International Journal of Mining Science and Technology 25 (2015) 635–640

Contents lists available at ScienceDirect

International Journal of Mining Science and Technology

journal homepage: www.elsevier.com/locate/ijmst

CFD modeling of methane distribution at a continuous miner face with various curtain setback distances

Zhou Lihong [a,*], Pritchard Christopher [b], Zheng Yi [a]

[a] *Office of Mine Safety and Health Research, National Institute for Occupational Safety and Health, Pittsburgh, PA 15236, USA*
[b] *Spokane Mining Research Division, National Institute for Occupational Safety and Health, Spokane, WA 99207, USA*

ARTICLE INFO

Article history:
Received 6 December 2014
Received in revised form 11 January 2015
Accepted 28 February 2015
Available online 27 June 2015

Keywords:
Mine ventilation
Computational fluid dynamics
Continuous mining face
Airflow pattern
Methane distribution

ABSTRACT

Knowledge of the airflow patterns and methane distributions at a continuous miner face under different ventilation conditions can minimize the risks of explosion and injury to miners by accurately forecasting potentially hazardous face methane levels. This study focused on validating a series of computational fluid dynamics (CFD) models using full-scale ventilation gallery data that assessed how curtain setback distance impacted airflow patterns and methane distributions at an empty mining face (no continuous miner present). Three CFD models of face ventilation with 4.6, 7.6 and 10.7 m (15, 25, and 35 ft) blowing curtain setback distances were constructed and validated with experimental data collected in a full-scale ventilation test facility. Good agreement was obtained between the CFD simulation results and this data. Detailed airflow and methane distribution information are provided. Elevated methane zones at the working faces were identified with the three curtain setback distances. Visualization of the setback distance impact on the face methane distribution was performed by utilizing the post-processing capability of the CFD software.

1. Introduction

Mining is often listed one of the world's most dangerous occupations. Many serious accidents have occurred in various parts of the world, often with significant loss of life. Methane emissions and accumulations at a working face in underground coal mines can lead to a major explosion if not managed effectively. Mine safety and health administration (MSHA) investigators determined that an accumulation of methane gas was the initiator of a massive explosion at the Upper Big Branch (UBB) coal mine, West Virginia, on April 5, 2010, claiming the lives of 29 miners. Effective ventilation is the primary approach to provide an adequate supply of fresh air to the working areas to dilute and remove harmful contaminants.

The distribution of ventilation airflow, as well as methane, at continuous miner faces can be affected by several factors, such as the method of ventilation employed (blowing or exhausting), rib to ventilation curtain distance, curtain setback distance, volume of air delivered at the end of the curtain, and volume of methane released in the face area. Among these factors, the curtain setback distance, known as the distance from the end of the ventilation curtain to the working face, is one of the critical factors to determine if sufficient airflow can be directed to the face to effectively sweep the face of harmful gases. Luxer [1] evaluated the air movement at an empty face with curtain setback distances varying from 1.5 m (5 ft) to 6.1 m (20 ft) by observing the movement of smoke generated with chemical tubes. Air velocities measurements were made using vane anemometers and chemical tubes, but vane anemometers can only measure flow velocity, not direction. Airflow patterns determined based this method was not effective in identifying the turbulence zone. Taylor et al. [2] replaced vane anemometers with ultrasonic anemometers which measure the magnitude and direction of velocity to perform measurements at a continuous miner face with 4.6, 7.6 and 10.7 m (15, 25, and 35 ft) setback distances to investigate the impact of the setback distance on airflow and methane distribution. However, the methane distributions in the both research methods were determined based on methane measurements from a limited number of sampling locations. The methane distribution map largely depends on the sampling locations, which cannot show continuous change in methane distribution. With the growing popularity of computational fluid dynamics (CFD) technology in mining research, its powerful analyzing and visualization abilities can benefit through performing studies which are impractical to conduct in a laboratory or field test facility. Most of CFD software can provide three-dimensional visualization via its post-processing software that creates maps of velocity vectors, streamlines, iso-value

---

* Corresponding author. Tel.: +1 412 3865711.
E-mail address: itn2@cdc.gov (L. Zhou).

http://dx.doi.org/10.1016/j.ijmst.2015.05.018
2095-2686/

contours, etc. Wala et al. [3] successfully applied CFD to interpret airflow and methane distribution at a continuous miner face, showing that CFD has the ability to predict, evaluate and design effective face ventilation systems. However, curtain setback distances were not varied in their study, which focused on airflow and methane behavior at a single 10.7 m (35 ft) setback distance in both box cut and slab cut configurations.

CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier–Stokes and allied equations) for predefined geometries and boundary conditions and has seen dramatic growth in use over the last several decades. This technology has been widely applied to various engineering applications in the mining field starting in 1990s [4–6]. Since then, this technique has modeled fluid flow behavior in underground mine workings, helping engineers analyze complex airflow fields and contaminant transport to design more effective ventilation systems [3,7–13].

The major objectives of this paper are to (1) develop a CFD model of a continuous mining machine face ventilated with a blowing curtain at various setback distances, and (2) to validate the base model with previously collected full-scale ventilation gallery test data. This analysis will illustrate the effects of three different curtain setback distances of 4.6, 7.6 and 10.7 m (15, 25 and 35 ft) on airflow patterns and methane distributions at the working face, which is difficult to determine from experimental testing alone.

2. Validation study

Some advantages of CFD over experimental approaches include low cost, speed, and the ability to simulate both realistic and ideal conditions. However, as a numerical solution method for complex problems, CFD cannot avoid engineering simplifications and mathematical approximations needed to effectively perform this research. In CFD, the conceptual model is dominated by partial differential equations (PDEs) for conservation of mass, momentum, and energy. There is no analytical solution for the PDEs of turbulent flow. The credibility of CFD as an engineering tool depends on the quantification of the error/uncertainty of the results. The CFD model must then be validated against a range of relevant experimental data before being successfully applied to parametric studies. Only after this validation can a CFD model generate reliable numerical results.

Validation is the primary means to assess accuracy and reliability in computational simulations such as CFD, and is done by comparing the accuracy of the computational results with the experimental data; as in this case, the experimental data from tests conducted at the national institute for occupational safety and health (NIOSH) full scale ventilation test gallery by Taylor et al. [2]. Over the last few years, office of mine safety and health research (OMSHR) researchers at NIOSH have conducted a series of experimental tests at the full-scale ventilation gallery to investigate how different operating conditions impacted airflow patterns and methane distributions near the face of a continuous mining machine [2,14]. These data sets remain accurate depictions of airflow and methane levels under a number of face ventilation configurations. Some of this experimental data has been used by Wala et al. and Kollipara et al. to validate their respective CFD models [3,9]. Wala et al. compared their CFD simulation results with Taylor et al.'s experimental data for both airflow and methane concentrations in an empty (containing no equipment) face area ventilated with a blowing curtain [3,14]. These authors were the first to validate CFD simulation results for methane concentration in the face area using full-scale, mine-related benchmark experiments. Kollipara et al. validated their model using Taylor et al.'s airflow data but did not consider variations in methane [9]. In this paper, CFD models of an empty continuous miner face were developed at three curtain setback distances of 4.6, 7.6 and 10.7 m (15, 25 and 35 ft) and validated using experimental airflow and methane concentration data generated by Taylor et al. [2].

2.1. Experimental test

The NIOSH ventilation test gallery is a full-scale facility located in an "L-shaped" building (Fig. 1), initially designed to conduct deep-cut face ventilation research. Airflow and methane conditions can be varied in the gallery to simulate a wide range of ventilation scenarios found in face areas. A vane-axial exhaust fan draws air through regulator doors and across the gallery face. The airflow volume flowing through the test area is adjusted by opening and closing the regulator doors. The test area dimensions are 2.2 m (7 ft) high and 5 m (16.5 ft) wide, which can be narrowed by moving a mobile wall inward. Tests are conducted with either blowing or exhausting face ventilation using curtains to direct air to the simulated face. For blowing ventilation, the curtain is placed 0.6 m (2 ft) from the left wall, directing air toward the face area, and the setback distance can be adjusted as needed.

The continuous miner was removed from the working face during these tests. A uniform methane gas release from the face was simulated using a manifold consisting of four 3 m (10 ft) long horizontal copper pipes drilled on the top and bottom with 2 mm (1/16 in) diameter holes 51 mm (2 in) apart. The four pipes were equally spaced and located 0.1 m (4 in) on the simulated face (Fig. 2). During evaluation of ventilation systems, it is important to precisely know the flow rate of gas entering the gallery and to ensure it remains constant for any given test. Gas flow rates were set and monitored using a rotameter.

Airflow velocity and methane concentration data were measured at the mid-line room level, 1.1 m (3.5 ft) from the roof. The 36 sampling locations in four columns and nine rows are shown in Fig. 3. The first four rows of 16 sampling locations were used to measure airflow and methane concentrations for the 4.6 m (15 ft) curtain setback distance test, 24 sampling locations for the 7.6 m (25 ft) setback distance, and all 36 sampling locations for the 10.7 m (35 ft) setback distance.

To maintain consistency with a series of tests performed in the ventilation gallery, an airflow of 2.8 m^3/s (6000 ft^3/min) and gas release rate of 0.016 m^3/s (34 ft^3/min) were used to validate the CFD model in this study.

2.2. CFD model development

2.2.1. Geometry

The geometry of the continuous miner face model (Fig. 4), including the ventilation curtain and methane gas release manifold, was developed using the ANSYS geometry model tool Design Modeler, incorporating the same dimensions as the

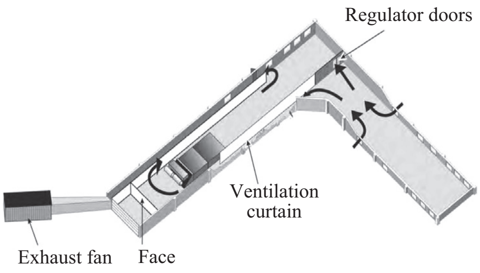


Fig. 1. NIOSH ventilation test gallery [2].

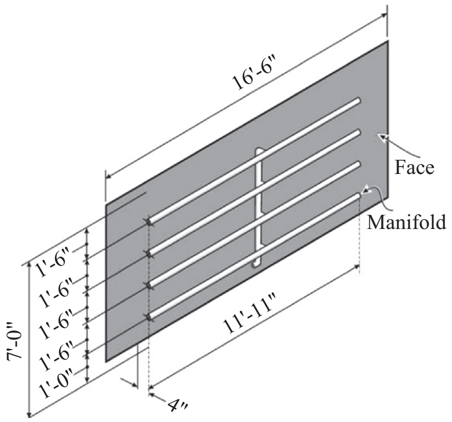


Fig. 2. Gas manifold for methane release at the ventilation test gallery face [2].

ventilation gallery. The CFD gas release model was simplified by assuming the gas was liberated from the outer surface of the tubes instead of through multiple holes. In this experiment, three CFD models with curtain setback distances of 4.6, 7.6 and 10.7 m (15, 25, and 35 ft) were constructed to investigate how these setback distances might affect face airflow patterns and methane distributions. The air density 1.220 kg/m^3 the same as it was measured in the gallery test was used in the models. Three boundary conditions were applied to the CFD models (Fig. 4):

- Velocity inlet: 2.18 m/s (430 ft/min).
- Gas inlet: 0.016 m^3/s (34 cfm).
- Pressure outlet: 0 Pa.

2.2.2. Meshing

Fig. 5 illustrates the computational mesh generated with ANSYS Meshing 13.0 used in this study. Three separate zones with two interfaces (Fig. 5) were used to apply hexahedral meshes to the curtain zone and middle zone. As a comparison, a tetrahedral mesh can be created with far fewer cells than the equivalent hexahedral element mesh, possibly reducing computational expense. Greater numbers of elements require more computer resources (memory/processing time). A total of 1,282,000 cells were generated in the models.

CFD models in this paper use a second-order upwind scheme and the SIMPLE algorithm. The Realizable k–ε model was employed to model the turbulent flow inside the face area and was solved using the commercial CFD program ANSYS Fluent 13.0. The Specie Transportation model was employed to simulate methane gas transportation in the face area.

2.2.3. Comparison of CFD results and experimental tests

The airflow velocities and the methane concentrations at each location in the mid-level plane were extracted and compared with the gallery experimental data. A mid-level plane and corresponding sampling locations were specified and defined in the CFD model in the ANSYS post-processing software CFD-POST. Airflow velocities at the sampling locations were compared point-to-point between the experimental tests and CFD results (Fig. 6).

Fig. 6 shows the comparisons of the experimentally-measured air velocity and the modeled air velocity at each sampling location for the 2.8 m^3/s (6000 cfm) flow rate at the three different setback distances. Only the first 25 sampling locations are shown in Fig. 6c for the 10.7 m (35 ft) case due to space limitations. The simulated air velocities and the experimental data generally show good agreement.

Fig. 7 compares the experimentally-measured and CFD-modeled methane concentrations at each sampling location and at each setback distance. Generally, these three figures show a reasonable match between the various data. For the 4.6 m (15 ft) setback distance case, the CFD-modeled methane concentrations are in a good agreement with those measured at most sampling locations except for Nos.7, 8, and 10. Similarly, in Fig. 7b for the 7.6 m (25 ft) setback distance case, the modeled methane concentrations at sampling locations 8, 10, 12, and 16 also show poor agreement with the measured methane data. As seen in Fig. 7c, CFD modeled the lowest and peak methane concentrations from the fresh intake side and return side respectively very well, but underestimated the methane concentrations for the sampling points in the mid-face area.

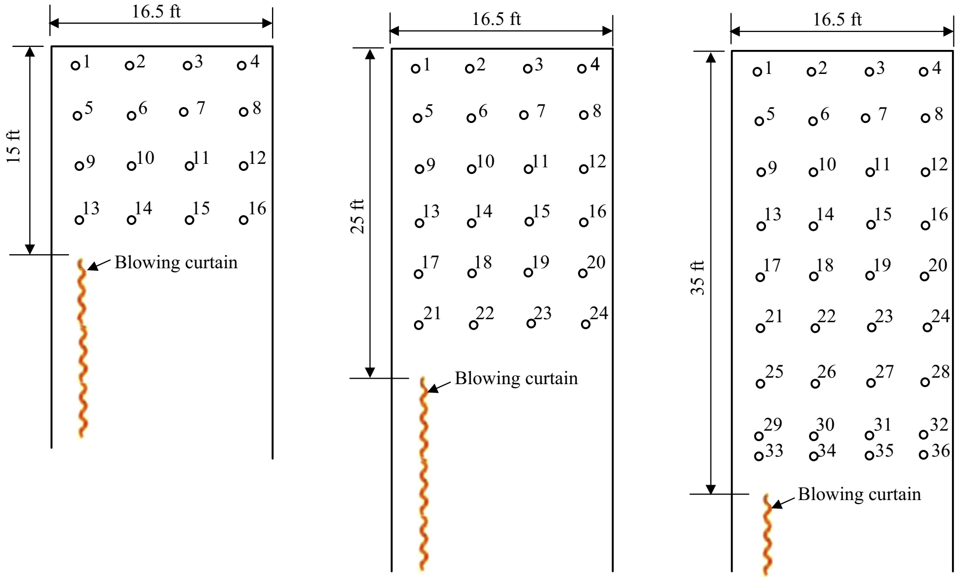


Fig. 3. Sampling locations for the 4.6 m (15 ft), 7.6 m (25 ft), and 10.7 m (35 ft) curtain setback distances [2].

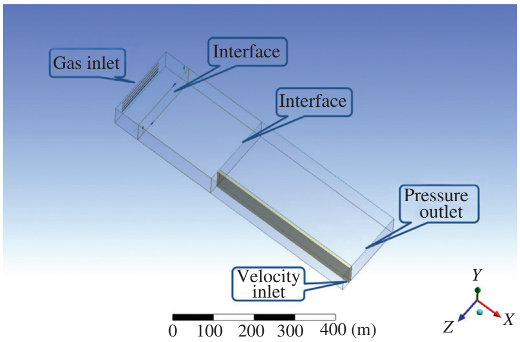


Fig. 4. Geometry and boundary conditions of the CFD model (25 ft setback as an example).

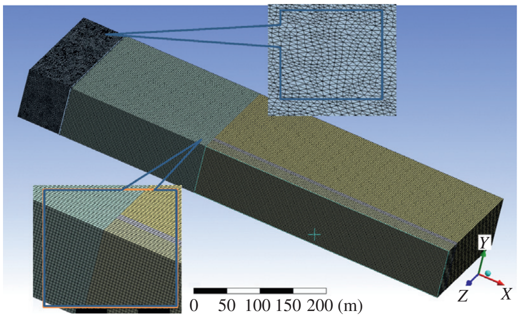
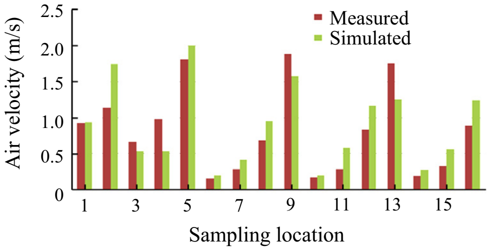
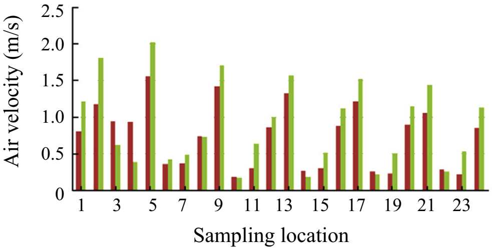


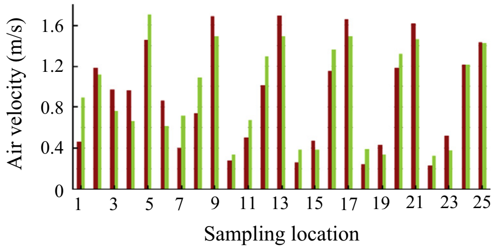
Fig. 5. Isometric view of the computational mesh of the flow domain.



(a) 4.6 m (15 ft)

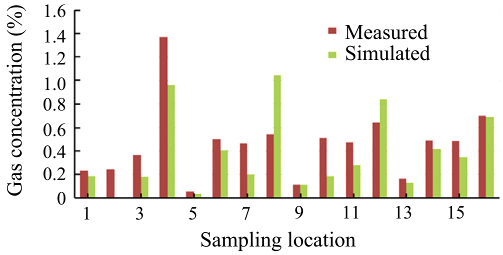


(b) 7.6 m (25 ft)

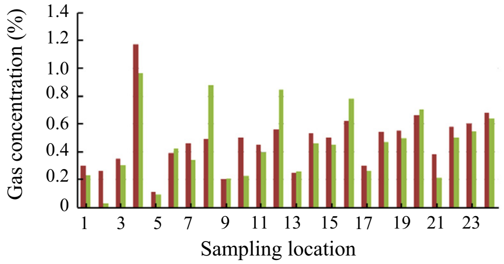


(c) 10.7 m (35 ft)

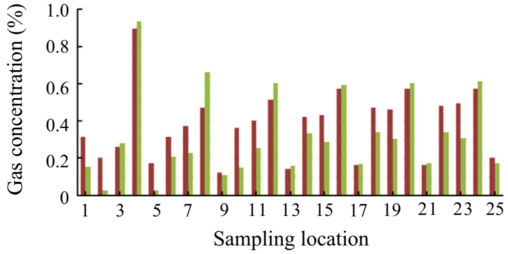
Fig. 6. Comparison of measured and modeled air velocities at the sampling locations for various setback distances.



(a) 4.6 m (15 ft)



(b) 7.6 m (25 ft)



(c) 10.7 m (35 ft)

Fig. 7. Comparison of the measured and simulated gas concentration at all sampling locations.

Many factors can contribute to the differences between the measured and CFD- modeled results at the above sampling locations. The authors agree with Wala et al. [3] that (1) numerical simulation was carried out with a steady-state assumption, whereas the actual flow behavior in the tests may have been unsteady; (2) velocity and methane concentration measurements were not taken simultaneously, which may increase the error if the flow was unsteady; and (3) the location and the way methane was introduced into the face were different for the numerical and experimental simulations.

Figs. 6 and 7 provide visual comparisons of modeled and measured air velocity and methane concentrations. However, a quantitative assessment of the degree to which the models match the observations can provide more confidence in model performance. Among the current model evaluation methods, the coefficient of determination R^2 is one of the most commonly and widely used. The coefficient of determination R^2 is the square of the Pearson's product-moment correlation coefficient and can be used to describe how well simulated data match measured data [15,16]. R^2 values range from 0 to 1, with higher values indicating less error variance, and with values greater than 0.5 considered acceptable [17,18]. Table 1 shows the coefficient of determination for each simulation and indicates better agreement between modeled and

Table 1
Value of R^2 for each case.

	15 ft		25 ft		35 ft	
	Airflow	Methane	Airflow	Methane	Airflow	Methane
R^2	0.87	0.78	0.90	0.81	0.92	0.85

measured airflows than modeled and measured methane levels. These values for airflow range from 0.87–0.92 and 0.78–0.85 for methane.

3. Visualization of airflow pattern and methane distribution

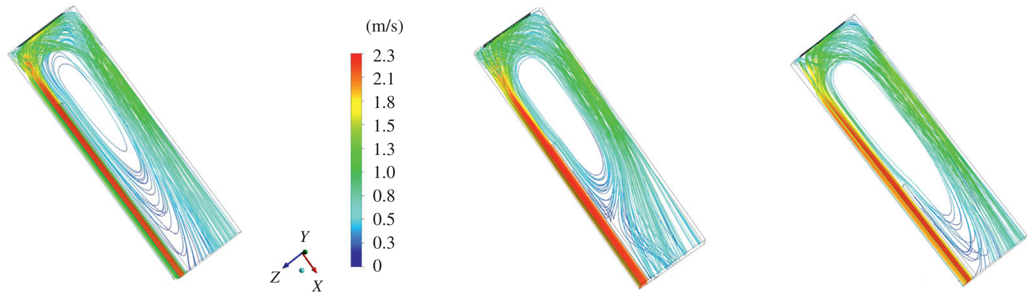
The validation of each case provides confidence to apply CFD for modeling airflow patterns and methane distributions at each setback distance. Model results are displayed via ANSYS CFD-Post. Fig. 8 illustrates the airflow streamlines for 4.6, 7.6, and 10.7 m (15, 25, and 35 ft) setback distances. These figures show that air was directed from the curtain to the face and the majority of air flowed back along the right side of the face. There are also minor air currents separating from the primary airstream forming a central recirculation zone. Most importantly, the size of the recirculation zone appears to increase with setback distance.

Fig. 9 shows the face area methane distributions for the various setback distances. Data collected from the ventilation gallery are not sufficient to create the entire picture of the methane cloud. Fortunately, ANSYS CFD-Post, an ANSYS post-processor, displays a three dimensional methane cloud based on the discrete methane concentration data set obtained from the simulation. Fig. 9 confirms previous work showing the highest methane level exists at the off-curtain side face corner and accumulations of gas increase in size with setback distance.

4. Integration of CFD modeling with experimental tests

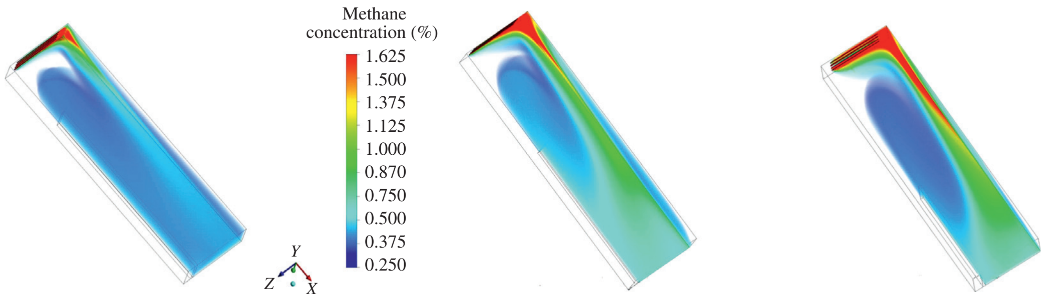
This paper delivers a clear concept for integrating CFD modeling with experimentation, in addition to investigating the impact of setback distance on the airflow profile and methane distribution at a continuous miner face. A well-designed full-scale experiment is reliable but can be time consuming and costly. In contrast, CFD provides powerful visualization capabilities that can evaluate the performance of a wide range of system configurations without the time, expense, and disruption required to make actual changes in test facilities. However, its credibility is often questioned due to discretization error, round-off error, and algorithm error associated with this approach. The practical strategy applied in this study validated the CFD model against experimental test data to more accurately interpret the results and also to extend the validated results for work that could not be done in a laboratory setting.

One of the significant advantages of CFD analyses over experimental approaches is the improved visualization and interpretation of the results. CFD can provide three-dimensional visualization via its post-processing software that creates maps of velocity vectors, streamlines, iso-value contours, etc. The methane distributions are well interpreted in the CFD modeling with a three-dimensional methane cloud clearly displayed at the face. More specifically, the methane distributions can be displayed at any user-specified location within the simulation domain, including lines, planes or volumes. Research could be significantly enhanced if the CFD simulation and full scale experiments could be integrated, as has been done in this study. Through CFD, the design and reliability of experiments can be significantly enhanced, the scope of experimental measurements extended and the credibility of the simulation results enhanced by the availability of suitable measurements from experimentation. CFD provided researchers with improved visualization of the methane distribution over the entire face and with insight into the dynamic behavior of a physical system that would otherwise be very difficult, time-consuming, and expensive to achieve using experimental methods.



(a) 4.6 m (15 ft) (b) 7.6 m (25 ft) (c) 10.7 m (35 ft)

Fig. 8. Airflow streamlines for different setback distances.



(a) 15 ft (4.6 m) (b) 25 ft (7.6 m) (c) 35 ft (10.7 m)

Fig. 9. Visualization of the face methane cloud for different setback distances.

5. Conclusions and discussions

ANSYS Fluent has the ability to present a comprehensive picture of a methane cloud at a continuous miner face which is difficult to obtain using gallery experimental data alone. The successful integration of experimentation with CFD modeling has shown potential to solve complicated test scenarios. A proven, practical strategy is to validate the CFD model with experimental test data and then use the CFD model for analyses that cannot be conducted practically in the lab.

A validated CFD model can improve the fundamental understanding of airflow and methane distribution in an underground mine working face. Three CFD models of a continuous miner face with setback distances of 4.6, 7.6 and 10.7 m (15, 25, and 35 ft) were constructed and validated with experimental ventilation gallery test data. Good agreement in both airflow and methane levels were achieved for each case. This work found that a zone of recirculation existed and increased in size with curtain setback distance. This distance also impacted the extent of the methane cloud at the face, with the size of the methane cloud increasing with setback distance, confirming previous research done by Taylor et al. [2].

A study limitation is the simplification of the CFD gas release model. In gallery tests, the gas enters the face from hundreds of small holes drilled at the top and bottom of each pipe. Due to computer resource limitations, the CFD model utilized the surfaces of the gas pipes as the release source, in lieu of specifying each hole individually. To maintain the same total gas release rate as the experiments, the CFD model release velocity was modified to be much smaller than that used in the gallery tests. This study only takes into account the empty working face. Adding the presence of a continuous miner would have considerable impact on face airflow and methane distribution.

Disclaimer

The findings and conclusions in this report are those of the authors and do not necessarily represent the views of the National Institute for Occupational Safety and Health. Mention of any company or product does not constitute endorsement by NIOSH.

Acknowledgments

The authors sincerely thank Chuck Taylor, retired NIOSH OMSHR mine ventilation researcher, for his valuable assistance in interpreting the experimental test data. The authors are grateful to Yuan Liming for his assistance on CFD modeling.

References

[1] Luxner JV. Face ventilation in underground bituminous coal mines: airflow and methane distribution patterns in immediate face area-line brattice. Washington DC: Bureau of Mines; 1969.
[2] Taylor CD, Chilton JE, Goodman GVR. Guidelines for the control and monitoring of methane gas on continuous mining operations. Depart Health Hum Serv, Inform Circulat 2010;3:322–40.
[3] Wala AM, Vytla S, Taylor CD, Huang G. Mine face ventilation: a comparison of CFD results against benchmark experiments for the CFD code validation. Min Eng 2007;59(10):49–55.
[4] Gong R, Bhaskar R. Three-dimensional mathematical modeling of airflow at a continuous miner face. AIME Trans 1992;292:1888–93.
[5] Heerden J, Sullivan P. The application of CFD for evaluation of dust suppression and auxiliary ventilation systems used with continuous miners. In: Proceedings of the 6th US mine ventilation symposium, US; 1993.
[6] Wala AM, Yingling JC, Zhang J. Evaluation of the face ventilation systems for extended cuts with remotely operated mining machines using three-dimensional numerical simulations. In: SME pre-print, SME annual meeting, Florida; 1998.
[7] Hargreaves DM, Lowndes IS. The computational modeling of the ventilation flows within a rapid development drivage. Tunnel Undergr Space Technol 2007;22:150–60.
[8] Aminossadati SM, Hooman K. Numerical simulation of ventilation air flow in underground mine workings, In: Proceedings of 12th U.S./North American mine ventilation symposium, Nevada; 2008.
[9] Kollipara VK, Chugh YP, Relangi DD. A CFD analysis of air flow patterns in face area for continuous miner making a right turn cut, In: SME Pre-print, SME annual meeting, Washington State; 2012.
[10] Xu G, Luxbacher K, Ragab S, Schafrik S. Development of a remote analysis method for underground ventilation systems using tracer gas and CFD in a simplified laboratory apparatus. Tunn Undergr Space Technol 2013;33:1–11.
[11] Ranjan M, Kumar K, Ghosh S. Mine ventilation in a board and pillar mines using CFD. Int J Emerg Technol Adv Eng, ICERTSD 2013;3(3):389–93.
[12] Sasmito AP, Birgersson E, Ly HC, Mujumdar AS. Some approaches to improve ventilation system in underground coal mines environment-a computational fluid dynamics study. Tunnel Undergr Space Technol 2013;34:82–95.
[13] Wala AM, Yingling JC, Zhang J, Ray R. Validation study of computational fluid dynamics as a tool for mine ventilation design, In: Proceedings of the 6th international mining ventilation congress, Pittsburgh; 1997.
[14] Taylor CD, Timko RJ, Thimons ED, Mal T. Using ultrasonic anemometers to evaluate factors affecting face ventilation effectiveness. In: The 2005 SME annual meeting, Salt Lake City; 2005.
[15] Legates DR, McCabe GJ. Evaluating the use of "goodness-of-fit" measures in hydrologic and hydro climatic model validation. Water Resour Res 1999;35(1):233–41.
[16] Moriasi DN, Arnold JG, Van Liew MW, Bingner RL, Harmel RD, Veith TL. Am Soc Agric Biol Eng 2007;50(3):885–900.
[17] Santhi C, Arnold JG, Williams JR, Dugas WA, Srinivasan R, Hauck LM. Validation of SWAT model on a large river basin with point and nonpoint sources. J Am Water Resour Assoc 2001;75(5):1169–88.
[18] Van Liew MW, Arnold JG, Garbrecht JD. Hydrologic simulation on agricultural watersheds: choosing between two models. Trans Am Soc Agric Eng 2003;46(6):1539–51.