The 56th International October Conference on Mining and Metallurgy
22-25 October 2025, Bor Lake, Bor, Serbia
www.ioc.tfbor.bg.ac.rs

DOI: 10.5937/IOC25025O *Invited lecture* *Original scientific paper*

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF VENTILATION STRATEGIES FOR MITIGATING TOXIC GAS LEAKAGE IN SUBSURFACE INFRASTRUCTURE PROJECTS

Cemalettin Okay Aksoy [1*], **Güzin Gülsev Uyar Aksoy**[2], **Hasan Berker Sarisan**[3]

[1*]Dokuz Eylül University Department of Mining Engineering, İzmir, Türkiye
[2]Hacettepe University Department of Mining Engineering, Ankara, Türkiye
[3]GOA Mining, Ankara, Türkiye
[1*] okay.aksoy@deu.edu.tr, 0000-0002-4328-4862
[2] gulsev.aksoy@hacettepe.edu.tr, 0000-0002-9328-9899
[3] berkersarisan@goamining.com, 0009-0005-8082-8186

Abstract

The removal of toxic gases leaking during excavation in infrastructure projects is crucial for both human health and environmental protection. In this process, Computational Fluid Dynamics (CFD) simulations are highly valuable for modeling and analyzing complex physical processes associated with the transport and dispersion of gases. During infrastructure projects in geothermal regions, various gases may leak from underground sources. These gases may include toxic and/or flammable substances such as hydrogen sulfide (H_2S), carbon dioxide (CO_2), and methane (CH_4). Designing an effective ventilation system is essential for the removal of toxic gases through ventilation. CFD simulations can be utilized to evaluate the design and performance of ventilation systems. Modeling gas leaks from sources encountered during excavation using CFD simulations can help understand how gases move throughout infrastructure projects and their environmental impacts. CFD simulations can assist in optimizing factors such as the placement of ventilation systems, duct design, air flow rates, and filtration efficiency. This optimization ensures the effective removal of toxic gases and minimizes environmental impacts. CFD simulations can also be used to assess the efficiency and safety of ventilation systems designed for the mitigation of toxic gases. Additionally, simulating the operation of ventilation systems during potential emergency situations can help in evaluating associated risks.

Keywords: *Computational Fluid Dynamics, gas leakage, ventilation strategies, gas dispersion, subsurface infrastructure*

1. INTRODUCTION

The removal of toxic gases during excavation in infrastructure projects is critical for both human health and environmental safety. Computational Fluid Dynamics (CFD) simulations are valuable tools for modeling the transport and dispersion of hazardous gases such as H_2S, CO_2, and CH_4, which may leak from underground sources, especially in geothermal regions. Designing an effective ventilation system is essential, and CFD can be used to assess and optimize system performance, including fan placement, duct layout, airflow rates, and filtration. This helps ensure efficient gas removal and reduced environmental impact. CFD simulations also support safety evaluations by modeling system behavior in potential emergency scenarios. The study was conducted by modeling the T2 tunnel located on the İzmir–Ankara High-Speed Rail line. During the excavation works of the T2 tunnel, gas leakage consisting of 8000 ppm CO_2 and 40 ppm H_2S was detected along a 400-meter stretch starting from the tunnel face. The gas leakages were modeled in the analyses based on field measurements. Although the T2 tunnel is located within a geothermal basin, the leakages encountered during excavation are not of geothermal origin. Furthermore, it is assumed that the ventilation ducts used are airtight and that there are no other factors (such as machinery, personnel, or vehicles) influencing the airflow inside the tunnel. All surfaces of the tunnel are supported and covered with Ground Sheet lining. Based on field investigations, a surface roughness of 3 cm was applied to all tunnel surfaces in the simulations.

The temperature of the gases leaking from the ground is 28°C, while the nominal temperature of the tunnel is 26°C. Therefore, the thermal effect of the leaking gases is considered negligible. ANSYS Fluent software was used to perform the CFD analyses.

2. COMPUTATIONAL FLUID DYNAMICS

To conduct CFD analyses, the three-dimensional geometry of the structure must be created using a computer-aided design (CAD) program. In this project, the SpaceClaim module of the ANSYS software was used as the design tool. The tunnel geometry created has a length of 815 meters and a surface area of 130 m². Within the tunnel geometry, a fan duct with a diameter of 2200 millimeters and a length of 765 meters was modeled (ANSYS Education Tutorials, 2016).

Figure 1. 3D Tunnel geometry

2.1 Analysis No 1

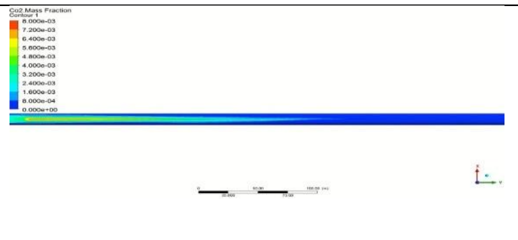
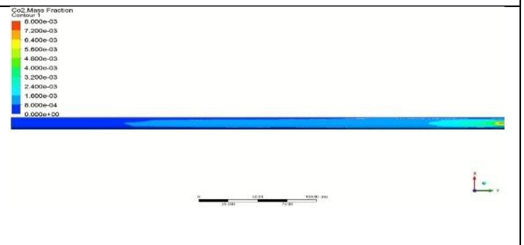
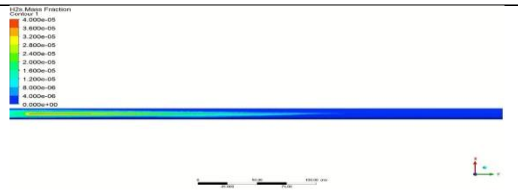
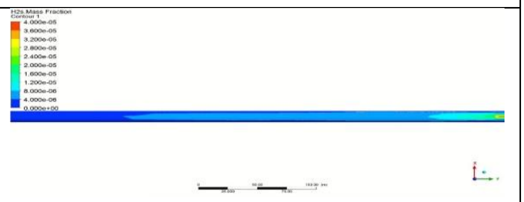
The airflow rate supplied to the tunnel by the fan is 39 m³/s. The surface area of the fan is 3.8013 m². The cross-sectional area of the tunnel is 130 m², and the average airflow velocity inside the tunnel toward the outlet (–Y direction) is 0.3 m/s.

The following analyses were carried out using a fan and duct system with a length of 765 meters and a diameter of 2200 mm, providing an airflow rate of 39 m³/s. This setup represents the existing ventilation system of the T2 Tunnel. To ensure an airflow rate of 39 m³/s, the fan's air discharge velocity was calculated (Goodfellow, 2020):

$$39\ m^3 = (3{,}8013\, m^2) * (V1) \tag{1}$$

As a result of these calculations, the fan's air discharge velocity V1 was determined to be 10.2596 m/s.

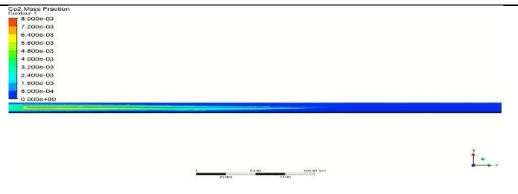
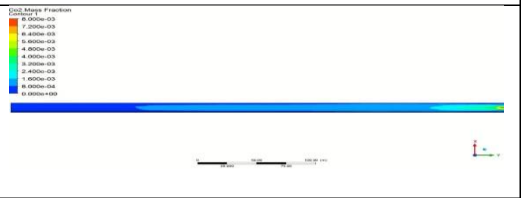
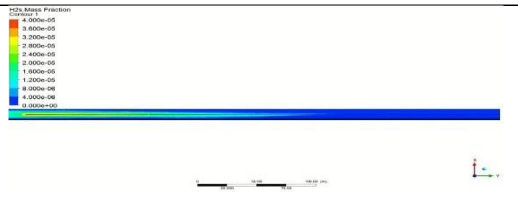
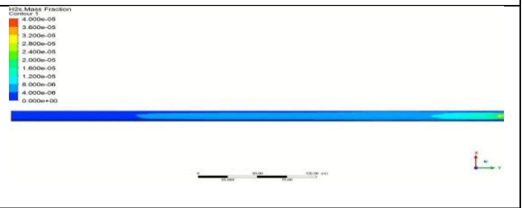
Table 1. CO_2 and H_2S Mass Contours on Tunnel Geometry (Analysis No 1)

	First 400 meters	Last 400 meters
CO_2 Mass Countours		
H_2S Mass Contours		

2.2 Analysis No 2

In addition to the ventilation configuration used in the first analysis, a fan was installed at the 100th meter of the tunnel, and a fan duct was defined from this fan to the tunnel outlet. This fan duct has a diameter of 2200 mm, like the initial fan duct. The second fan extracts air from within the tunnel at a flow rate of 39 m³/s.

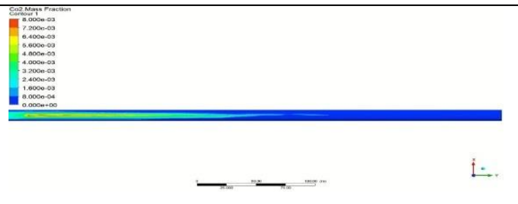
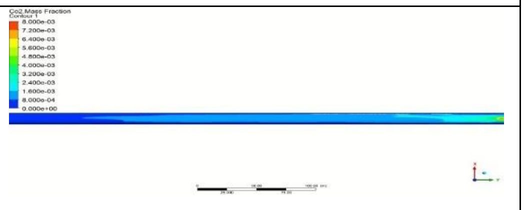
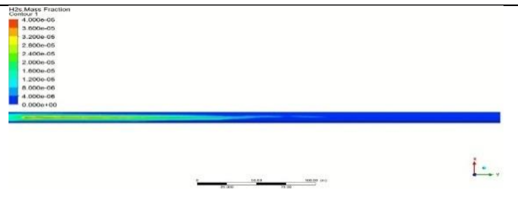
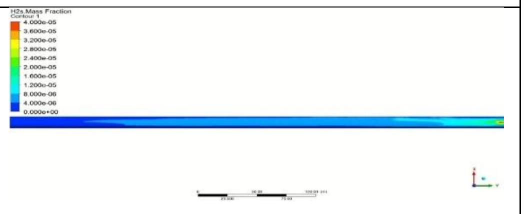
Table 2. CO_2 and H_2S Mass Contours on Tunnel Geometry (Analysis No 2)

	First 400 meters	Last 400 meters
CO_2 Mass Countours		
H_2S Mass Contours		

2.3 Analysis No 3

In this scenario, the length of the fan duct was increased from 765 meters to 790 meters. The second fan was not used. The diameter of the fan used is 2200 mm, and the airflow rate supplied from the fan duct into the tunnel is 39 m³/s.

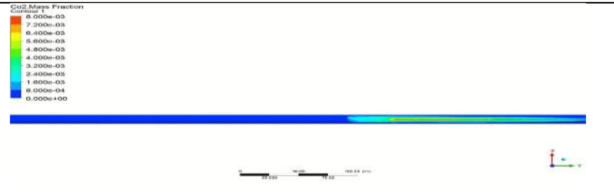
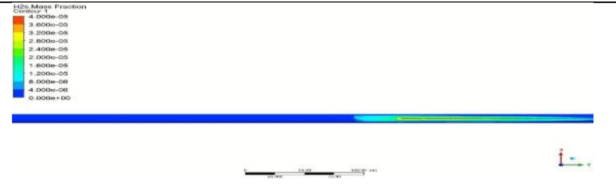
Table 3. CO_2 and H_2S Mass Contours on Tunnel Geometry (Analysis No 3)

	First 400 meters	Last 400 meters
CO_2 Mass Countours		
H_2S Mass Contours		

2.4 Analysis No 4

In the fourth analysis, a second 2200 mm diameter fan was added 375 meters from the tunnel outlet, with a duct of the same diameter extending to the exit. Both the first (790 m) and second (375 m) fan ducts operate with an airflow rate of 39 m³/s.

Table 4. CO_2 and H_2S Mass Contours on Tunnel Geometry (Analysis No 4)

CO_2 800 meters mass contours	H_2S 800 meters mass contours
	

3. RESULTS AND DISCUSSION

Flow analyses show that airflow near the tunnel face reaches approximately 3 m/s, reducing CO_2 and H_2S concentrations to 0 ppm in this region. Toward the tunnel outlet, airflow velocity decreases to nearly 0 m/s, causing gas accumulation around the 400th meter—up to 8000 ppm CO_2 and 40 ppm H_2S in the first analysis. In the second analysis, a fan added 100 meters from the outlet had no effect, as it was positioned in a gas-free zone. In contrast, the fourth analysis—where the second fan was placed closer to the source (375 m from the outlet) and connected via a duct—significantly improved gas evacuation, reducing CO_2 and H_2S concentrations to 0 ppm downstream. Extending the first fan duct from 765 m to 790 m, however, did not yield notable improvement. These results highlight that the position of the second fan is the most influential parameter for effective ventilation. CFD simulations proved valuable in optimizing fan placement, duct configuration, and airflow rates, helping minimize toxic gas accumulation. Moreover, CFD serves as a powerful tool for evaluating ventilation strategies under varying structural conditions, supporting safer and more efficient infrastructure development.

REFERENCES

[1] J.F. Brune, Mine Ventilation Systems. In: Hartman, H. L., Mutmansky, J. M., Ramani, R. V., Wang, Y. J. (Eds.), Mine Ventilation and Air Conditioning, Wiley-Interscience, 1996.

[2] J. Bear, Dynamics of Fluids in Porous Media, Dover Publications, 1972.

[3] M.J. O'Sullivan, K. Pruess, M.J. Lippmann, Geothermics, 30 (4) (2001) 395–429.

[4] ANSYS Training Courses, SpaceClaim Introduction, Meshing Introduction, Fluent Introduction, 2016.

[5] Y.A. Çengel, J.M. Cimbalak, Fluid Mechanics: Fundamentals and Applications, 2004.

[6] H.D. Goodfellow, R. Kosonen, Industrial Ventilation Design Guide: Volume 1: Fundamentals, 2020.

[7] C. Liu, G.H. Yeoh, J. Tu, , Computational Fluid Dynamics: A Practical Approach, 2007.

[8] J.H. Ferziger, M. Perić, Computational Methods for Fluid Dynamics, 2002.