INTERNATIONAL JOURNAL OF MULTIDISCIPLINARY RESEARCH AND ANALYSIS

ISSN(print): 2643-9840, ISSN(online): 2643-9875

Volume 06 Issue 08 August 2023

DOI: 10.47191/ijmra/v6-i8-56, Impact Factor: 7.022

Page No. 3827-3837

Numerical Simulation of Airflow Distribution in the Tunnel of Cairo Metro Line 3 Using CFD

Mohamed Abuelkassem Mohamed [1], **Elseman I. Abdelrasoul** [2], **Sayed R. Hamed**[3]*

[1,2,3] Department of Mining and Metallurgical Engineering, Engineering College, Assiut University, Egypt, 71515

[3]**ORCID:** https://orcid.org/0000-0001-9526-9480

ABSTRACT: One of the problems facing the underground tunnels is lacking enough fresh air for passengers inside the subway. Also, because of the friction of the train with the railways, high heat is generated. So, the use of computational fluid dynamics to distribute the required fresh air flow at the lowest possible cost is critical for the tunnels. In this research, Computational Fluid Dynamics (CFD) is used to perform 3D modelling and simulation for the tunnel of Cairo Metro Line No. 3. The standard k-e turbulence model was used in the CFD analysis to simulate the ventilation airflow in a 1075m tunnel length. The simulation reveals that the tunnel airflow rate induced by the speed of the fan is more desirable. Also, the addition of a jet fan causes an eddy current that improving the efficiency of tunnel ventilation, thereby greatly reducing ventilation time, and increasing efficiency in cases where the diameter of the tunnel equals 15 m, the speed of the fan is 1480 r.p.m, and the air flow rate is 80-120 m^3/s. This led to improve fan speed efficiency and airflow distribution in the tunnel. Similar way of simulation is used for road tunnel ventilation.

KEYWORDS: Tunnels, Ventilation systems, Numerical simulation, Computational Fluid Dynamics (CFD)

NOMENCLATURE

ρ, Is the fluid density, Kg/m^3
μ, Is the molecular(dynamic) viscosity, Pa.s
$\vec{f}$, Is the body force per unit mass, N/kg
k, Is the turbulence kinetic energy, KJ/ kg
u_{avg}, Is Reynolds averaged velocity vector, m/s
ε, Is the turbulence kinetic energy dissipation rate, m^2/s^3
δ_{ij}, Are the components of the velocity vector in the x_i and x_j directions
λ, Is the coefficient of bulk viscosity NS/m^2
x_i, x_j, Are denote mutually perpendicular coordinate directions
p , Is pressure Pa
C_μ, Constant
t, Is the time, sec
τ_{ij} , Is the viscous stress tensor
$\vec{V}$, Direction of tunnel velocity, m/s
μ_t, The turbulence viscosity, m^2/s
$\bar{\bar{\tau}}$, Is the transport coefficient
V_i, V_j, Are vector velocities m/s
r.p.m, Revolution per minute

1. INTRODUCTION

Today, online monitoring systems can detect the tunnel airflow volume and velocity. However, there are different techniques for studying the non-uniform distribution of airflow and the sensor-measured velocity, as stated by Parra et al[1, 2]. As such, it is quite necessary to study numerical simulation of airflow distribution in the tunnel at present. So, mathematical models based on Computational Fluid Dynamics (CFD) are becoming popular as predictive tools for airflow distribution in the tunnel, as stated by Richardson and Harlow [3, 4]. To further increase the use of these models in the design methods for improving the efficiency of tunnel ventilation, they often use general-purpose computer programmes and require extensive training [5]. Also, they used the buoyancy K-omega SST model to represent the turbulent transport, component models for jet fans, and ventilation ducts of airflow distribution in the tunnel, as stated by Tu et al [6].

Vega et al.[7] found that (CFD) is considered very efficient for improving the efficiency of tunnel ventilation. Passi et al. and Hofer et al. [8, 9] they said that tunnel ventilation is a crucial component of road and subway designs. Caliendo et al. and Li et al.[10, 11] also, they said that mechanical ventilation and natural ventilation are the two main types of tunnel ventilation. Also, Wang et al.[12] recommended using (CFD) in tunnels to help the jet fans exhaust polluted air. Myrvang and Khawaja [13] used (CFD) for a minimum number of fans to analyze how air develops inside a tunnel from the inlet to the outlet. Betta et al.[14] performed a numerical simulation of the jet fan's airflow distribution in the tunnel using (CFD).

Yang et al.[15] used numerical simulation for the operation of pumps and fans by ratio ranging between 59.5 and 73.4% to improve efficiency in tunnels. In 2014, zhao and co-workers [16] a postulated that few have used experimental and theoretical techniques to quantitatively study the average velocity distribution in circular and rectangular tunnels. Kamal et al.[17] used a 2D finite element commercial software, of airflow distribution in the tunnel of Cairo Metro Line No 4. A model for the development of air flow and the effect of jet fans for longitudinal and curved tunnels using computational fluid dynamics CFD, has been used by Jacques and Wauters [18] and Wang et al.[19].

Numerical simulation of airflow distribution in the tunnel is essential in the case of (CFD) to improve distribution of speed and pressure in an appropriate manner from inside a tunnel inlet to the outlet. Hence the paper covers highlighted research gap, which has been ignored by some researchers. Hence, the CFD modelling by using ANSYS FLUENT software has been applied for numerical simulation of airflow distribution in the tunnel. So, the case study would be in the tunnel of Cairo Metro number 3 with a length of 1075m.

Objectives of the work:

In this paper, it is intended to study numerical simulation of airflow distribution in the tunnel of Cairo Metro Line 3 to improve the ventilation system , the paper aims to:

- Apply computational fluid dynamics (CFD) modelling using ANSYS FLUENT software for numerical simulation of airflow distribution in the tunnel.
- Determine the maximum speed and appropriate pressure inside the tunnel for air flow in an appropriate manner.
- Reduce ventilation time and increasing fan efficiency to improve airflow distribution in the tunnel.
- Determine the best curve of velocity distribution inside the tunnel.

2. LITERATURE REVIEW

Many studies have been conducted on modelling and simulation of tunnel ventilation systems.

Adjiski and Despodov (2019) utilized a 3D computational fluid dynamics (CFD) model with "ANSYS FLUENT" software to evaluate various factors, such as face velocity and turbulent airflow patterns, to optimize ventilation system efficiency. They found the model was able to reduce energy consumption and increase auxiliary ventilation system[20].

McPherson (1993) and Acuña and Lowndes (2014) used computational fluid dynamics in the layout of the underground ventilation system in mines and tunnels. They have found that the underground network of tunnels is connected to the surface through vertical shafts or ramps, acting as inlets and outlets for ventilation and exits for personnel and equipment. They concluded that using CFD improves the efficiency of tunnel ventilation [21, 22].

Hamed et al. (2023) used computational fluid dynamics (CFD) to design and calculate the tunnel ventilation system for Cairo Metro Line 3. They intended to study the numerical simulation of airflow distribution in the tunnel of Cairo Metro Line 3 to improve the ventilation system. As a result, by using (CFD) they found that the determination of air quantity, location of fans, and regulators to distribute the required fresh air flow at the lowest possible cost has lead to improve efficiency of the tunnel ventilation[23].

Wang et al. (2012) recommended using computational fluid dynamics (CFD) in tunnels to help the jet fans exhaust polluted air[12]. Also, Myrvang and Khawaja (2018) used (CFD) for a minimum number of fans to analyze how air develops inside a tunnel from the inlet to the outlet. They found that pumping huge amounts of fresh air into the tunnel and controlling the speed of the fans led to lower power consumption and an increased rate of ventilation[13]. And Betta et al (2010) performed a numerical simulation of the jet fan's airflow distribution in the tunnel using (CFD). They found an increase in fresh air for passengers inside the subway[14].

Lewus (2023) demonstrated that using program CFD (ANSYS FLUENT) of tunnels greatly affects the airflow field inside the tunnel. He found that the air flow from the jet fan spreads more quickly and less energy consumption and high fan efficiency[24]. Additionally, in (2023) Jafari et al investigated the effect of a moving vehicle on the airflow field in curved and straight tunnels by using numerical simulation (CFD). They found that the speed of the fan, has been increased in straight tunnels by using numerical simulation[25].

3. MODEL DESCRIPTION

3.1. The tunnel and train

The previous studies showed that the numerical analysis method is sufficiently accurate, thus it was used to simulate the distribution of airflow in the tunnel[1]. In this paper, the tunnel investigated is part of the tunnel of Cairo Metro Line No.3. Fig. 1 shows the dimensions of the tunnel and train. The maximum width and height of the tunnel cross section are 9.8 m and 6.3 m, respectively. The large arch radius is 4.9 m. The maximum width and height of the train are 3.3 m and 3.9 m, respectively as in shown Figure.1. The speed of the train ranges from 80–120 km/hour and the total length is 137.4 m. There is a numerical model of the tunnel with a metro train inside by software CFD and the length of the tunnel under study is 1075 m, which is shown in Figure.2.

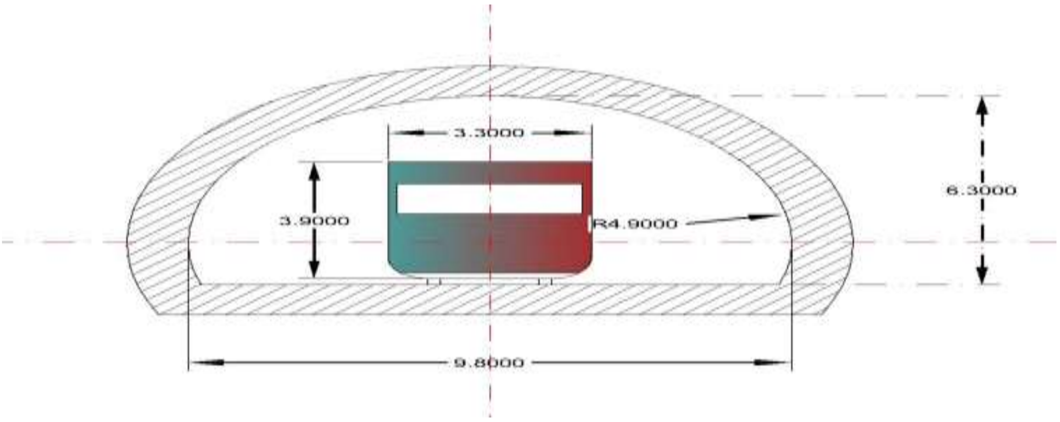

Figure.1. Cross-section of the tunnel

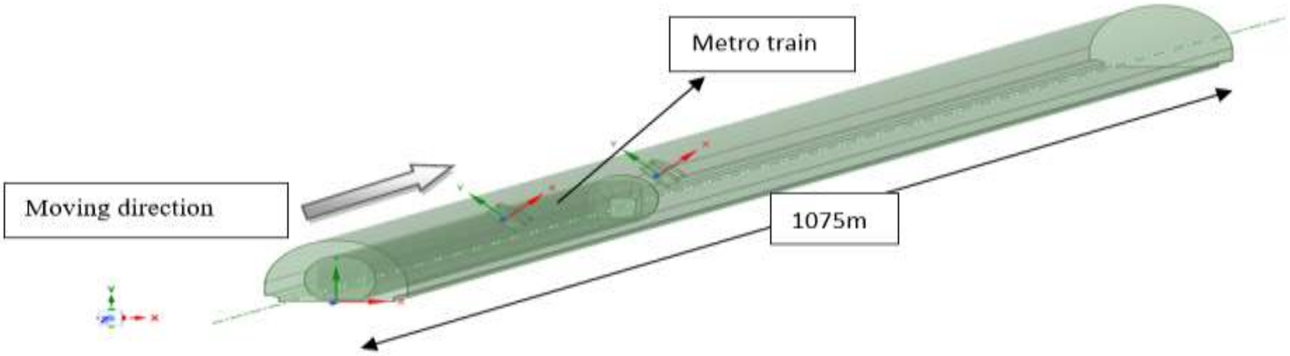

Figure.2. Numerical model of the metro tunnel with a train inside

4. NUMERICAL SIMULATION

4.1. Governing equations

The numerical analysis of Greater Cairo Metro tunnels in this research will include airflow distribution in the tunnel of Cairo Metro Line No.3.

The governing equation determines whether the flow in the tunnel is ideal gas, Rayleigh flow, Fano flow, viscous or inviscous, laminar, or turbulent, compressible, or incompressible. Any flow, whether internal or external, can be described by three fundamental governing equations: the continuity equation (conservation of mass), the conservation of momentum (to describe the 3D change in shear forces and pressure), and the conservation of energy (to describe the heat transfer according to the pressure difference).

The following assumptions were adopted when creating the mathematical model: the airflow is incompressible, the tunnel wall is not adiabatic, there are passengers, and smoke and dust are disregarded in order to determine the least amount. So, according to the fluid dynamics theory, the following Equations (1), (2), (3), (4), (5), (6), and (7) affiliated with the Ansys company are used to describe the airflow in tunnels. Also, the energy equation is not activated in order to maintain the temperature inside the tunnel and determine the air quantity to distribute the required fresh air flow fans under different speeds in the tunnel (r.p.m).

- **Conservation of mass:**

The first term of the equation indicates that if the flow is compressible so the density of the flow changes with time.

$$\frac{\partial\rho}{\partial t} + \nabla\cdot\left(\rho\vec{V}\right) = 0 \tag{1}$$

The second term in the equation indicates the variation of the density in x-direction and y-direction and z-direction.

$$\frac{\partial\rho}{\partial t} + \rho\left(\nabla.\vec{V}\right) = 0 \tag{2}$$

- **Conservation of momentum in 1D or (Navier stokes equation).**

The Navier-Stokes Equation, which also refers to the equation for the conservation of linear momentum, is frequently used in computational fluid dynamics (CFD) to refer to both of the momentum and continuity equations as well as the energy equation. To represent the turbulent transport, component models for jet fans, and ventilation ducts of airflow distribution in the tunnel, as seen in equation (3).

$$\rho\frac{D\vec{V}}{Dt} = -\nabla p + \Delta\cdot\bar{\bar{\tau}} + \rho\vec{f} \tag{3}$$

In order to be able to use a eulerian description equation can get parameter $\vec{f}$ which is the body force per unit mass. And τ is the viscous stress tensor for Newtonian fluids viscous stresses only. Also, it is known that $\bar{\bar{\tau}}$ needs to be symmetric for velocity and density. Therefore, the friction and heat generated by the train during its movement inside the tunnel at different speeds can be known from Equations (4),(5),(6), and (7).

$$\rho\left[\frac{\partial\vec{V}}{\partial t} + (\vec{V}\cdot\nabla)\vec{V}\right] = -\nabla p + \nabla\cdot\bar{\bar{\tau}} + \rho\vec{f} \tag{4}$$

$$\tau_{ij} = \mu\left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}\right) + \lambda\left(\nabla.\vec{V}\right)\delta_{ij} \tag{5}$$

$$\lambda + \frac{2}{3}\mu = 0 \tag{6}$$

$$\tau_{ij} = \left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} - \frac{2}{3}\left(\nabla.\vec{V}\right)\delta_{ij}\right) \tag{7}$$

4.2. Turbulence modeling

The standard model k-Ɛ was used to calculate the turbulence and diffusion of the airflow. Also, to obtain the velocities and pressure inputs and to define the wall zones, the temperature and rotational rpm of two fans, the velocity of the train, and the viscosity and relative humidity of the air inside the tunnel and outside the tunnel can be calculated from Equations (8) and (9). The sources for k and Ɛ are given by:

k equation

$$k = 1.5\left(\mathrm{Iu}_{\mathrm{avg}}\right)^2 \tag{8}$$

Ɛ equation

$$\varepsilon = C_\mu\frac{k^2}{\mu_t} \tag{9}$$

4.3. Boundary conditions

The aims of the boundary conditions of the tunnel are as follows[7].

- The velocity and scalar characteristics of air flow at intake borders are defined.
- The total pressure at flow inlets is defined.
- In compressible flows, mass flow intake is used to specify a mass flow rate at an inlet.
- In the event of backflow, additional scalar variables are used to specify the static pressure at flow outlets using the pressure outlet.

So, in this case, to increase the velocity of air flow entering the tunnel, two suction fans are used on each side of the tunnel, one from the inlet by 90 m and the other from the inlet by 190 m. Also, the velocity of the train is approximately 120 km/h (33.33 m/s).

To determine the flow rate of each fan and the pressure drop caused by each fan in another case. The train's velocity will be 126 km/h (35 m/s), and the length of the tunnel will be 1075 m, so the time the train will move in the tunnel will be 20 seconds. So, the minimum time the fluid properties will change is 20 seconds when the velocity of the train is 126 km/h (35 m/s). So, the problem in the tunnel is a lack of fresh air for passengers inside the subway. As a result of the increase in the maximum speed of the train. So, as a result of the friction of the train with the railways inside the tunnel, more heat is generated.

4.4. Solver settings

CFD (ANSYS FLUENT) has been selected for numerical simulation of airflow distribution in the tunnel of Cairo Metro Line 3 induced by single and double trains. In this study, 3D, unstable, compressible, turbulent models, k-ε turbulence, and Navier-Stokes equations have been used. The k-ε turbulence and Navier-Stokes equations model is commonly utilized in the numerical simulation of trains passing through tunnels, as stated by Deng et al. (2020) and Izadi et al. (2020) [26, 27]. Such method has been validated by many researchers[28, 29]. Table 1 gives the detailed solver settings for the numerical simulation of airflow distribution in the tunnel.

Table1: Detailed solver settings of the numerical simulation

Parameters	Setting
Turbulent model	*k-ε* model
Governing equations	Conservation of mass and Navier stokes equation
Gradient	Green-Gauss cell-based

From Table (1), it is noticed that the k-ε model, Conservation of mass, Navier-Stokes equation, and Green-Gauss cell-based numerical simulation of trains passing through tunnels have been used. And the k-ε turbulence and Navier-Stokes equations model is commonly utilized in the numerical simulation in CFD software.

4.5. Mesh analysis

Meshes around the tunnel body were discretized into hexahedral grids from type (poly hexacore). Because the tunnel is a two-directional tunnel (1075 m in length) and the length of the train is 183.2 m, at each node, the pressure, fluid density, air velocity, and temperature of the tunnel are calculated through the node located in the center of the elements. The programme distributes the solution to each node and then divides the whole body. The tunnel's hexahedral grid has a length of 0.1 m along the x axis and a maximum size of 0.2 m in the y and z directions, which leads to a lot of element meshes as shown in Figure. 3.

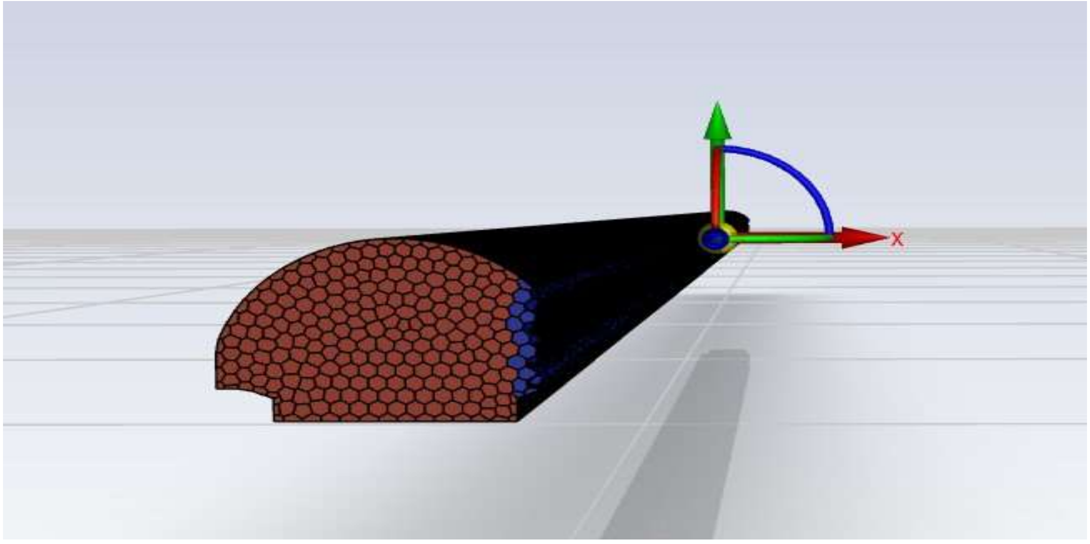

Figure.3. Meshes

5. RESULTS AND DISCUSSION

The residuals and iteration processes are derived from mesh analyses. The iterations are the number of calculation times for the accuracy of the solution, meaning that the more it increases, the residuals decrease. That is, the relationship between the iterations and the residuals is an inverse relationship, and this gives the correct values and high accuracy in the solution. As the residuals represent the error values resulting from the numerical analysis, the more iterations, the closer get to the correct values of the error indicator at the residuals. This gives correct values and high accuracy in the solution, as shown in Figure.4.

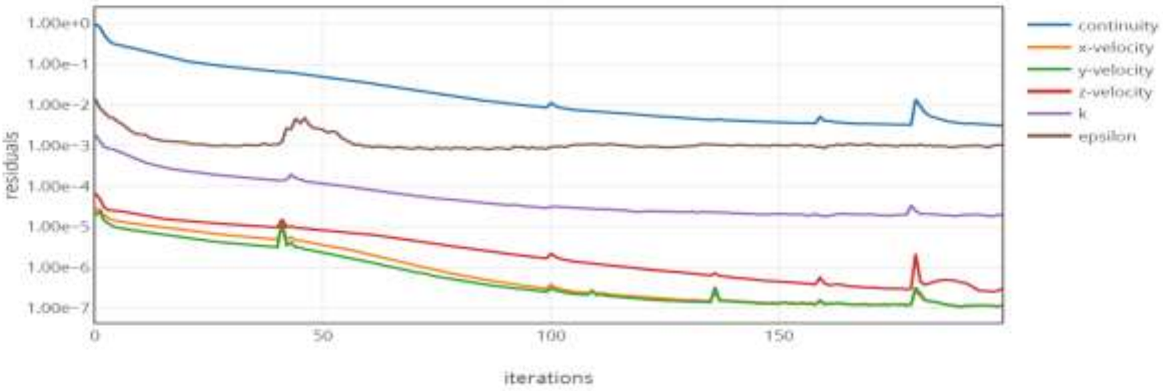

Figure. 4. Relationship between residuals and iterations

From Figure. (4), It can be seen that the higher the number of iterations, the lower the error values in residuals at 100 iterations. This demonstrates the accuracy of the solution resulting from the numerical analysis process, saving time and energy. also gives correct values and high accuracy in the solution.

The comparison of the relation between time of train movement and pressure in the tunnel until the train reaches the end of the tunnel is presented in Figure.5. The train speed is maintained until it reaches 80 seconds. This led to an improvement in the tunnel's atmosphere.

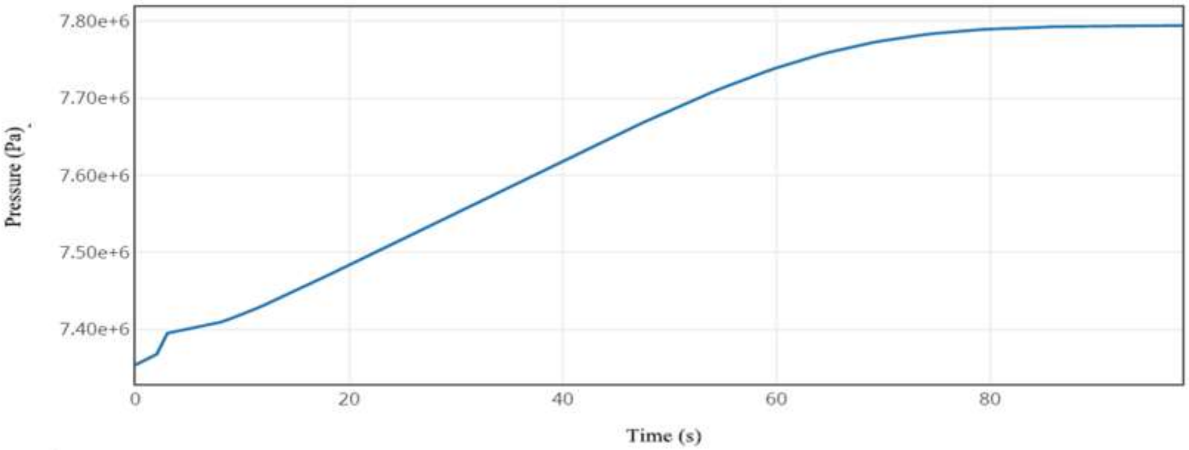

Figure. 5. Relation between time of train movement and pressure in the tunnel

From Figure. (5), it can be noticed that the train speed is maintained until it reaches 80 seconds. As a result, the amount of ventilation exiting the fan at equal speeds along the tunnel is sufficient. This indicates the accuracy of the solution and the correct results of the mesh analysis.

5.1. Velocity variation

Velocity variation is a process called mesh optimization, which is a relationship that relates the fan's speed with the number of elements in the cells, to flow the air inside the tunnel at equal speeds along the tunnel. Figure.6. illustrates the fan(s) speed, r.p.m., in relation to the number of elements, millions, in the cells to determine the value of the fan(s) speed at a specific element of the cells.

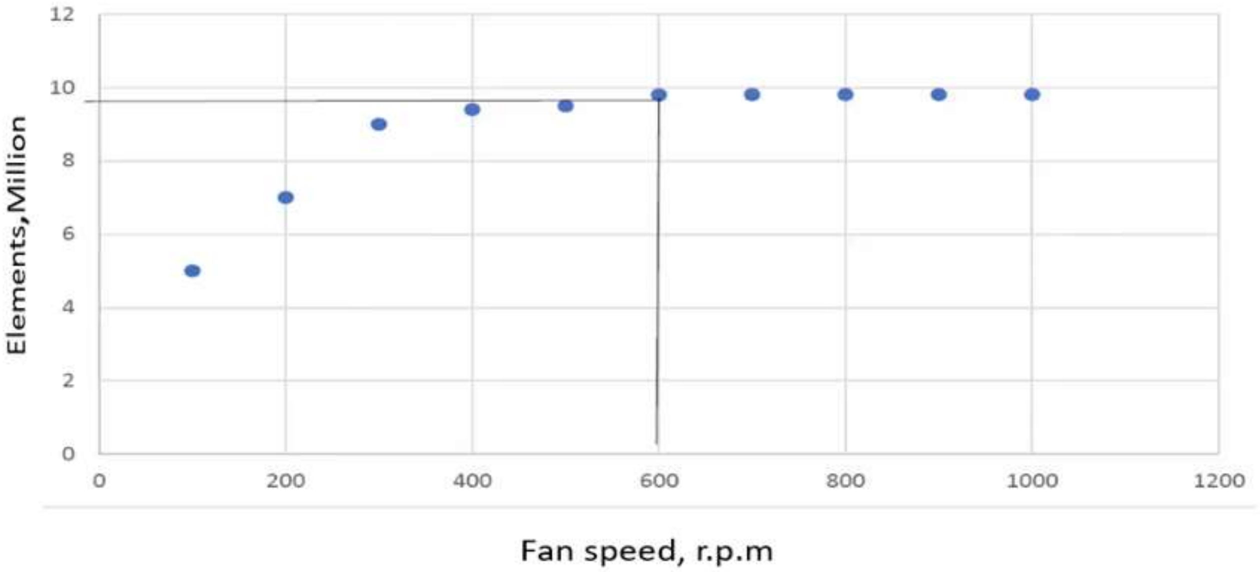

Figure.6. Velocity variation for fans

From Figure. (6), it can be found that the speeds change from 200 to 1200 r.p.m. At 9.5 million cells, the actual speed (600 r.p.m) is attained with acceptable accuracy. This led to improving fan speed efficiency and airflow distribution in the tunnel.

Different values of velocity magnitude to determine the best velocity simulation of airflow distribution in the tunnel are shown in Figure.7. Contours are at the vertical center plane of the tunnel with a traditional jet fan ventilation system. The magnified view shows the details of the airflow field, and the velocity magnitudes are in m/s.

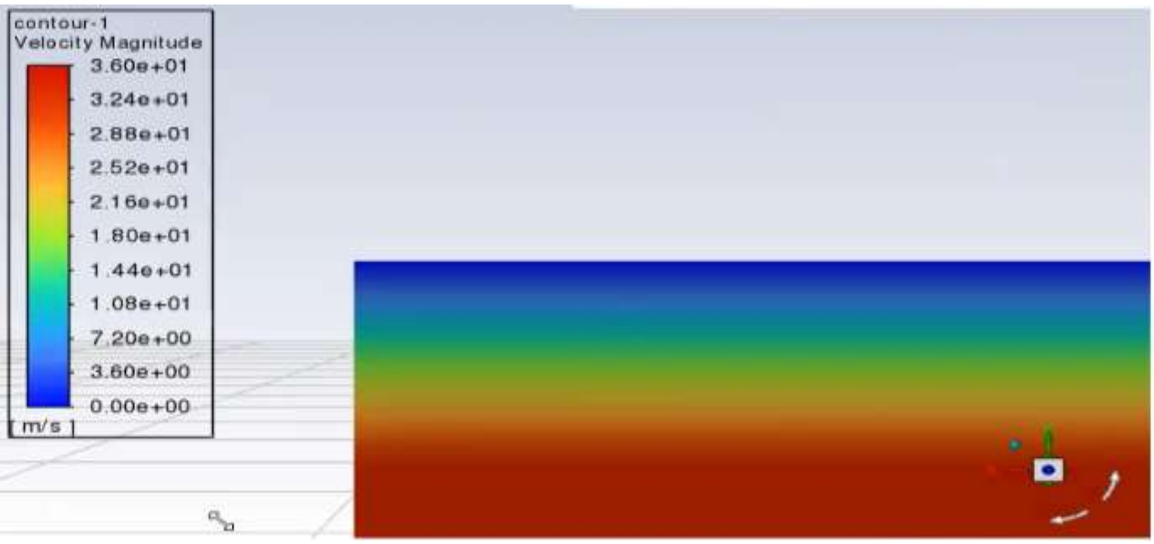

Figure. 7. Layers of velocity magnitude at the vertical center plane of the tunnel

From Figure. (7), it can be noticed that the change in velocities as a result of the different layers of the fluid in relation to the length of the tunnel, from 0.0 m/s to 3.6e+00 m/s, and velocity variation for fans. This leads to reducing fan energy costs, including their operating performance.

5.2. Pressure Field

The dispersion of ventilation rate pumped by the fan into the ventilation tunnel at the moment the train enters the tunnel is illustrated in Figure. 8.

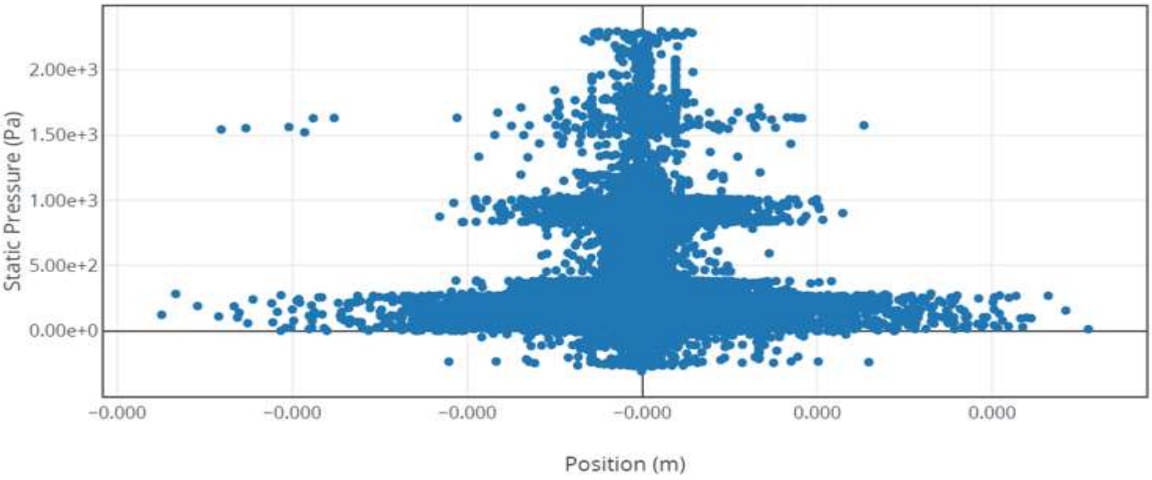

Figure.8. Shows a 3D of the dispersion of ventilation rate according to static pressure

From Figure. (8), It is noticed that the largest amount of air dispersion starts at the moment the train enters the tunnel and returns to normal after train departure. Where (-0.000 Pascal) is the time when the air meets the fan when the train passes by, and there is a significant change in pressure.

Figure. 9. shows different values of pressure magnitude to determine the best velocity simulation of airflow distribution in the tunnel. Pressure magnitude contours at the vertical center plane of the tunnel with a traditional jet fan ventilation system. The magnified view shows the details of the airflow field.

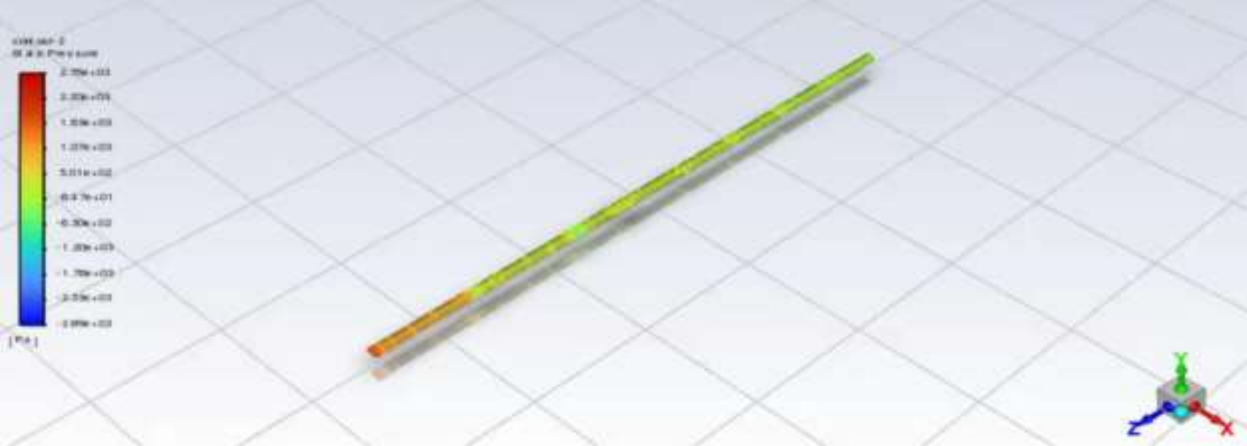

Figure. 9. Layers of pressure magnitude, and the pressure magnitudes are in Pascals.

From Figure. (9), it is noticed that the change in pressure as a result of the different layers of the fluid in relation to the length of the tunnel starts at -2.89e+03 pascal and ends at 2.76e+03 pascal. This improves the performance of the fan, leading to low energy consumption.

5.3. Effect of inlet airflow velocity on the studied tunnel

Tunnel ventilation is essential for either a curved or longitudinal tunnel. Curved tunnels are more problematic than longitudinal tunnels, so they need continuous ventilation, and the problem of the unavailability of ventilation as a result of high heat generation in the tunnel has a detrimental effect on passengers. Using the modeling, all the ventilation strategies have been analyzed and it has been demonstrated that the actual ventilation system, if fully operational, is able to provide sufficient ventilation in all zones. The results of the effect of tunnel dimensions by changing the diameter and length of the tunnel with fan speed (r.p.m) differences are given in Table 2.

Table 2: Impact of dimensions on the tunnel ventilation

Length of tunnel, m	Diameter of tunnel, m	Speed of fan, r.p.m	Air flow rate m^3/s
260	5	440	37.5
537.5	10	880	80
1075	15	1480	80-120

From Table (2), it can be found that the velocity distribution inside the tunnel for air flow is in an appropriate manner at a speed of 1480 (r.p.m) and the air flow rate is 80–120 m^3/s . It is the best way to improve the efficiency of tunnel ventilation and study the dimensions of the tunnel.

Figure. 10. Presents the relationship between critical velocity, heat release rate, and tunnel diameter. It is the best curve for velocity distribution inside the tunnel for air flow in an appropriate manner at a fan speed of 1480 (r.p.m), tunnel length of tunnel of 1075m, and an air flow rate of 80-120 m^3/s . And the addition of a jet fan causes an eddy current that improving the efficiency of tunnel ventilation, thereby greatly reducing ventilation time, and increasing efficiency.

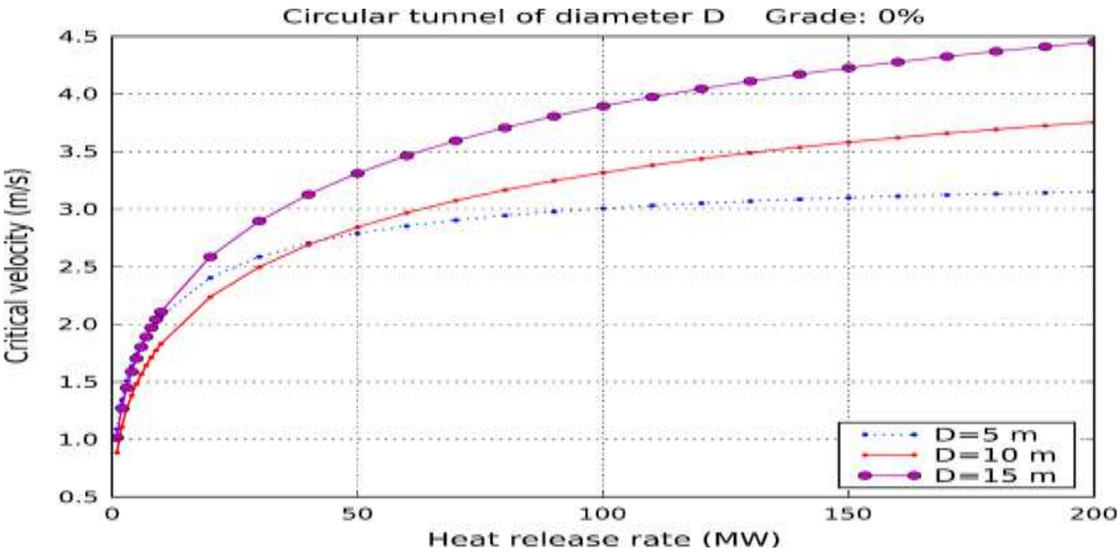

Figure.10. The relationship between critical velocity, heat release rate, and tunnel diameter

From Fig. (10), it can be seen that study of the effect of tunnel dimensions influences the efficiency of tunnel ventilation through the study of ventilation parameters. By changing air velocity, tunnel diameter, length of tunnel, speed of the fan (r.p.m), and other characteristics and studying this change in the shape. So, Simulation results reveal that the computational fluid dynamics (ANSYS FLUENT) software deals with the sections and lengths of the tunnel to sense the change in the amount of air using fans at different speeds (r.p.m). Thus, it gives flexibility in dealing with tunnels of different diameters and lengths. So that it is used in different ventilation conditions and different dimensions of a mine or a tunnel

6. CONCLUSIONS

The dynamic mesh of the CFD method is used in numerical simulation of airflow distribution in the tunnel of Cairo Metro Line 3. The numerical method is verified by the data of the National Authority for Tunnels, from which the ventilation rate, Q, and static head pressure, H, were obtained. Afterwards, pressure changes on the surface of the tunnel in the above scenarios are analyzed. From the present study, the followings can be concluded:

- The Characteristic equations obtained from the application of CFD have been used to describe the average velocity distribution, which provides the theoretical basis for accurately measuring the average velocity and ventilation flow rates in tunnels.
- The higher the number of iterations, the lower the error values in residuals. This demonstrates the accuracy of the solution resulting from the numerical analysis process, saving time and energy. also gives correct values and high accuracy in the solution.

- The speed is maintained until it reaches 80 seconds. As a result, the amount of ventilation exiting the fan at equal speeds along the tunnel is sufficient.
- The speeds change from 200 to 1200 r.p.m. At 9.5 million cells, the actual speed (600 r.p.m) is attained with acceptable accuracy. This led to improving fan speed efficiency and airflow distribution in the tunnel.
- It is noticed that the largest amount of air dispersion starts at the moment the train enters the tunnel and returns to normal after train departure. Where (-0.000 Pascal) is the time when the air meets the fan when the train passes by, and there is a significant change in pressure.
- The change in pressure as a result of the different layers of the fluid in relation to the length of the tunnel starts at -2.89e+03 pascal and ends at 2.76e+03 pascal. This improves the performance of the fan, leading to low energy consumption.
- Computational Fluid Dynamics (ANSYS FLUENT) software gives flexibility in dealing with tunnels of different diameters and lengths. So that it is used in different ventilation conditions and different dimensions of an underground mine or tunnel.

ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to the National Authority of Cairo Metro Tunnels for their assistance and cooperation to complete this work.

REFERENCES

1) Ding, C., X. He, and Nie, B. 2015. Numerical and experimental research on “key ring” distribution of ventilation in mine tunnels. Journal of Liaoning Technical University (Natural Science), 34(10): pp. 1131-1136.
2) Ding, C. 2016. Numerical and experimental research on the average velocity distribution in the trapezoid tunnels. Journal of Safety Science and Technology,12(1): pp. 28-32.
3) Richardson, L.F. 1922. Weather prediction by numerical process.:p.50.
4) Harlow, F.H. NUMERICAL METHODS FOR FLUID DYNAMICS, AN ANNOTATED BIBLIOGRAPHY. 1969, Los Alamos National Lab.(LANL), Los Alamos, NM (United States).pp.1-13.
5) Xu, G., Luxbacher, KD., Ragab, S., and Xu, J. 2017. Computational fluid dynamics applied to mining engineering: a review. International Journal of Mining, Reclamation and Environment, 31(4): pp. 251-275.
6) Tu, J., Yeoh, G., Liu, C., and Tao, Y. 2023. Computational fluid dynamics: a practical approach.: p.50.
7) Vega, M.G., Diaz, KMA., Ore, JMF., and Tajadura, RB. 2008. Numerical 3D simulation of a longitudinal ventilation system: memorial tunnel case. Tunnelling and Underground Space Technology, 23(5): pp. 539-551.
8) Passi, A., Nagendra, S.S., and Maiya, M. 2021. Characteristics of indoor air quality in underground metro stations: A critical review. Building and Environment.198: pp. 107-907.
9) Hofer, P., Eder, J., and Strau, C. 2021. Decision Support within Complex Subterranean Operations. in NATO Science and Technology Organization—Meeting Proceedings Paper (STO-MP-MSG-184).NATO STO: North Atlantic Treaty Organization.p.184.
10) Caliendo, C., Genovese, G., and Russo, I. 2020. Risk analysis of road tunnels: A computational fluid dynamic model for assessing the effects of natural ventilation. Applied Sciences, 11(1): p. 32.
11) Li, A. and Jones, P. 2000. Developments in strategies used for natural and mechanical ventilation in China. Indoor and Built Environment, 9(2): pp. 65-74.
12) Wang, F., Wang, M., and Wang, Q. 2012. Numerical study of effects of deflected angles of jet fans on the normal ventilation in a curved tunnel. Tunnelling and Underground Space Technology, 31: pp. 80-85.
13) Myrvang, T. and Khawaja, H. 2018. Validation of air ventilation in tunnels, using experiments and computational fluid dynamics. pp.1-16.
14) Betta, V., Cacetta, F., Musto, M., and Rotondo, G. 2010. Fluid dynamic performances of traditional and alternative jet fans in tunnel longitudinal ventilation systems. Tunnelling and Underground Space Technology, 25(4): pp. 415-422.
15) Yang, Z., Yu. Z., Yu, L., and Ma, F. 2014. Research on frequency conversion technology of metro station's ventilation and air-conditioning system. Applied thermal engineering, 69(1-2): pp. 123-129.
16) Zhao, D., Huang, F., Chen, S., Wang, d and Wang, D. 2014. Relationship between point air velocity and average air velocity in circular roadway. Journal of Liaoning Technical University (Natural Science), 33(12): pp. 165-165.
17) Kamal, H., Hussein, M., and Akl, A. 2021. Parametric study of twin tunnel under seismic loads for Cairo Metro Line No. 4. HBRC Journal, 17(1): pp. 137-156.
18) Jacques, E. and Wauters, P. 1999. Improving the ventilation efficiency of jet fans in longitudinally ventilated rectangular ducts.p.13.

19) Wang, F., Wang, M., He, S., Zhang, J., and Deng,Y. 2010. Computational study of effects of jet fans on the ventilation of a highway curved tunnel. Tunnelling and underground space technology, 25(4): pp. 382-390.
20) Adjiski, V., and Despodov, Z. 2019. Determining optimal distance from outlet of auxiliary forcing ventilation system to development of heading in underground mines. Journal of Mining and Environment,10(4): pp. 821-832.
21) Acuña, E.I. and Lowndes, I.S. 2014. A review of primary mine ventilation system optimization. Interfaces,44(2): pp. 163-175.
22) McPherson, M.J. 1993. Subsurface ventilation systems. Subsurface ventilation and environmental engineering: pp. 91-133.
23) Hamed, S., Abdel Rasoul, E., and Abuel-Kassem, M.2023. Design and calculation of tunnel ventilation system for Cairo Metro Line 3 (A case study). JES. Journal of Engineering Sciences, 51(1): pp. 1-14.
24) Lewus, D.C. Simulation of High Tunnel Ventilation Using Computational Fluid Dynamics. 2023, Rutgers The State University of New Jersey, School of Graduate Studies. p.13.
25) Jafari, S., Farhanieh, B., and Afshin, H. 2023. Numerical investigation of critical velocity in curved tunnels: Parametric study and establishment of new model. Tunnelling and Underground Space Technology, 135: pp. 10-21.
26) Deng, E., He, X., Ye., Y., and Zhu, Z. 2020. Transient aerodynamic performance of high-speed trains when passing through an infrastructure consisting of tunnel–bridge–tunnel under crosswind. Tunnelling and Underground Space Technology, 102: pp. 103-440.
27) Izadi, T., Aboualia, O., Mehrabian, MA., and Salmanzadeh, M. 2020. Investigation of the effects of different parameters on the generated pressure waves inside the tunnels. SN Applied Sciences, 2: pp. 1-13.
28) Chen, Z.,Liu, T., Zhou, X., and Niu, J. 2017. Impact of ambient wind on aerodynamic performance when two trains intersect inside a tunnel. Journal of Wind Engineering and Industrial Aerodynamics,169: pp. 139-155.
29) Rabani, M. and Faghih, A.K. 2015. Numerical analysis of airflow around a passenger train entering the tunnel. Tunnelling and Underground Space Technology,45: pp. 203-213.