APPLICATION OF COMPUTATIONAL FLUID DYNAMICS (CFD) IN PREDICTING AIRFLOW BEHAVIOR IN COPPER MINES

Arkadiusz Macek *[1]

[1] *Wroclaw University of Science and Technology, Faculty of Mechanical Engineering, Wrocław*

*Corresponding author: arkadiusz.macek@pwr.edu.pl

Keywords: CFD, fluid dynamics, copper mine, air flow, ventilation

Numerical fluid mechanics is a powerful tool for simulating and predicting airflows in many ranges of the industry, including underground mines, where ventilation plays a crucial role in ensuring worker safety and optimizing operational efficiency. This paper presents a Computational Fluid Dynamics (CFD) approach to modeling airflow behavior in copper mine tunnels. The CFD simulations are based on solving the Navier-Stokes equations, which describe the motion of viscous fluids, and incorporate turbulence models to account for the chaotic and variable flow patterns typically observed in mine ventilation. The study focuses on the effect of tunnel shape on airflow patterns, keeping factors such as air velocity, tunnel cross-sectional area, and other operating conditions constant. By analyzing different tunnel shapes and their impact on airflow behavior, the numerical simulations provide insights into how changes in shape alone influence pressure distribution, air velocity fields, and turbulence intensity within the mine tunnels. The results help identify optimal tunnel shapes that improve ventilation efficiency and maintain appropriate air quality without altering other geometric or operational parameters. Overall, this research demonstrates the significant role of tunnel shape in determining airflow behavior, offering valuable guidelines for designing safer and more energy-efficient ventilation systems in copper mines.