 32 (4) (2002) 259–289.
[10] M.S. Tomar, S. Khurana, Impact of passive fire protection on heat release rates in road tunnel fire: A review, Tunn. Undergr. Space Technol. 85 (2019) 149–159.
[11] Z. Gao, J. Ji, H. Wan, K. Li, J. Sun, An investigation of the detailed flame shape and flame length under the ceiling of a channel, Proc. Combust. Inst. 35 (2015) 2657–2664
[12] F. Tang, Z. Cao, A. Palacios, A study on the maximum temperature of ceiling jet induced by rectangular-source fires in a tunnel using ceiling smoke extraction, Int. J. Therm. Sci. 127 (2018) 329–334.
[13] António Galhardo, João Carlos Viegas, and P. J. Coelho, "The influence of wind on smoke propagation to the lower layer in naturally ventilated tunnels," Tunnelling and Underground Space Technology, vol. 128, pp. 104632–104632, Oct. 2022, doi: https://doi.org/10.1016/j.tust.2022.104632.
[14] H.X. Wan, Z.H. Gao, J. Ji, J. Fang, Y.M. Zhang, Experimental study on horizontal gas temperature distribution of two propane
[15] L. Zeng, Y. Yang, C. Liu, and M. Zhong, "Study on the optimal operation mode of ventilation system during metro double-island platform fire," Building Simulation, vol. 14, no. 3, pp. 779–792, Sep. 2020, doi: https://doi.org/10.1007/s12273-020-0692-4.
[16] F. Tang, Z. Cao, A. Palacios, A study on the maximum temperature of ceiling jet induced by rectangular-source fires in a tunnel using ceiling smoke extraction, Int. J. Therm. Sci. 127 (2018) 329–334.
[17] Ahmed Hussein Hafez, Tamer Heshmat Mohamed Aly Kasem, Basman Elhadidi, and Mohamed Madbouly Abdelrahman, "Modelling Three Dimensional Unsteady Turbulent HVAC Induced Flow", J. Adv. Res. Fluid Mech. Therm. Sc., vol. 87, no. 1, pp. 76–90, Sep. 2021.
[18] Li, Ying, and Haukur Ingason. 2010. "Study of Critical Velocity and Backlayering Length in Longitudinally Ventilated Tunnel Fires." Fire Safety Journal - FIRE SAFETY J 45 (November):361–70. https://doi.org/10.1016/j.firesaf.2010.07.003