DOI: https://doi.org/10.5592/CO/EUROENGEO.2024.260

THE USE OF COMPUTATIONAL FLUID DYNAMICS (CFD) IN THE VENTILATION-BASED REMOVAL OF TOXIC GASES LEAKING FROM GEOTHERMAL SOURCES IN INFRASTRUCTURE PROJECTS

C. OKAY AKSOY[1], G. GÜLSEV UYAR AKSOY [2], H. BERKER SARISAN [3]

[1]*Dokuz Eylül University Mining Engineering Department, Turkey, okay.aksoy@deu.edu.tr*[1]

[2]*Hacettepe University Mining Engineering Department, Turkey, gulsevaksoy@hacettepe.edu.tr*

[2] *GOA Mining, Turkey, berkersarisan@gmail.com*

Abstract

The ventilation-based removal of toxic gases leaking from geothermal sources 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 related to the transport and distribution of gases. During infrastructure projects in geothermal areas, various gases can leak from underground geothermal sources. These gases may include toxic and/or combustible gases such as hydrogen sulfide (H_2S), carbon dioxide (CO_2), and methane (CH_4). Effective design of a ventilation system is crucial for removing toxic gases through ventilation. Computational Fluid Dynamics (CFD) simulations can be used to evaluate the design and performance of ventilation systems. Modeling the gases leaked from geothermal sources 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 effective removal of toxic gases and minimizes environmental impacts. CFD simulations can be used to assess the effectiveness and safety of ventilation systems for removing toxic gases. Additionally, simulating the operation of ventilation systems during potential emergencies can help evaluate risks.

Key words

Calculational Fluid Dynamics, tunnel, fan, geothermal source, infrastructure

1 Introduction

Infrastructure projects may encounter geothermal gas leaks, which can have serious effects on the environment. These gases are often toxic and/or combustible. The most common ones include hydrogen sulfide (H_2S), carbon dioxide (CO_2), and methane (CH_4). The release of these gases into the environment can lead to various harmful consequences.

Firstly, these gases can directly harm human health. Hydrogen sulfide can cause serious effects on the respiratory system with short-term exposure and can lead to severe health problems with long-term exposure. Carbon dioxide in high concentrations can cause respiratory problems and even death. Methane is a flammable gas and can increase the risk of explosions.

Additionally, geothermal gas leaks can also negatively impact natural habitats. Damage to vegetation and soil by these gases can disrupt ecosystems' balance. High concentrations of carbon dioxide can negatively affect plant growth and lead to soil acidification. Methane leaks can contribute to climate change by creating a greenhouse effect in the atmosphere.

In cases where geothermal gas leaks are not cleaned through ventilation, serious risks can arise. Especially in enclosed spaces or tunnels, gas concentrations can reach dangerous levels. This situation can endanger the health of workers and increase the risk of explosions or poisoning in emergencies.

Therefore, it is crucial to effectively clean geothermal gas leaks through ventilation. Ventilation systems help to extract these gases and release them outside, thus protecting the environment and the health of workers. Additionally, proper design and regular maintenance of these systems are of critical importance.

In conclusion, the environmental effects of geothermal gas leaks and the risks associated with not cleaning them through ventilation should be taken seriously. Without effective measures, potential dangers to both the environment and human health may arise. Therefore, managing gas leaks and ensuring the effectiveness of ventilation systems are crucial in geothermal infrastructure projects.

2 Methods

2.1 The Importance Of Using The CFD Method

In the analysis of ventilation systems for managing geothermal gas leaks, Computational Fluid Dynamics (CFD) plays a pivotal role. CFD is a powerful computational tool used to simulate and analyze the behavior of fluids and gases in complex systems. It employs numerical methods and algorithms to solve the governing equations of fluid flow, heat transfer, and other related phenomena.

CFD allows engineers and researchers to model and visualize the distribution and movement of gases within infrastructure projects, such as tunnels or enclosed spaces. By dividing the domain into discrete computational cells, CFD simulates the interactions between fluid particles and boundary surfaces, providing insights into flow patterns, turbulence, temperature distributions, and gas concentrations.

In the context of ventilation analysis for geothermal gas management, CFD enables detailed simulations of airflow dynamics, helping to optimize the design and performance of ventilation systems. By inputting parameters such as fan characteristics, duct geometries, environmental conditions, and gas properties, CFD simulations can predict how gases disperse and are removed through ventilation.

One of the key advantages of CFD is its ability to assess multiple design scenarios rapidly and cost-effectively, allowing engineers to iterate and refine ventilation system designs before implementation. This iterative process helps identify potential issues and optimize system parameters to achieve efficient gas removal while minimizing energy consumption and environmental impacts.

Moreover, CFD can simulate various operating conditions and emergency scenarios, providing valuable insights into the effectiveness and safety of ventilation systems under different circumstances. This predictive capability is crucial for ensuring the reliability and resilience of ventilation systems in mitigating the risks associated with geothermal gas leaks.

CFD simulations provide powerful tools for understanding and predicting complex fluid dynamics, but they also come with certain limitations and uncertainties. Factors such as model simplifications, grid resolution, turbulence models, and boundary conditions can affect the accuracy of simulation results. Therefore, collecting experimental data and validating simulations with this data is critical for ensuring the reliability of the models. Accurate data collection and continuous model calibration enhance the accuracy of CFD simulations and ensure the reliability of the results obtained. In this type of study, the detection of gases seeping from the ground and the rate at which they are seeping are of great importance.

Additionally, identifying leaks or seepages in the existing ventilation structure, as well as permeable areas in the tunnel structure that could affect ventilation, will enhance the accuracy of the simulation

In summary, Computational Fluid Dynamics (CFD) plays a crucial role in the analysis and optimization of ventilation systems for managing geothermal gas leaks in infrastructure projects. By providing detailed insights into airflow dynamics and gas dispersion, CFD enables engineers to design effective and reliable ventilation systems that protect both human health and the environment.

In this study, an example tunnel geometry and fan duct structure were analyzed using the Computational Fluid Dynamics (CFD) method to investigate the optimization of a ventilation system in the event of a potential geothermal gas leak.

2.2 Geometry

To perform CFD analyses, the three-dimensional geometry of the structure must first be created in a computer-aided design program. In this project, the ANSYS program's SpaceClaim module was utilized as the design tool. The created tunnel geometry has a length of 815 meters and a surface area of 130 m^2. Within the tunnel geometry, a fan duct geometry with a diameter of 2200 millimeters and a length of 765 meters was constructed (ANSYS Education Tutorials, 2016).

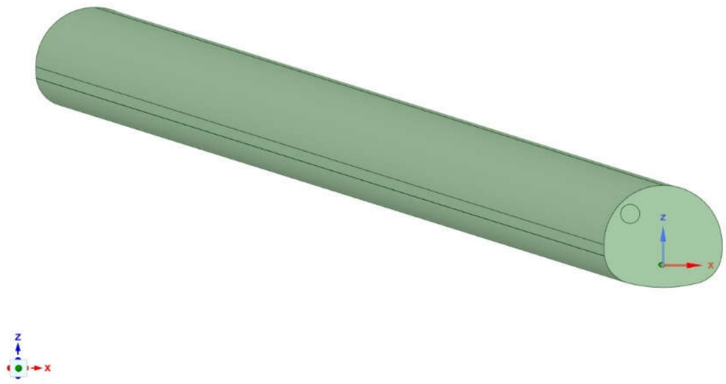

Figure 1. Sample tunnel geometry

2.3 Meshing

Meshing is the process of dividing a physical domain into smaller intervals. The purpose of meshing is to facilitate the solution of a differential equation. In this study, the Cutcell Mesh method was used as the meshing technique. Cutcell Mesh is an efficient mesh model due to its high orthogonal quality and low number of resulting elements. On the geometry, a high-quality mesh with 2,426,253 elements and an orthogonal quality value of 0.07 was created.

Boundary names were defined on the mesh, including inlet, internal_fantup1, outlet, wall_fantup1, and fluid_source. The inlet boundary name represents the entrance of the fan duct, internal_fantup1 represents the surface where the fan duct opens into the tunnel, outlet represents the tunnel exit, wall_fantup1 represents the walls of the fan duct, and fluid_source represents the areas on the tunnel floor where gas is released (ANSYS Education Tutorials, 2016).

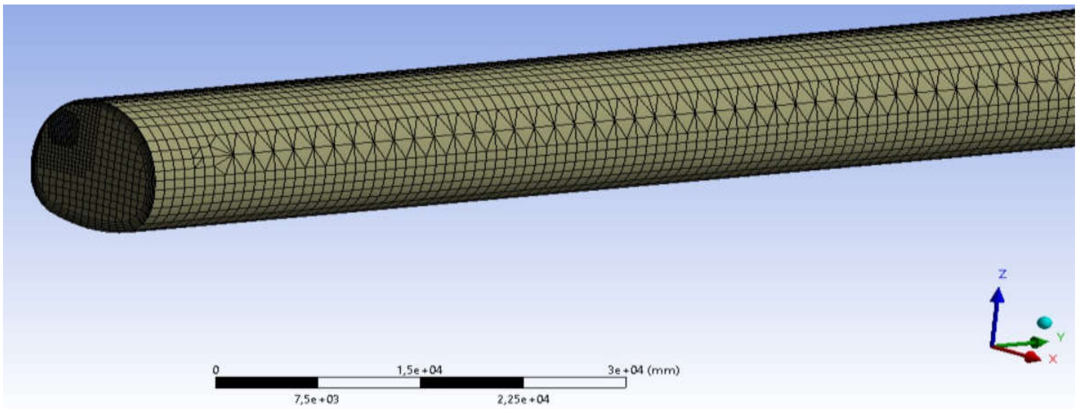

Figure 2. Mesh structure created on tunnel geometry

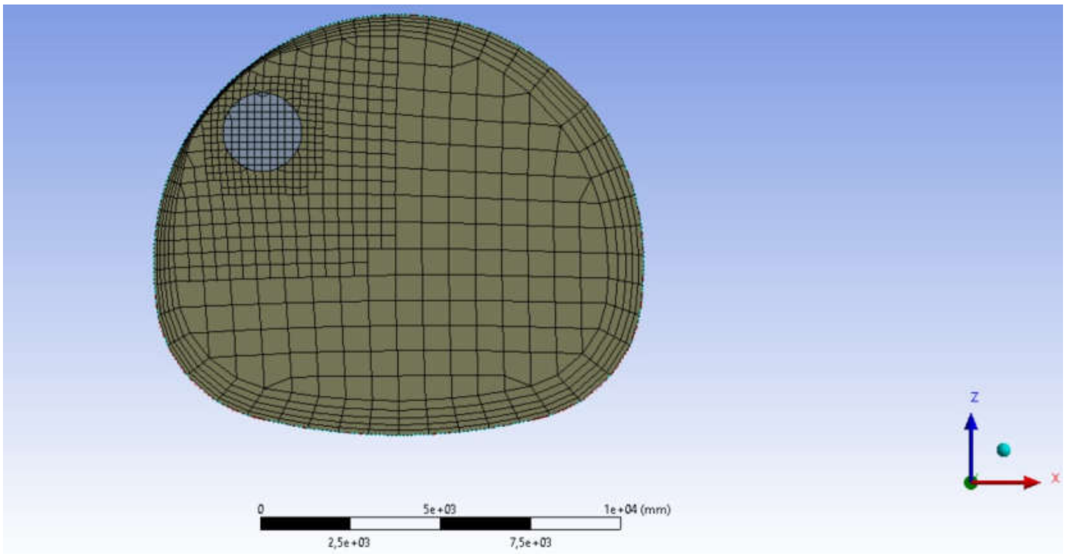

Figure 3. Cross-sectional view of the mesh structure

2.4 CFD Analyses

Computational Fluid Dynamics (CFD) is an engineering tool used to analyze the movement and interactions of fluids (typically liquids and gases) using computer-based numerical methods. CFD analyses utilize computer programs and mathematical methods to model and simulate fluid flow, temperature distributions, pressure differences, and other physical properties.

CFD analysis involves mathematically modeling the geometry of a system or device and conducting equation-based simulations of fluid behavior consistent with this model. Fundamental equations such as the Navier-Stokes equations, momentum equations, and energy equations form the basis of the analysis. These equations are solved using numerical methods such as finite element method, finite volume method, finite difference method, etc., to model fluid behavior (ANSYS Education Tutorials, 2016).

In this study, 8000ppm CO_2 and 40ppm H_2S gases are assumed to leak from a geothermal source for 400 meters from the tunnel mirror. It is also assumed that the fan tubes used are sealed and that there are no other factors affecting the air flow inside the tunnel.

2.4.1 Analysis No 1

The air flow rate supplied from the fan to the tunnel is 39 m^3/s. The surface area of the fan is 3.8013 m^2. The cross-sectional area of the tunnel is 130 m^2, and the average air flow velocity inside the tunnel is 0.3 m/s, directed towards the tunnel exit (-Y direction).

The following analyses were conducted with a fan and fan duct with a length of 765 meters and a diameter of 2200 mm, achieving an airflow rate of 39 m^3/s. The fan's air blowing velocity to achieve the 39 m^3/s flow rate was calculated (Goodfellow, 2020).

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

$$V1 = 10{,}2596\ m/s \quad (2)$$

The airflow velocity of the fan has been determined to be 10.2596 m/s.

In the analyses conducted, starting from the mirror and extending to a distance of 400 meters, a release of 8000 ppm CO_2 and 40 ppm H_2S has been made from the fluid_source boundary.

The following results have been obtained:

When examining the mass contours of CO_2 and H_2S gases for the last 400 meters, it is observed that approximately 100 meters from the tunnel exit, the concentrations of CO_2 range between 800 ppm and 1600 ppm, and for H_2S, they range between 4 ppm and 8 ppm. It is evident that the gas density, which is higher in the middle of the tunnel, decreases towards the tunnel exit. Although not very efficient, it can be seen from the shapes formed by the mass contours that the gases are being swept towards the tunnel exit (Figure 4, Figure 5).

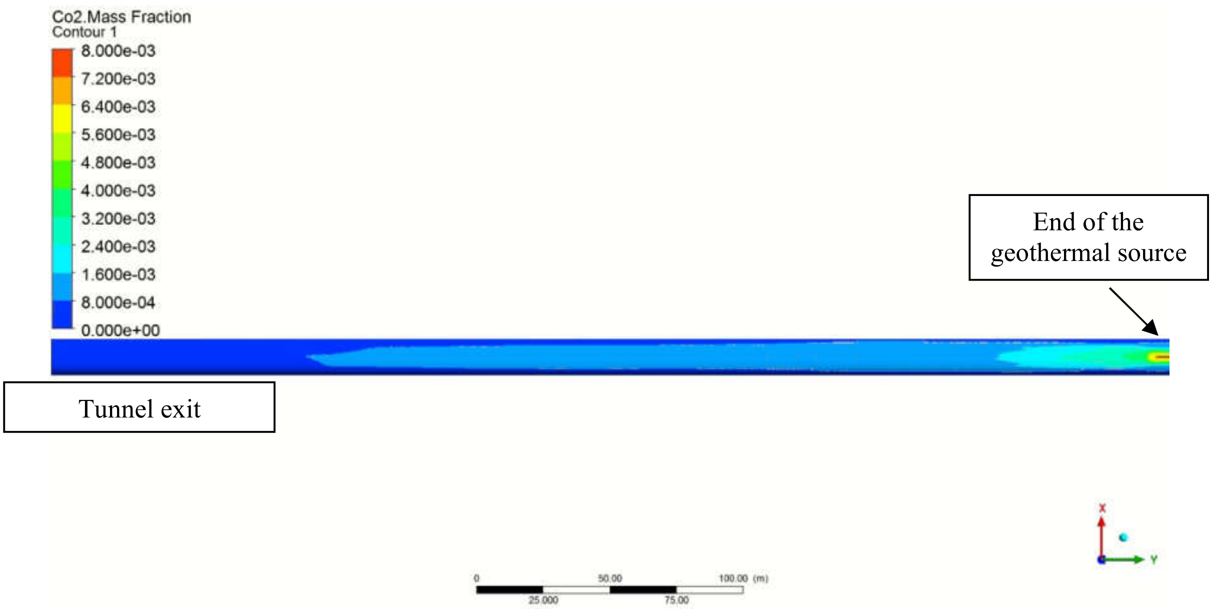

Figure 4. CO_2 mass contour view of the section from the end of the geothermal source to the exit of the tunnel

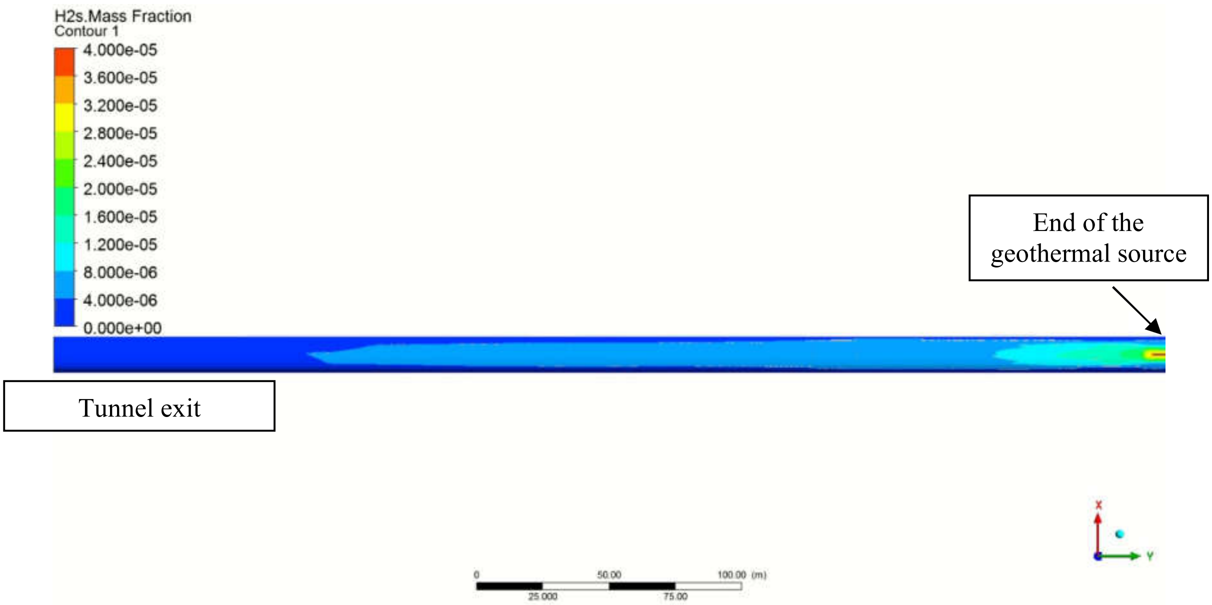

Figure 5. H_2S mass contour view of the section from the end of the geothermal source to the exit of the tunnel

2.4.2 Analysis No 2

In addition to the first analysis, a fan has been placed at the 100th meter of the tunnel, and a fan duct has been defined from its end to the tunnel exit, similar to the first fan duct. The diameter of the fan duct is 2200 mm. Like the first fan, the second fan also intakes the airflow inside the tunnel with a flow rate of 39 m^3/s.

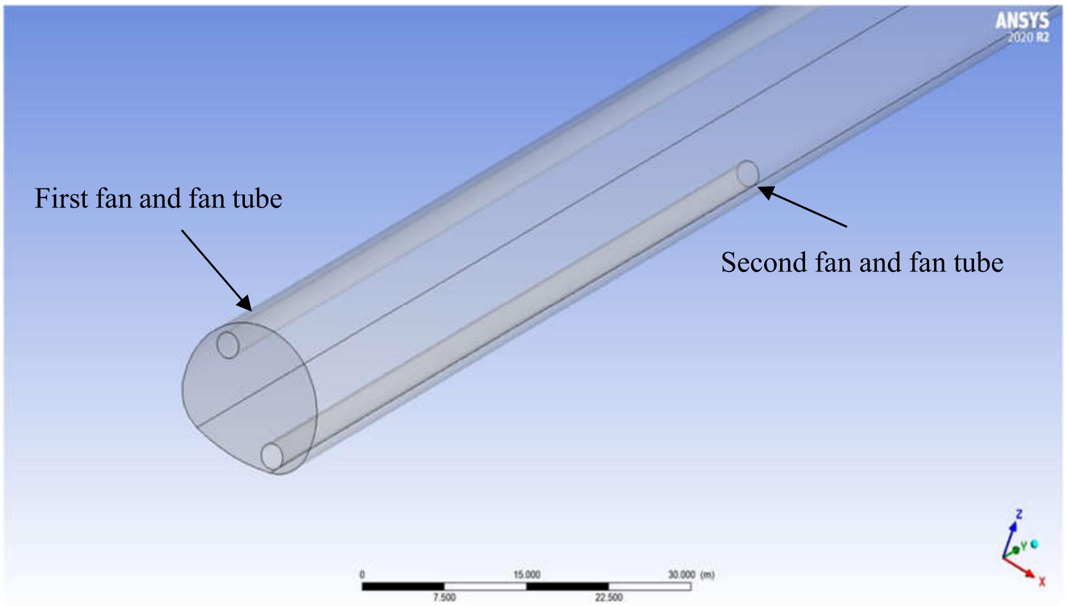

Figure 6. Second fan and fan tube defined in the tunnel geometry for ventilation optimization

When examining the mass contours for CO_2 and H_2S gases for the last 400 meters, similar results to the first analysis have been obtained. However, the placement of the second fan at a distance of 100 meters from the exit, where the emitted gases are at 0 ppm and the airflow is weak, has not been effective in removing the gases (Figure 7, Figure 8).

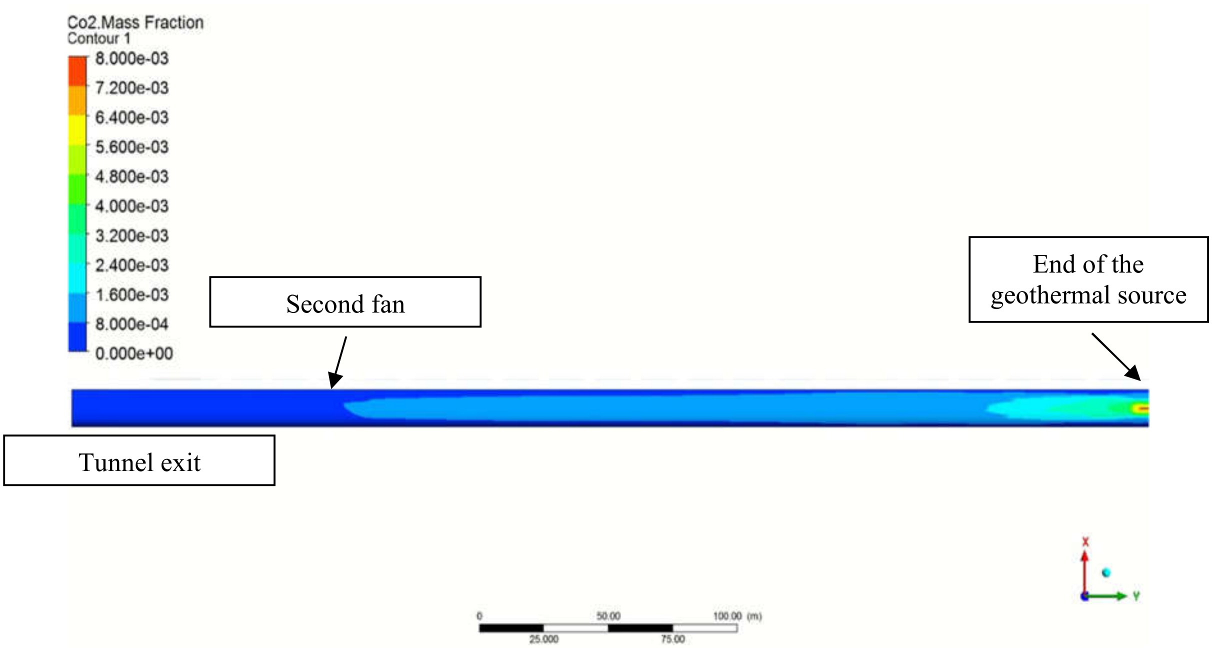

Figure 7. CO_2 mass contour view of the section from the end of the geothermal source to the exit of the tunnel

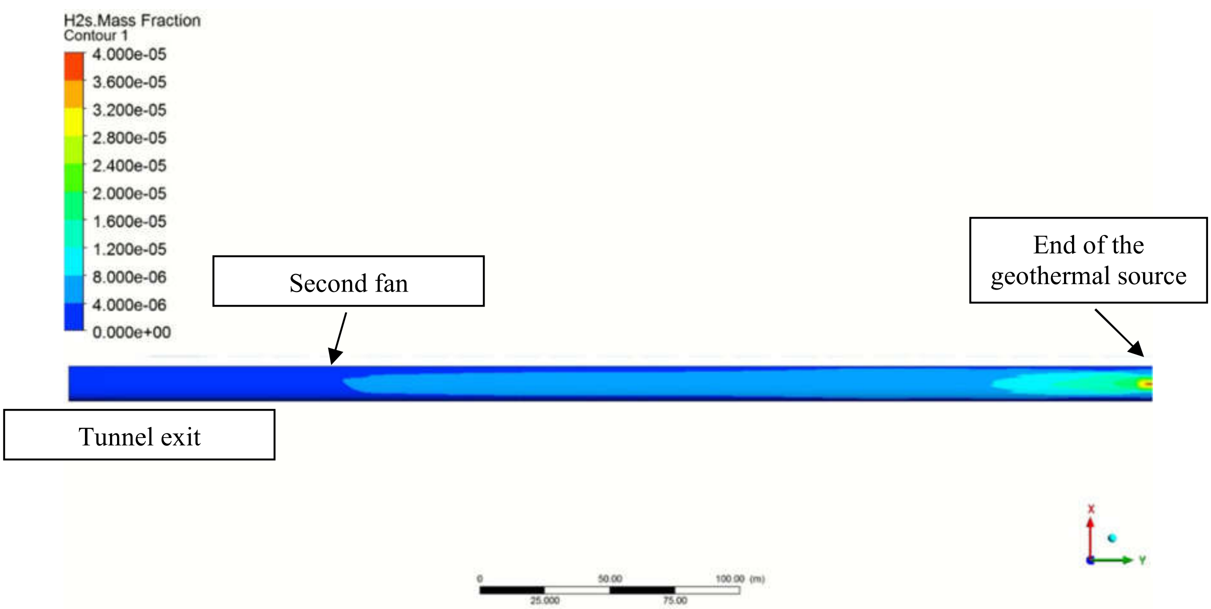

Figure 8. H_2S mass contour view of the section from the end of the geothermal source to the exit of the tunnel

2.4.3 Analysis No 3

In this scenario, the length of the fan duct, which was initially 765 meters, has been increased to 790 meters. The second fan has not been used.

The structure observed in the mass contours for the last 400 meters is similar to the results obtained in the first analysis. While 0 ppm values are observed for both gases in the last 100 meters, accumulations of 2400 ppm for CO_2 and 12 ppm for H_2S have formed from the end of the source (Figure 9, Figure 10).

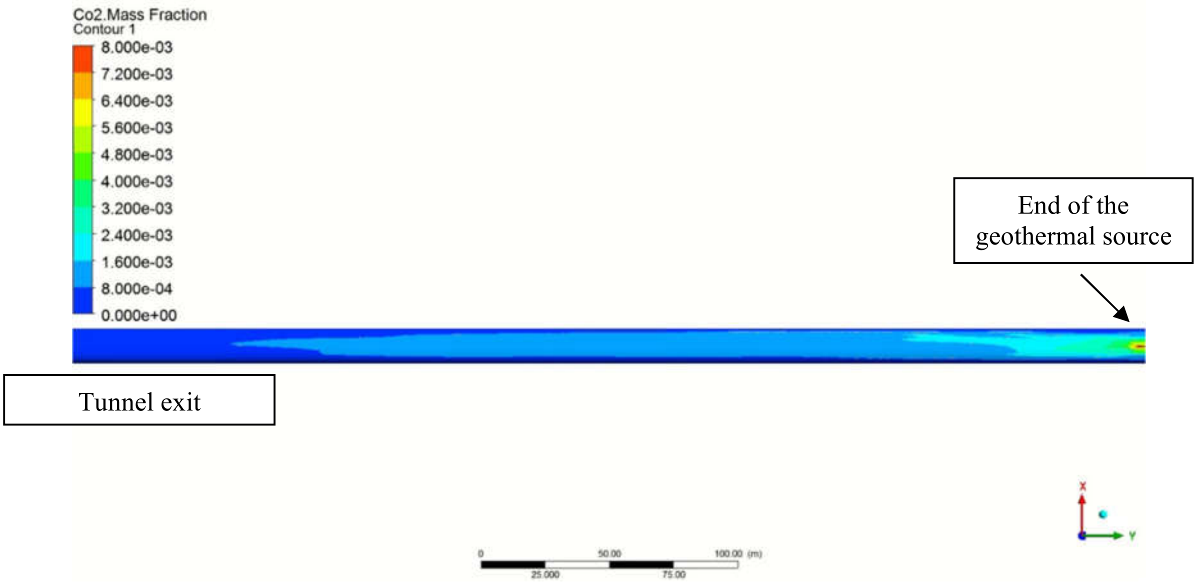

Figure 9. CO_2 mass contour view of the section from the end of the geothermal source to the exit of the tunnel

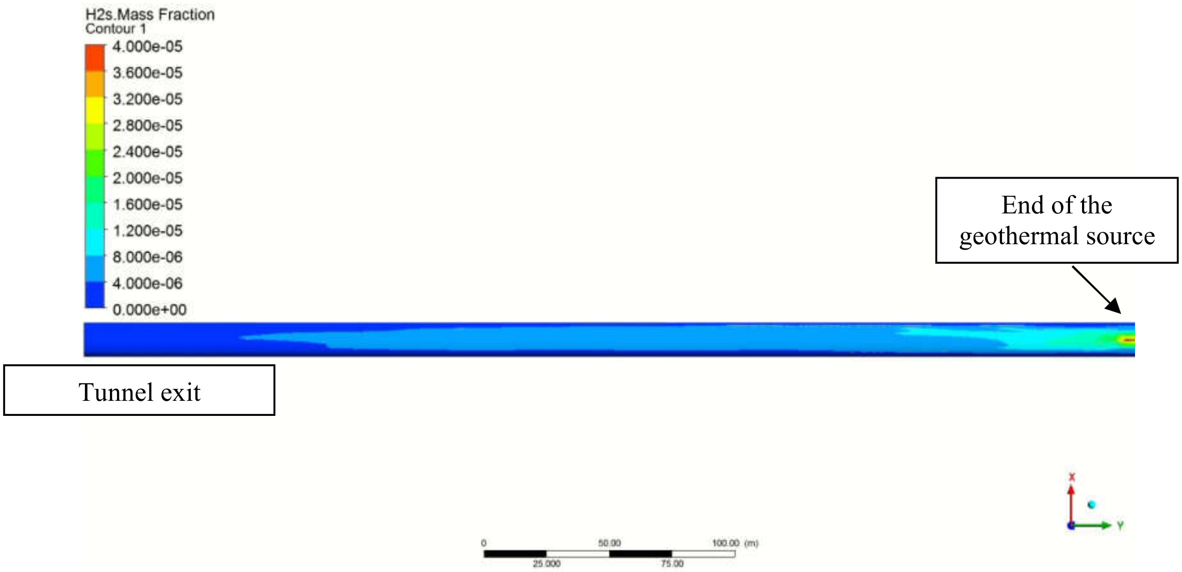

Figure 10. H_2S mass contour view of the section from the end of the geothermal source to the exit of the tunnel

2.4.4 Analysis No 4

In the 4th analysis, there is a second fan. The diameter of this fan is 2200 mm. The fan is defined at a distance of 375 meters from the tunnel exit. A fan duct with a diameter of 2200 mm is defined for this fan up to the tunnel exit. Similar to the first fan, an airflow intake rate of 39 m^3/s is defined.

When examining the mass contours of the gases, the effect of the second fan is clearly visible. When the emitted gases reach the influence area of the second fan, they are vacuumed and discharged. As a result, CO_2 and H_2S gases are at 0 ppm values after the second fan towards the tunnel exit. However, accumulations of 1600 ppm to 6400 ppm for CO_2 and 8 ppm to 32 ppm for H_2S are observed above the source, as in previous analyses (Figure 11, Figure 12).

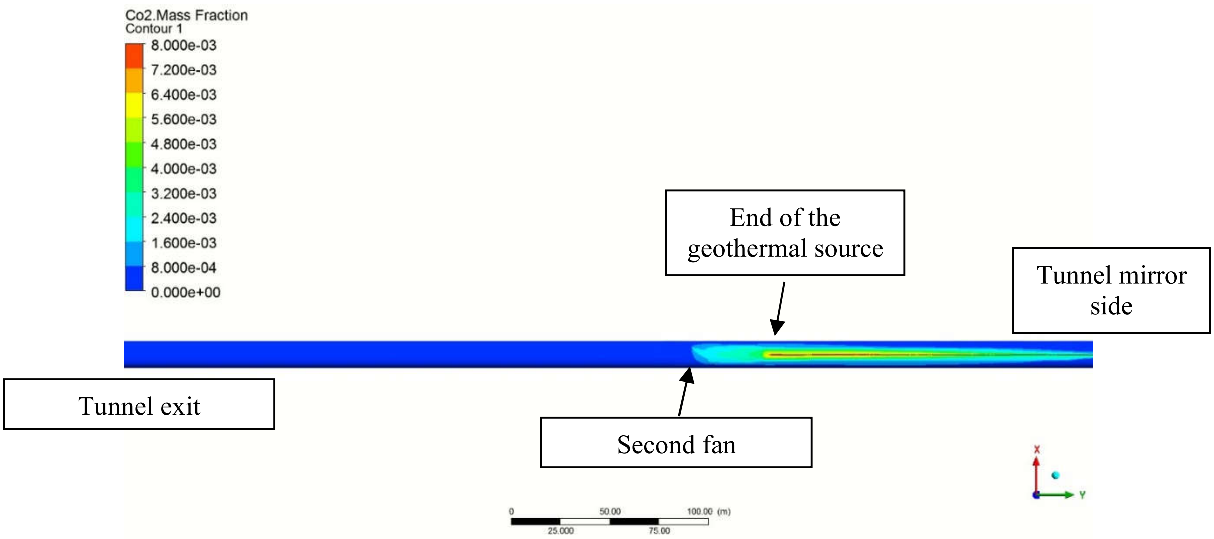

Figure 11. CO_2 mass contour view of the section from the mirror side of the tunnel to the exit of the tunnel

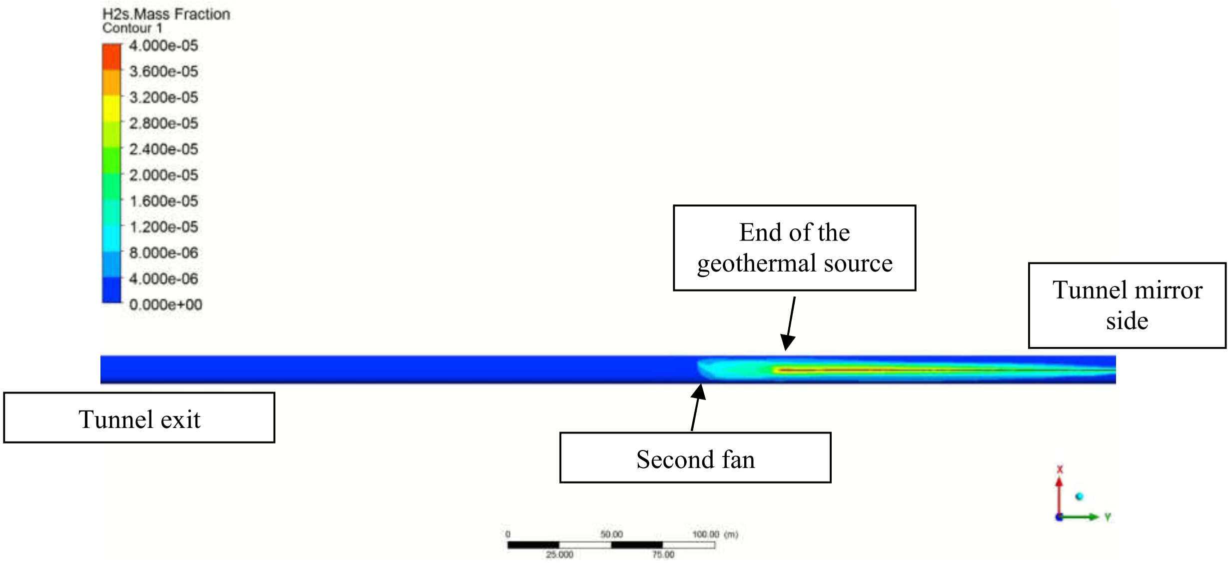

Figure 12. H_2S mass contour view of the section from the mirror side of the tunnel to the exit of the tunnel

3 Results

It is observed that CO_2 and H_2S gases near the mirror are at 0 ppm values. This is because the airflow is strong in the areas close to the mirror, effectively sweeping the gases.

In the first analysis, accumulations of CO_2 and H_2S gases of up to approximately 8000 ppm and 40 ppm, respectively, were observed in the tunnel at approximately 400 meters from the mirror.

The second fan added in the 2nd analysis, positioned near the tunnel exit where the gases are at 0 ppm values (100 meters from the tunnel exit), did not contribute positively to the evacuation of gases.

By moving the position of the second fan closer to the source (375 meters from the tunnel exit) and extending the fan duct it is connected to, the performance of gas sweeping has noticeably improved. From the second fan defined in the 4th analysis onwards, the CO_2 and H_2S gases in the tunnel are at 0 ppm values.

4 Conclusion

The results of the analyses highlighted the importance of proper ventilation design in the effective removal of geothermal gases from infrastructure projects. By optimizing fan placement, duct configurations, and airflow rates, CFD simulations demonstrated the potential to minimize the accumulation of geothermal toxic gases in tunnel environments.

This study demonstrates that the use of CFD methodology provides valuable information on the dynamics of geothermal gas distribution and facilitates the development of efficient ventilation systems to protect both human health and the environment in infrastructure projects. The analyses performed in this study underline the importance of CFD in addressing the challenges posed by geothermal gas leaks and highlight its role in ensuring the safety and sustainability of such projects. Likewise, the analysis and testing of different removal methods for geothermal gas and liquid leaks that may occur in different geometries, structures, resource types can be performed with the CFD method. Thus, time and resource management can be realized efficiently.

Acknowledgements

This paper is derived from the work carried out within the scope of TUBITAK TEYDEB project number 3220754 named THE USE OF NITROGEN CYLINDERS IN FIRE FIGHTING IN UNDERGROUND MINES.

References

ANSYS Training Courses (2016). SpaceClaim Introduction, Meshing Introduction, Fluent Introduction.

Çengel Y. A., Cimbalak J. M. (2004). Fluid Mechanics: Fundamentals and Applications.

Goodfellow H. D., Kosonen R. (2020). Industrial Ventilation Design Guide: Volume 1: Fundamentals.

Liu C., Yeoh G. H., Tu J. (2007). Computational Fluid Dynamics: A Practical Approach.

Ferziger, J.H., & Perić, M. (2002). *Computational Methods for Fluid Dynamics*.