27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-1985
NUMERICAL SIMULATION OF EXHAUSTION AND VENTILATION SYSTEMS IN AN UNDERGROUND SUBWAY STATION IN BRAZIL

Beatriz Cortez Rodriguez dos Santos
Amanda Sayuri Oizuni
Gustavo Saturnino
Cibele Silva
Fernando Kurokawa
Escola Politécnica da USP - Avenida Professor Luciano Gualberto, Travessa do Politécnico, 380, São Paulo – SP, Brasil.
beatriz.cortez@usp.br
amandasayuri@usp.br
gustavo2003@usp.br
cibele.civil@gmail.com
fernando.kurokawa@usp.br

Abstract. *The metro-railway system in Brazil transports millions of passengers daily, but there are currently no regulations governing the ventilation and exhaust systems used in these trains. Furthermore, they provide a viable solution to road overcrowding as they are underground constructions that do not disturb the chaotic dynamics that occur on the surface. Because unforeseeable accidents can occur during subway operations, it is difficult to predict the potential damage they may cause. Moreover, in the case of fire, smoke is the most dangerous factor for the lives of users. So, it is essential that the administration of this complex has an emergency management plan to deal with these eventualities. Studying the progression of smoke through fire simulations can provide guidance for a safe exit during the evolution of combustion within the subway station. It can also provide information on changes that can be made in the operating configurations and composition of the set: platform, tunnel, and wagon, in order to minimize damage in this scenario. Numerical simulation is a method that allows the analysis and understanding of the behavior of physical phenomena in the real world through mathematical modeling using computers. So, when creating a model, it is possible to predict a range of possible occurrences in a given process and clarify potential responses or decisions in the event of setbacks. Thus, an open source CFD software was used to perform two-dimensional simulations on a geometry of a subway station in order to test different scenarios of smoke propagation and air movement. Therefore, this study aims to address the greatest number of possible consequences of the movement of air and smoke in case of fire in the grouping of subway lines and stations. Additionally, it aims to analyze the ventilation and exhaustion systems and the operating modes implemented in the metro station and check whether they are efficient. So that in the future, the developed work may serve as a basis for a transformation in the panorama that involves the lack of standardization in the examination area.*

Keywords: *CFD (Computational Fluid Dynamics), Exhaustion System, Ventilation, Subway Station.*

1. INTRODUCTION

The city of São Paulo is renowned for its extensive population and vast territory, necessitating an efficient public transportation system to cater to its citizens. The metro plays a crucial role in this context since the inauguration of its first line in the 1970s (Ayub and Koury, 2017). Presently, São Paulo's metro network transports over 4 million passengers daily, becoming essential for urban mobility in the city (Metrô, 2021).

However, the city's haphazard growth has resulted in an imbalance between commercial areas, housing, and employment centers. This disparity requires citizens to travel long distances daily to carry out their basic activities. Consequently, urban roads are congested, and the metro has emerged as a viable solution to alleviate traffic congestion as its underground construction does not interfere with the chaotic dynamics on the surface (Fan et al., 2013).

To keep pace with the city's growth, expanding metro lines is a priority with significant investments from the government (Valor Econômico, 2022). However, regulation and standardization of metro transportation are inadequate in Brazil, particularly regarding ventilation and exhaust systems. The companies responsible for managing metro lines establish their internal standards to ensure passenger thermal comfort, but there is a lack of comprehensive regulation in this aspect (Pereira, 2019).

Ventilation and exhaust systems in metro stations are pivotal in air exchange, climate control, and pollutant removal. Air quality in these stations is of utmost importance, considering that passengers spend hours daily in underground environments. Furthermore, proper ventilation is essential to mitigate the spread of diseases, as demonstrated during the COVID-19 pandemic.

Computational simulation, employing Computational Fluid Dynamics (CFD) tools, has been widely used in international studies to assess ventilation and air quality in underground environments. This approach allows predicting fluid flow behavior and heat transfer, providing valuable information for the design and improvement of ventilation systems. In this regard, open source code softwares have been gaining popularity, mainly due to the freedom provided to the users and the ability to implement and create new functionalities based on the main code. One of the most widely known open source CFD tools is OpenFOAM, an Eulerian mesh-based software, which was selected for this study. OpenFOAM has been extensively used in numerous applications for solving complex fluid dynamics problems, such as studied by Biswas et al., 2022, for complex dispersions, and by Limane et al., 2015, for thermally induced airflow.

Additionally, safety in metro stations is a constant concern, particularly in emergency situations such as fires (Ji et al., 2017). Modeling and analysis are tools used to develop emergency management plans and evaluate the effectiveness of safety measures, such as platform screen doors deployment and material behavior in case of fire (Li et al., 2016; Roh et al., 2009; Park et al., 2006).

Studies have shown that proper distribution of ventilation openings and the combination of mechanical and natural ventilation systems can be more effective in smoke and toxic gas elimination during fires (Yan et al., 2011; Luo et al., 2014). Understanding these phenomena through computational simulations contributes to the continuous improvement of safety in metro stations.

The design of ventilation systems in complex tunnels requires further research and optimization to dilute pollutant concentrations and overcome obstacles in their geometry. Tunnel ventilation velocity needs enhancement, especially in curved regions where air jet development and contaminant propagation may be affected (Li et al., 2019). Studies have employed scale and CFD simulations to analyze ventilation in metro stations, with results indicating a proportional relationship between critical flow rate and the number of station entrances (Giachetti et al., 2017).

Scale modeling is an experimental option with numerical equivalents to reality, but it focuses on qualitative analyses. On the other hand, CFD tools allow for quantitative observations and simulations of fluid flows in complex environments. The combination of these methods has improved fire prevention and containment measures (Zhang et al., 2018).

In the context of São Paulo's metro lines, no research has been conducted addressing ventilation and exhaust. This highlights the lack of concern for safety and the absence of standardization in this area. Considering the airborne transmission of COVID-19, investigating the biosafety of these spaces is essential.

The objective of this study is to address ventilation and exhaust aspects in metro stations and tunnels of the metro lines, aiming to transform the current situation and emphasize the importance of this research field.

2. NUMERICAL SIMULATIONS

2.1 Numerical tool

OpenFOAM is an open-source software used to perform numerical simulations of Computational Fluid Dynamics (CFD) problems. The software is based on the discretization of the governing equations for fluid flow using the Finite Volume Method and employs the object-oriented programming language C++ (OpenFOAM, 2022).

Moreover, granting access to its source code to all users, OpenFOAM allows them to modify and implement their own codes. This enables users to gain a deeper understanding of the calculations performed in their simulations and offers them more freedom to create and develop new computational methods and routines (Masoomi et al., 2023).

In the simulations conducted in this study, no turbulence model was applied. By not using turbulence models, the simulation is capable of directly capturing the smaller scales of turbulence, resulting in a more accurate representation of fluid flow. Consequently, the uncertainties associated with approximations and simplifications in these models are eliminated.

2.2 Governing equations

The simplifying assumptions adopted in the simulations performed in this work were: newtonian fluid, incompressible laminar flow and transient regime.

The main equations governing the conditions formulated for the flow throughout this project are:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_i}{\partial x_j}\right) + f_i, \tag{2}$$

respectively, the mass conservation equation, Eq. (1), and the momentum conservation equation, Eq. (2), well-known in the context of fluid mechanics studies. In these equations, u represents the velocity component, t is time, ρ is mass density, p is pressure, μ is dynamic viscosity, and f_i represents possible external forces.

2.3 Numerical modeling

The numerical model was built based on the Pinheiros metro station of the Yellow Line 4, located in the city of São Paulo, Brazil. The choice of Pinheiros station as the study location was due to its six underground levels, which made it the most critical case to evaluate. A computational domain scheme is shown in Figure 1, with all dimensions in meters.

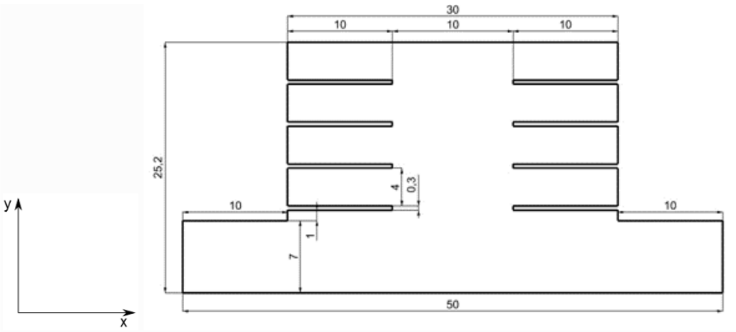

Figure 1. Dimensions of the geometry (m).

The model was developed using arbitrary measurements and data based on qualitative assessments of the site, as the necessary official documents could not be obtained. By using publicly available videos and photos of the location, as well as conducting a site visit, a similar two-dimensional geometry was reproduced, taking into account the station's entrances/exits and tunnel as sources of airflow, as no apparent mechanical ventilation was observed.

The geometry represents a cross-section of the station, where the first floor is combined with the second floor, as there is an opening connecting the two levels in the physical environment. The spacing of 0.3 m between the other levels represents the floor/ceiling that separates them. Therefore, the computational domain for this case is 50 x 25.2 = 1,260 m².

The boundary conditions serve to configure the simulated scenario according to what is desired to reproduce from reality. In this work, they were applied to replicate the flow conditions of the station and tunnel in the modeled geometries. The following boundary conditions were used:

- Inlet condition: fixed value- the x and y velocity components are set to specific values, creating a uniform velocity field;
- Outlet condition: zero gradient - the gradient of velocity normal to the established field is zero, and the normal velocity to the field is zero;
- Wall condition: no slip - the velocity is zero at the surface;
- Empty boundary: used for two-dimensional problems where velocity component z set to zero.

Furthermore, the simulations were developed using the icoFOAM solver, which is suitable for laminar and incompressible flow of Newtonian fluids. The geometry and mesh were created using the openFOAM generator called blockMesh. The variable discretization scheme employed was Linear Gaussian. Regarding the solution of linear systems, the PCG (Preconditioned Conjugate Gradients) was used for pressure, and for velocity, the smoothSolver with the Gauss-Seidel method was employed, both with relative tolerance. As for temporal integration, the solver's default option, which is the Euler scheme, was implemented.

2.4 Determination of grid size

The grid size is a key parameter to ensure reliability in a CFD simulation. The numerical stability of the simulation can be verified by the Courant number, Eq. (3), which relates the spatial discretization used to the time step:

$$C = \frac{U * \Delta t}{\Delta S}, \tag{3}$$

where t is the time step, U is the velocity in the x direction, and S is the spatial discretization size.

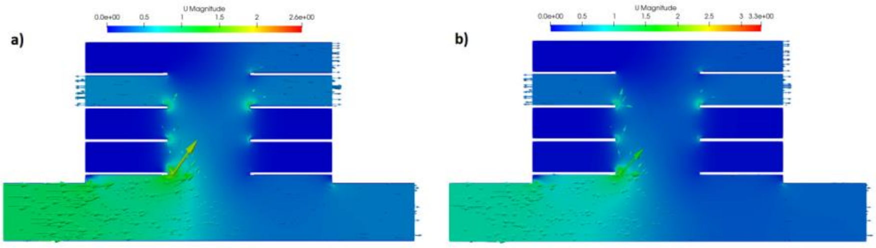

Figure 2. Velocity vectors: (a) grid system 0.1 m; (b) grid system 0.05 m.

It is recommended to keep the Courant number below 1 to obtain a reliable result. In this study, two different grid systems were developed, with the spatial discretization size 0.05m and 0.1m. The systems were compared based on the number and the results obtained in the simulations. Figure 2 presents the longitudinal distribution of velocity in the modeled space with different grid sizes. No significant difference in the simulations was observed when the grid size is greater than 0.1 m, but the processing time increases considerably. Therefore, the mesh used in all cases in this study has the following spatial discretization: x=y=0.1 m, as shown in Figure 3.

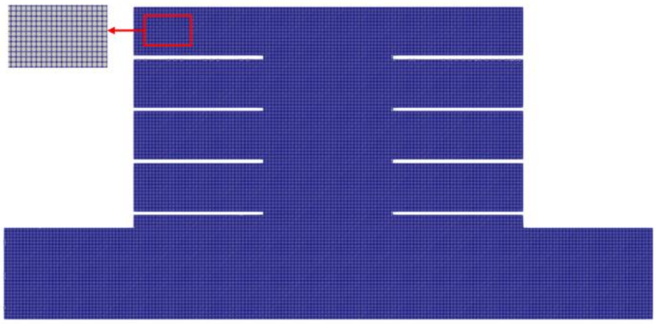

Figure 3. Mesh designated for simulation.

3. RESULTS AND DISCUSSION

3.1 Ventilation system efficiency

A simulation was conducted to assess the ventilation in the geometry. Air inlets were established and are indicated in Figure 4 by blue arrows, with the first corresponding to air arrival through the tunnel and the last two to air arrival through the passenger access opening. Additionally, the air outlets are indicated by a red arrow, with one referring to the connection with the existing pedestrian passage in the station and the other to the exit through the tunnel. The data used was grouped in Table 1.

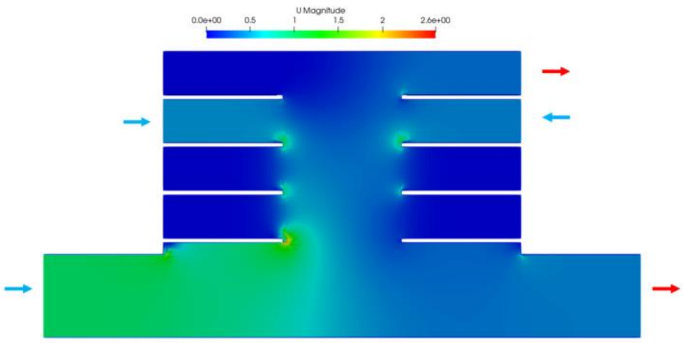

Figure 4. Flow velocity gradient for ventilation analysis at 1 s.

Table 1.Ventilation analysis simulation data.

Parameters	**Formula**	**Value**
Inlet length (L)	L = h	7m; 4m
Velocity (U)	U	1 m/s
Time step (ΔT)	ΔT	0.001s
Spatial discretization (ΔS)	ΔS	0.1m
Total time	T_F	40S
Kinematic viscosity coefficient	ϑ	0.000015m²/s

Examining the fluid movement in Figure 5 and Figure 6, it can be observed that after 20 and 40 seconds, respectively, of simulation, the air entering through the tunnel and existing accesses in the station is not sufficient to maintain ventilation in all areas of the space, let alone promote complete air exchange. Moreover, in the produced simulation, the fluid is constantly injected into the geometry at a constant velocity. However, in the station, air only enters when wind passes through the access or when the subway train arrives at the station. Thus, it is certain that the real scenario can be worse than what was found in the test. During the visit to the station, discomfort related to high temperature and lack of ventilation was noticed. Hence, the presented result seems consistent.

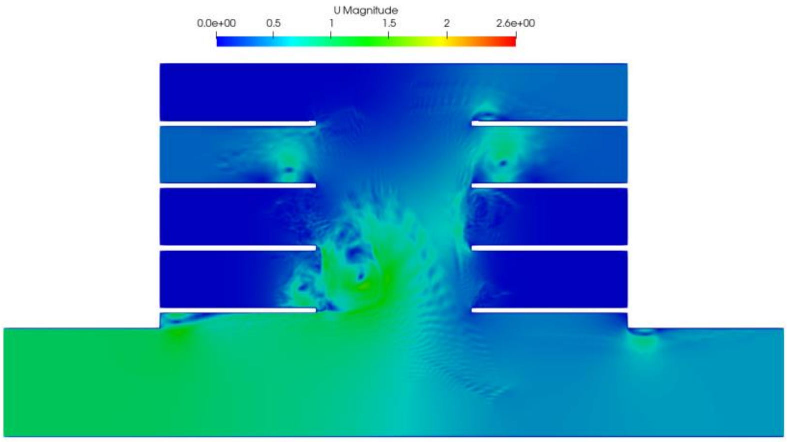

Figure 5. Flow velocity gradient for ventilation analysis at 20 s.

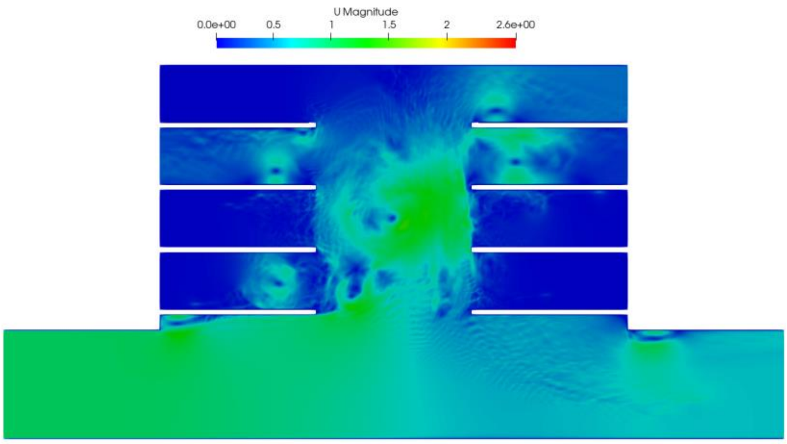

Figure 6. Flow velocity gradient for ventilation analysis at 40 s.

Figure 7 depicts the pressure profile obtained in the simulation conducted at different time intervals, namely a- 1s, b- 20s and c- 40s. A pressure drop zone is observed in the vortex formation regions, while the remainder of the geometry exhibits a nearly uniform pressure field. This consistency with the previously obtained velocity profiles indicates coherence.

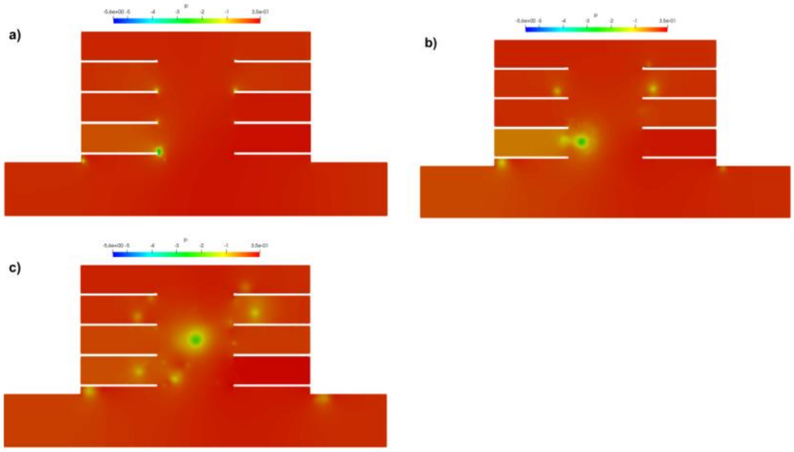

Figure 7. Pressure profiles for ventilation system analysis: a) at 1s, b) at 20s and c) at 40s.

3.2 Exhaust system efficiency

To inspect the exhaust aspects of the geometry, a fluid inlet was established, indicated in Figure 8 by a blue arrow, representing the arrival of smoke through the tunnel. Four air outlets were positioned, indicated by red arrows: two to represent the connection with the existing pedestrian passage in the station, one corresponding to the passenger access opening, and the other referring to the exit through the tunnel. The data used in the simulation was grouped in Table 2.

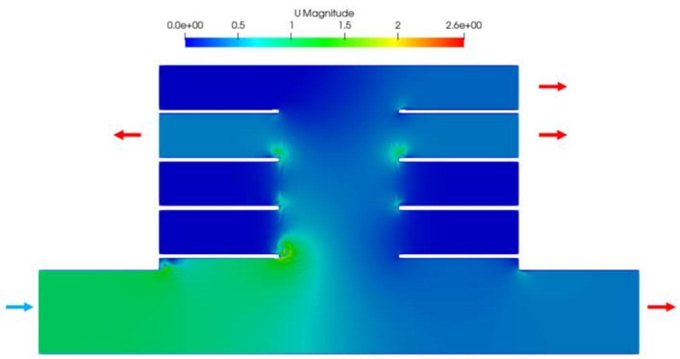

Figure 8. Flow velocity gradient for exhaustion analysis at 40 s.

Table 2. Exhaust analysis simulation data.

Parameters	**Formula**	**Value**
Inlet length (L)	L = h	7m
Velocity (U)	U	1 m/s
Time step (ΔT)	ΔT	0.001s
Spatial discretization (ΔS)	ΔS	0.1m
Total time	T_F	80S
Kinematic viscosity coefficient	ϑ	0.000021m^2/s

After 40 and 80 elapsed seconds, as seen in Figure 9 and Figure 10 respectively, the smoke has already filled the central space, where the escalator is located in the station, and continues to propagate through the subway tunnel. As the simulation progresses, it can be observed that the smoke is expelled through the existing openings in the station, but it does not prevent the smoke from spreading throughout all the floors of the geometry. Consequently, indicating the need for other methods and/or more fluid outlet points to mitigate the advancement of the fluid in case of an accident. Furthermore, since the station is connected to others, and after 80 seconds have elapsed, the smoke continues to propagate through the station tunnel, in the event of a fire, the adjacent stations will also be affected.

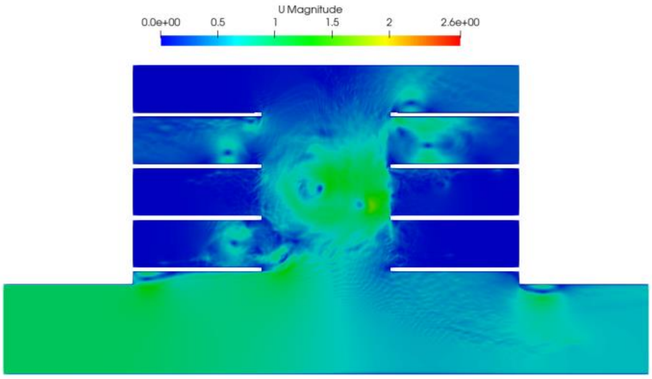

Figure 9. Flow velocity gradient for exhaustion analysis at 40 s.

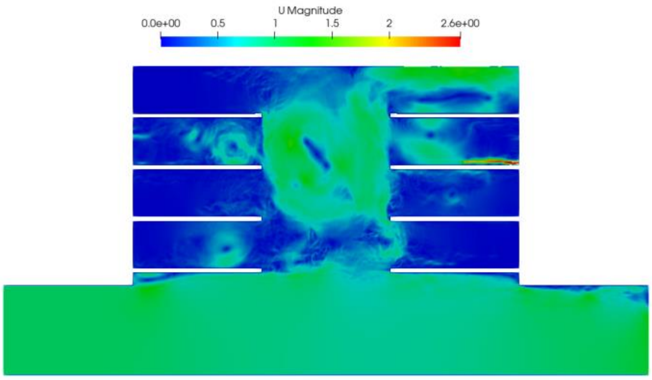

Figure 10. velocity gradient for exhaustion analysis at 80 s.

Figure 11 displays the pressure profile obtained in the simulation conducted at different time points, namely a- 2s, b- 40s, and c- 80s. Additionally, a pressure drop zone can be observed in the vortex formation regions, along with a nearly uniform pressure field in the remaining geometry. This consistency indicates agreement with the obtained velocity profiles.

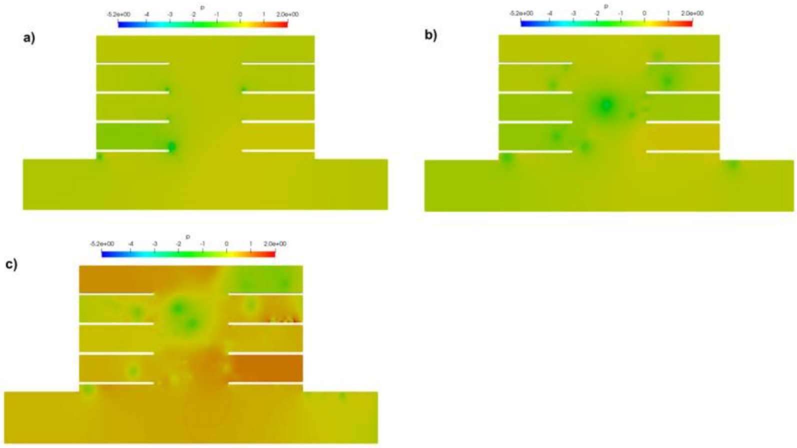

Figure 11. Pressure profiles for ventilation system analysis: a) at 2s, b) at 40s and c) at 80s.

4. CONCLUSION

In this study, the efficiency of ventilation and exhaust systems in the analyzed geometry was evaluated through simulations. The results showed that the air entering through the tunnel and existing access points in the station is insufficient to maintain adequate ventilation and promote complete air exchange within the space. Additionally, the constant injection of fluid at a fixed velocity does not accurately represent the real scenario, where air only enters the station when wind passes through the access or when the subway train arrives. During the visit to the station, discomfort related to high temperature and lack of ventilation was observed, corroborating the simulation results. Therefore, it is concluded that the real scenario is worse than simulated.

It was found that the smoke from the tunnel propagated through all floors of the station, even with partial expulsion through the existing openings. This indicates the need to adopt additional methods and fluid outlet points to effectively mitigate the advancement of smoke in case of accidents. The lack of adequate ventilation and the potential impact on adjacent stations are concerning aspects, emphasizing the importance of implementing corrective measures to ensure passenger safety.

The pressure profiles obtained in the simulations at different time intervals demonstrated consistency with the previously obtained velocity profiles. The presence of pressure drop zones in the vortex formation regions and a nearly uniform pressure field in the remaining geometry indicated coherence and reliability in the simulation results.

In conclusion, this study highlights the importance of assessing and optimizing the efficiency of ventilation and exhaust systems in metro stations to ensure passenger safety and comfort. These results underscore the need for further research and implementation of appropriate measures to address the ventilation challenges identified in this study and improve the overall performance of metro systems.

5. ACKNOWLEDGMENTS

The authors would like to thank CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico - Brazil) for grants 2022/136715-2 and 2022/160064-8, and FAPESP (Fundação de Amparo à Pesquisa do Estado de São Paulo - Brazil) for grant 2023/07771-4.

6. REFERENCES

Ayub, U., Koury, A., 2017. “Mobilidade urbana: uma questão em aberto no novo plano diretor da cidade de São Paulo”. Revista Projetar, v.2, n.1.

Biswas, R., Pal, A., Pal, R., Sarkar, S., 2022. “Risk Assessment of COVID infection by respiratory droplets from various ventilation scenarios inside an elevator: An OpenFOAM- based computational fluid dynamics analysis”. Physics of Fluids. 34, 013318.

Fan, C.G., Ji, J., Gao, Z.H., Han, J.Y., Sun, J.H., 2013. “Experimental study of air en-trainment mode with natural ventilation using shafts in road tunnel fires”. Int. J. Heat Mass Transf. 56, 750–757.

Giachetti, B; Couton, D; Plourde, F., 2017. “Smoke spreading analyses in a subway fire scale model”. Tunnelling and Underground Space Technology, 70: 233-239.

Ji, J., Guo, F.Y., Gao, Z.H., Zhu, J.P., Sun, J.H., 2017. “Numerical investigation on the effect of ambient pressure on smoke movement and temperature distribution in tunnel fires”. Appl. Therm. Eng. 118, 663–669.

Li, Q; Deng, Y; Liu, C; Zeng, Q; Lu, Y., 2016. "Modeling and analysis of subway fire emergency response: An empirical study". Safety Science, 84: 171–180.

Li, Y; Chen, T; Xu, Z., Kong, J., Wang, M., Fan, C., 2019. "Influence of winding wall on the entrainment characteristics of air jet in curved road tunnels". Tunnelling and Underground Space Technology, 90: 330-339.

Limane, A; Fellouah, H; Galanis, N., 2015. "Thermo-ventilation study by OpenFOAM of the airflow in a cavity with heated floor". Building Simulation. 8, 271-283.

Luo, N; Li, A; Gao, R; Song, T; Zhang, W; Hu, Z., 2014. "Performance of smoke elimination and confinement with modified hybrid ventilation for subway station". Tunnelling and Underground Space Technology, 43: 140–147.

Metrô. Quem somos, Institucional, Governo do Estado de São Paulo, http://www.metro.sp.gov.br/metro/institucional/quem-somos/index.aspx. Accessed 15 September 2021.

Masoomi, M; Rezanejad, K; Mosavi, A., "Numerical study of a novel ventilation system added to the structure of a catamaran for different slamming condition using OpenFOAM". Journal Pre-proof, S2092-6782(23)00001-8.

OpenFoam. User Guide, OpenFoam, https://www.openfoam.com/documentation/user-guide. Accessed 25 September 2021.

Park, W; Kim, D; Chang, H., 2006. "Numerical predictions of smoke movement in a subway station under ventilation". Tunnelling and Underground Space Technology, Elsevier: 304.

Pereira, N., 2019. "Sistema de Ventilação Primária e Extração de Fumaça da Linha 4 do Metrô do Rio de Janeiro". 25ª Semana de Tecnologia Metroferroviária.

Roh, J; Ryou, H; Park, W; Jang, Y., 2009. "CFD simulation and assessment of life safety in a subway train fire". Tunnelling and Underground Space Technology, 24: 447-453.

Valor Econômico, 2022. "Senado aprova empréstimo de R$2,9 bi para expansão do metrô em São Paulo", https://valor.globo.com/politica/noticia/2022/08/03/senado-aprova-emprestimo-de-r-29-bi-para-expansao-do-metro-em-sao-paulo.ghtml. Accessed 02 September 2022.

Yan, X; Wu, P; Wang, D., 2011. "Study of the Performance of Smoke Control under Non-uniform Smoke Exhaust Velocity". Tunnelling and Underground Space Technology, 11: 385-393.

Zhang, J; Li, Y; Dai, B; Li, X; Huang, Y., 2018. "The Effect of Exhaust Velocity on Smoke Exhaust in Subway Platform Fire". Procedia Engineering, 211: 1018–1025.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.