Mine face ventilation: a comparison of CFD results against benchmark experiments for the CFD code validation

A.M. WALA, S. VYTLA, C.D. TAYLOR AND G. HUANG

A.M. Wala, member SME, is associate professor with the Department of Mining Engineering, University of Kentucky, Lexington, KY; **S. Vytla** is graduate student Ph. D, Department of Mechanical and Materials Engineering, Wright State University, Dayton, OH; **C.D. Taylor** is industrial hygienist with NIOSH Pittsburgh Research Laboratory, Pittsburgh, PA; **G. Huang** is professor and chair, with the Department of Mechanical Engineering, Wright State University, Dayton, OH.

Abstract

Computational fluid dynamics (CFD) is a continually developing numerical technique by which complex fluid-flow problems can be solved on computers. To be able to transfer CFD expertise to analyze and design a face-ventilation system, a comprehensive validation study of current CFD tools against mining-related benchmark experiments is required. In the last four years, such studies, funded by National Institute for Occupational Safety and Health (NIOSH) under Grant #R01 CCR415822, were conducted in the Department of Mining Engineering at the University of Kentucky. A wide array of ventilation schemes and mining configurations were considered. Recently, an additional study was performed in cooperation with the NIOSH Pittsburgh Research Laboratory. This study was dedicated to the methane behavior in the empty (containing no equipment) face area with a blowing curtain and a 10.7-m (35-ft) setback. In this paper, the authors present and discuss the computer-simulation data and compare them with data collected during the laboratory studies at the NIOSH lab.

Introduction

The use of extended-cut (deep-cut) mining with remotely controlled continuous miners is common in the U.S. coal industry. Operators adopt this method to maximize the productivity of their continuous miner sections. The accompanying higher advance rates introduce the problem of directing a sufficient quantity of air to the face when setback distances are 6 m (20 ft) and greater. This can require an increase in the quantity of air directed behind the blowing curtain. Higher air velocities are a consequence that, in turn, leads to the entrainment of more dust generated during the coal-extraction procedure. Frequently, the increased dust entrainment is countered by the use of large-capacity scrubbers that recirculate and filter air in the immediate face area. These circumstances give rise to many concerns regarding the health and safety of miners.

The former U.S. Bureau of Mines (currently NIOSH), the Mine Safety and Health Administration (MSHA) and the coal industry have conducted significant studies to evaluate the performances of face ventilation systems through the use of full-scale tests (underground/surface) or scaled physical modeling. These studies have led to recommendations that tend to make these systems more effective. However, because of the aerodynamic complexity in the face area arising from the variety of ventilation arrangements amid limitations of the experimental methods, doubts still exist when evaluating the effectiveness of such a system.

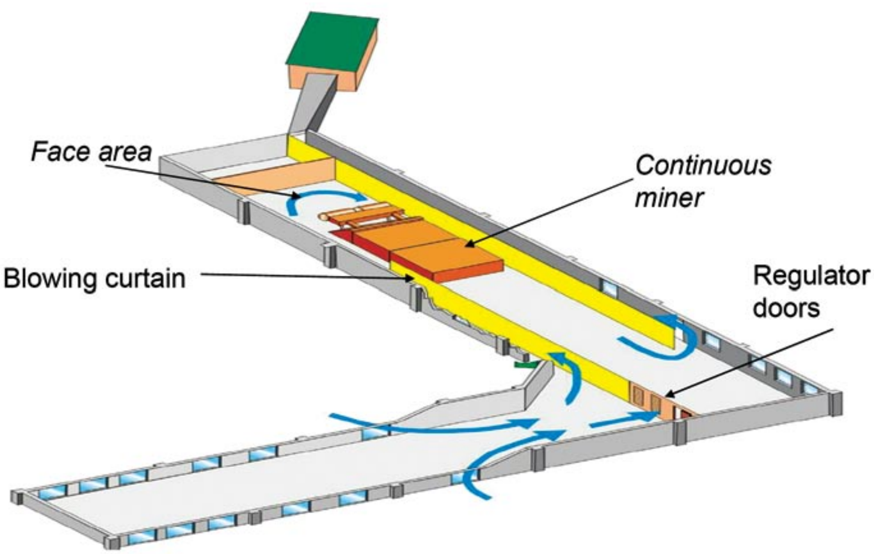
The design of a balanced ventilation system for the scenarios described above requires consideration of the system in a three-dimensional, rather than a two-dimensional, manner. Traditional theoretical and experimental methods are available for obtaining useful results, but the former is limited to simple geometries and experimental methods are often slow and both approaches are limited in the completeness, accuracy and generality of the results that they provide. Computational fluid dynamics (CFD) is a promising design methodology by which many complicated fluid flow problems can be solved with numerical codes. CFD embraces a variety of technologies, including mathematics, computer science, engineering and physics, and has the potential to generate face-ventilation designs without the previously mentioned disadvantages.

During the last four years, a validation study, funded by NIOSH under Grant #R01 CCR415822, was carried out in the Department of Mining Engineering at the University of Kentucky. The goal of this study was to prove a CFD code's ability to predict, evaluate and design effective face-ventilation systems. To achieve this goal, the authors did the following:

- designed and built a scaled physical model of selected face-ventilation systems (Wala et al., 2003),
- performed a series of measurements using particle image velocimetry (PIV) (Turner et al., 2002) to deter-

FIGURE 1

Ventilation test gallery.



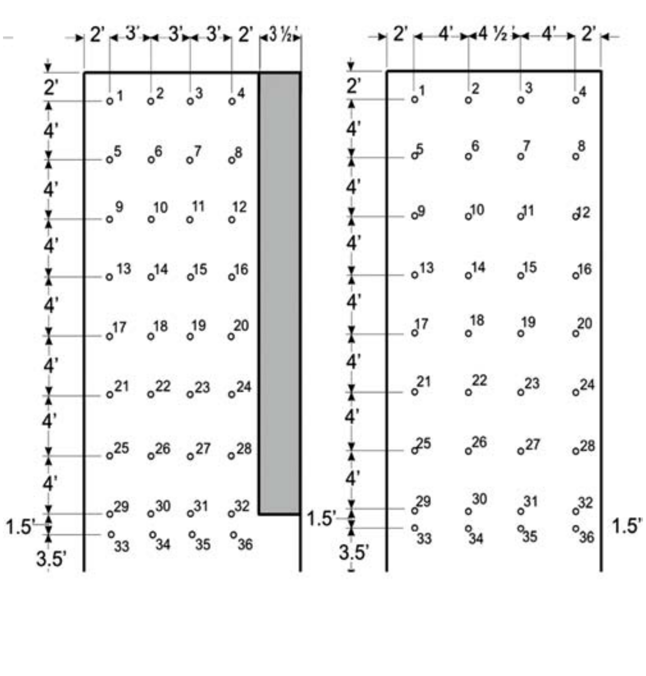
mine the flow patterns in the scaled physical model,

- developed three-dimensional numerical models of the physical model and performed computer simulation studies using CFD codes (Wala et al., 2000, 2001, 2003, 2004) and
- compared experimental and numerical data for the purpose of CFD code validation (Wala et al., 2003, 2004, 2005).

In all experimental and numerical studies performed so far, the whole attention was concentrated on understanding the airflow distribution (airflow patterns) at the face area. The methane gas, as the second species, was never introduced during the experimental study using scaled physical models because of the safety issue. Therefore, after completion of the project described

FIGURE 2

Sampling locations.



above, there was a need for studies concerning the methane behavior (methane distribution) in the face area in conjunction with ventilation.

Recently, an effort was made to perform a comprehensive validation study of the CFD codes for simulation of the flow and methane behavior at the face area. These studies were the combined effort of the Department of Mining Engineering at the University of Kentucky and the NIOSH Pittsburgh Research Center. The experimental part of the study, which was coordinated by the mining engineering faculty, was carried out by researchers at the NIOSH Pittsburgh Research Laboratory using a full-scale ventilation gallery. The procedures for the laboratory tests were designed at the University of Kentucky and were discussed with the research staff at the NIOSH laboratory to provide data for validation of the CFD simulation results.

Experimental studies at the NIOSH laboratory

Although the results of the experimental studies have been previously reported (Taylor et al., 2005), to make this paper easier to understand, the authors have included the following information from the earlier paper:

Test facility. Tests were conducted in the NIOSH Pittsburgh Research Laboratory's Ventilation Test Gallery, which is shown in Fig. 1. One side of the gallery was designed to simulate a mining entry with a 2.2-m- (7-ft-) high roof and with ribs 5 m (16.5 ft) apart. To be able to study the flow and methane distribution in the empty face area (no equipment) during two parts of the continuous mining cut sequence, i.e., box cut and slab cut, the 5-m (16.5-ft) total entry width was reduced to 4 m (13 ft) by building a wall 1 m (3.5 ft) from the right rib.

The exhaust fan draws approximately 5.9 m^3/s (12,500 cfm) of air through the gallery. A part of this air was directed toward the entry face using a curtain that was constructed 0.6 m (2 ft) from the left side of the entry. The curtain was positioned so that the setback distances between the curtain and the face were 10.7 m (35 ft). Regulator doors were adjusted to provide intake flows behind the curtain of either 2.8 or 4.7 m^3/s (6,000 or 10,000 cfm).

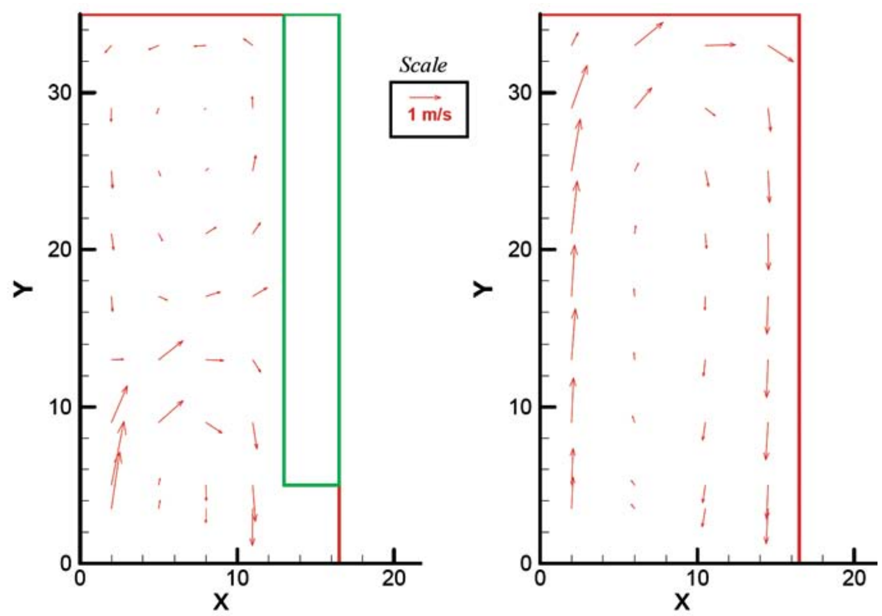
Airflow and methane measurements. Airflow and methane concentration measurements were made at the same sampling locations. There were a total of 36 sample locations, arranged in four columns and nine rows, between the curtain and the face. The sampling locations for the 10.7-m (35-ft) setback distance and two entry widths are shown in Fig. 2.

The three-dimensional airflow measurements were made between the end of the curtain and the face at the mid-height of the entry, using "Windmaster" three-axis ultrasonic anemometers manufactured by Gill Instruments Ltd., Great Britain. However, two-dimensional components of velocity in the plane of measurement are provided for comparison. The methane gas was released into the test area through four 3-m- (10-ft-) long horizontal copper pipes that were located at the mining face. The pipes were located 100 mm (4 in.) away from the face, and they were equally spaced horizontally to provide a relatively uniform release of gas. On the top and bottom of each of the 3-m- (10-ft-) long pipes, 2-mm- (1/16-in.-) diameter holes were drilled 65 mm (2.5 in.) apart. For the slab-cutting scenario, the methane flow rate was 0.016 m^3/s (34.0 cfm). However, for the box cut mining scenario, due to the flow separation phenomena that resulted in higher concentrations at the face, the methane flow into the gallery was reduced to 0.0032 m^3/s (6.8 cfm) to prevent methane concentration levels in the gallery from exceeding 2 percent.

To make methane measurements, four air-sampling tubes were suspended from the overhead support system. The hose inlets were positioned so the methane concentration measurements were made at the three entry heights; top level being 1.1 m (4 ft) from roof, the mid-level and bottom level being 1.1 m (4 ft) from the floor.

FIGURE 3

Flow patterns: box-cut scenario (left) and slab-cut scenario (right), for 10.7-m (35-ft) setback with 2.8 m^3/s (6,000 cfm) airflow.



Results — airflow patterns and methane distribution. The measured airflow patterns in the face area (area between the end of the blowing curtain and the face) of the ventilation gallery are shown in Fig. 3 for both box-cut and slab-cut scenarios and a curtain flow of 2.8 m^3/s (6,000 cfm).

The contours in Fig. 4 show the distribution of the methane concentration at the face area when the amount of air delivered for ventilation behind the curtain was 2.8 m^3/s (6,000 cfm).

Computer simulation study for the CFD code validation

There are three major steps in any CFD solution process:

- preprocessing (mesh generation),
- processing (CFD simulation and refinement/adaptation of grid) and
- post-processing (visualization and analysis of results).

FLUENT 6.X, a commercially available CFD solver, together with the GAMBIT mesh generator (preprocessor), which comes as a package along with FLUENT, was used to simulate the methane and flow behaviors for the same scenarios used during the laboratory tests. The results of these simulations are visualized using the post-processing capabilities of FLUENT and are graphically shown using the Excel plots capability. The results of these simulations were tested (compared) against the experimental data for CFD code validation. For validation purposes, the CFD simulation data were extracted at the same locations as the experimental data were collected.

Preprocessing. The model's geometry, shown in Fig. 5, includes:

- the flow path between the rib and brattice from the velocity inlet (air) to the discharge location at the end of the blowing curtain,
- flow through the interface zone,
- flow in the face area,
- flow return toward the outlet and
- velocity inlet (methane).

Two ventilation arrangements for box cut and slab cut scenarios, with a 10.7-m (35-ft) setback,

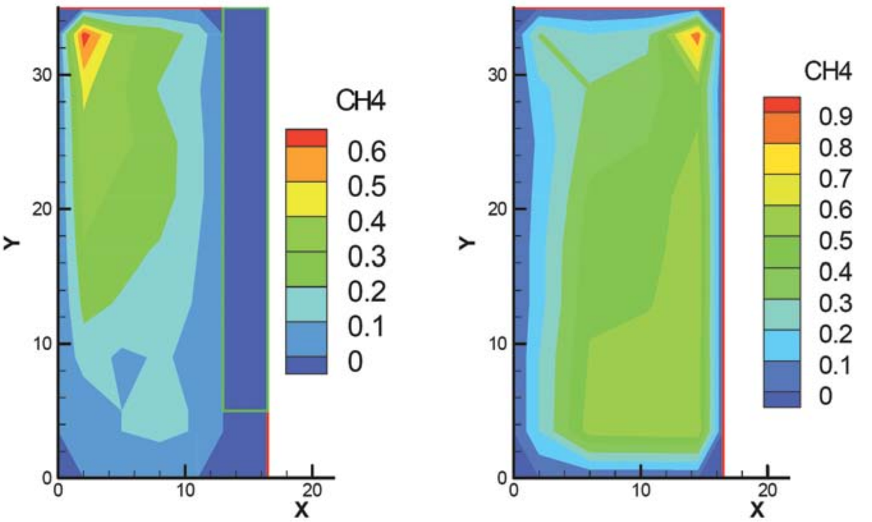
Table 1

Data used to calculate the boundary condition.

				Methane flow			
		Airflow,		Measured		Corrected	
Sl No.	Scenario	m^3/s	(cfm)	m^3/s	(cfm)	m^3/s	(cfm)
1	Box cut	2.8	(6,000)	0.0032	(6.8)	0.0025	(5.27)
2	Box cut	4.7	(10,000)	0.0032	(6.8)	0.0025	(5.27)
3	Slab cut	2.8	(6,000)	0.016	(34.0)	0.016	(34.0)
4	Slab cut	4.7	(10,000)	0.016	(34.0)	0.016	(34.0)

FIGURE 4

Methane concentration: box-cut scenario (left) and slab-cut scenario (right), for 10.7-m (35-ft) setback with 2.8 m³/s (6,000 cfm) airflow.



were considered. The configuration of the model for the box-cut scenario is shown in Fig. 5.

The computational mesh (grid) was generated using the GAMBIT 2.1 mesh generator. In the CFD model, the methane boundary condition represents 192 nozzles that bring the methane into the face area. These nozzles are evenly distributed on the face surface. The most important zone in this study is the area between the end of blowing curtain and the face. To have enough grid resolution in the area of importance, the entire flow region was divided into two zones. First, the zone of the face area and second, the interface zone. The mesh generation for each zone was performed independently. These two zones are connected by an interface boundary condition.

The velocity inlet boundary condition was applied at the air and methane inlet sites. The outlet boundary condition was applied at the outflow site. All the other surfaces are treated as adiabatic walls with a no-slip boundary condition. Table 1 shows the experimental values used to calculate the boundary conditions. Correction was applied to the methane density calculations due to the temperature fluctuations.

The computational mesh of the test box cut configuration is shown in Fig. 6. The mesh with around 1,190,000 cells was accepted based on previously performed studies concerning the grid independence results.

Processing. A three-dimensional, steady state incompressible solution for Navier-Stokes equations with species transport without chemical reactions was performed using FLUENT. FLUENT solves the Reynolds averaged form of Navier-Stokes equations considering the conservation of mass, momentum, energy and species transport. The analysis was performed using different turbulence models to identify the model that can best predict both the flow and the methane distribution. In this study, only the analysis results using two turbulence models, i.e., shear-stress transport and Spalart-Allmaras (SA) models, are discussed. Pressure velocity coupling of momentum and continuity equations is obtained using the SIMPLE algorithm. The outflow boundary condition is applied at the outlet. Buoyancy is introduced into the model by switching on gravity. To adjust for this, the default value for the turbulence Schmidt number is adjusted to be 0.5 instead of 0.7. Further details are discussed in the results section. As mentioned above, the test was carried out in the beginning of the analysis to identify the mesh size that gives grid independence and the same mesh is used for further analysis.

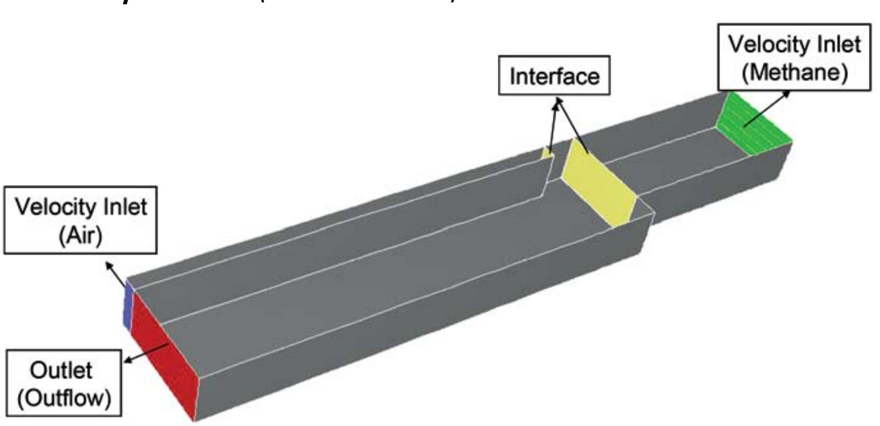
Comparison of the experimental and simulation data

The study involves the analysis of both 2.8 and 4.7 m³/s (6,000 and 10,000 cfm) airflow rates. In this section, the simulation results are compared with the experimental results. Only the results for the 2.8 m³/s (6,000 cfm) airflow rates are shown and discussed in this section. Figure 7 shows the flow pattern for both box and slab cut test scenarios. It can be observed that the flow pattern is highly complicated in the box cut case and is highly three dimensional in nature, whereas in the case of the slab cut case it is less complex. The experimental data provided for velocity vectors are two dimensional in nature, so the authors decided to extract the corresponding two components of velocity from the simulation results for point-to-point comparison. Similarly, the methane concentration at the sample locations is extracted from the simulated results for comparison.

Figure 8 shows the comparison of the flow pattern and methane concentration for the box-cut scenario. It can be seen that the flow separation location predicted by the SA turbulence model is in good agreement with the experimental results. However, the

FIGURE 5

Boundary conditions (box-cut scenario).



magnitude of the velocities is not exactly the same because of the three-dimensional nature of the flow. The methane concentrations predicted by both the turbulence models are also in good agreement with experimental results. In this plot, the maximum methane concentration is observed to be at the left top corner region, which is close to the face. This is the result of the flow pattern observed in the recirculation region. Figures 9a, 9b and 9c show comparisons of the methane concentration at each sample locations at three different heights. It is observed that even though the qualitative comparison is in good agreement with the experimental data, the quantitative comparison shows that the simulation results predict a high concentration at certain locations especially in the area near the separation region. The reasons for this might be one of the following:

- numerical simulation is carried out with a steady-state assumption, whereas the actual flow behavior in this scenario might be unsteady;
- velocity and the methane concentration measurements were not taken simultaneously, which might intensify the error if the flow were unsteady; and
- the location and the way that methane was introduced into the systems were different for the numerical and experimental simulations.

FIGURE 6

Computational mesh: (a) top view and (b) side view of the mesh in the boxed area.

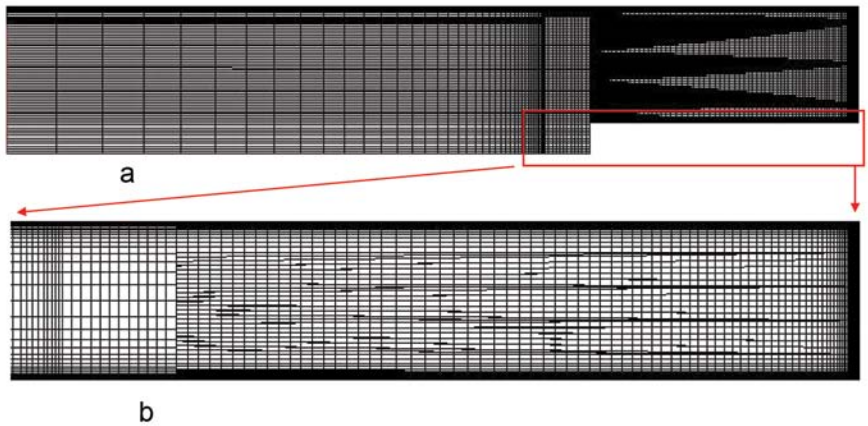


FIGURE 7

Path lines colored by particle numbers: box-cut scenario (left) and slab-cut scenario (right), for 10.7-m (35 -t) setback and 2.8 m^3/s (6,000 cfm) airflow.

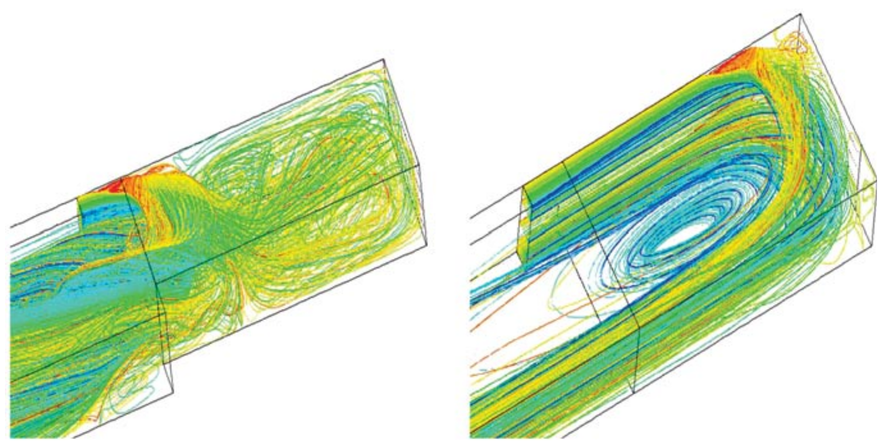


FIGURE 8

Box-cut scenario results comparison.

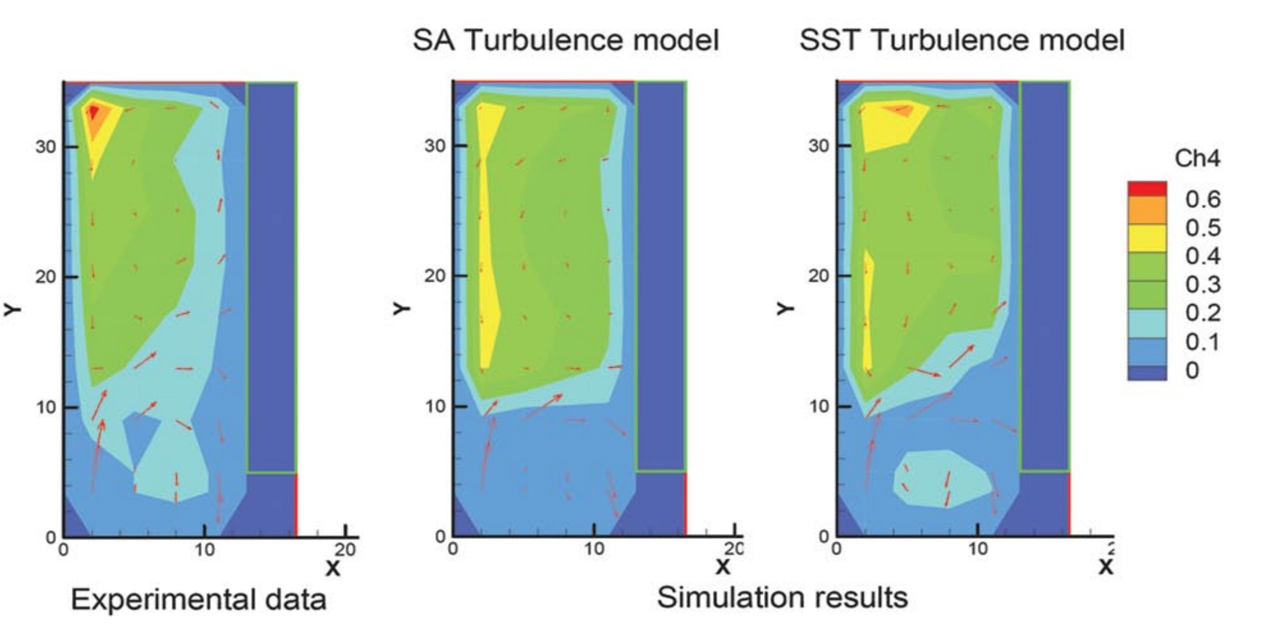


FIGURE 9A

Methane concentration comparison at plane 1 m (3.5 ft) from the ground, for 2.8 m³/s (6,000 cfm) airflow.

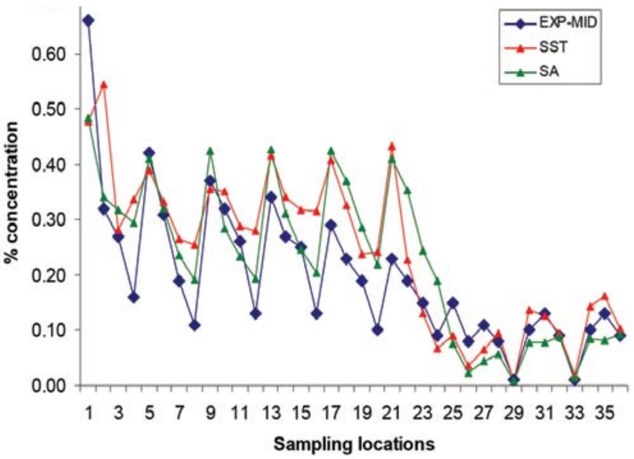


FIGURE 9B

Methane concentration comparison at plane 0.6 m (2 ft) from the ground, for 2.8 m³/s (6,000 cfm) airflow.

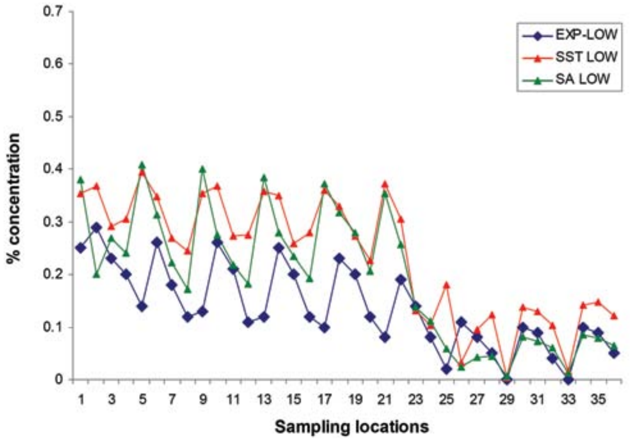
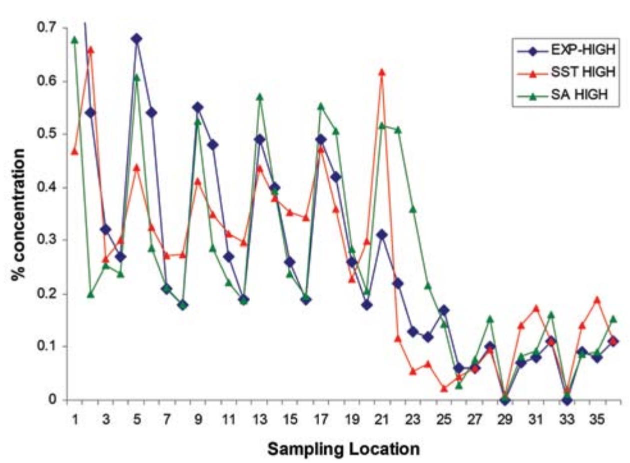


FIGURE 9C

Methane concentration comparison at plane 1.5 m (5 ft) from the ground, for 2.8 m³/s (6,000 cfm) airflow.



For the laboratory experiments, methane was introduced through pipes that are 100 mm (4 in.) away from the wall and the holes on the top and bottom of the pipes, making the methane entry direction normal to the main flow direction. However, in the computer simulation, the methane was introduced normal to the face. The authors noted similar results when comparing data from the 2.8- and 4.7-m³/s (6,000- and 10,000-cfm) tests (not shown in this paper).

Figure 10 shows the comparison of the flow pattern and methane concentration for the slab-cut scenario. It can be seen that the flow patterns predicted by the SA turbulence model are in good agreement with the experimental results with one exception. The exception is that the turbulence model predicted the point of highest methane concentration to be at the left top corner of the face area, whereas in the experiments it was observed at the right top corner. Results from the simulations using the SST turbulence model show that, in the region along the right hand-side wall, there is higher cross-flow in the z-direction than that of the SA of model. This three-dimensional effect may lead to a different concentration profile in the mid-plane of face area. Figures 11a and 11b show comparisons of methane concentration at each sample location for the slab cut scenario and 2.8- and 4.7-m³/s (6,000 or 10,000 cfm) airflow at the end of the blowing curtain. It can be seen that both qualitative and quantitative simulation results are in good agreement with the experimental data, except at the first location.

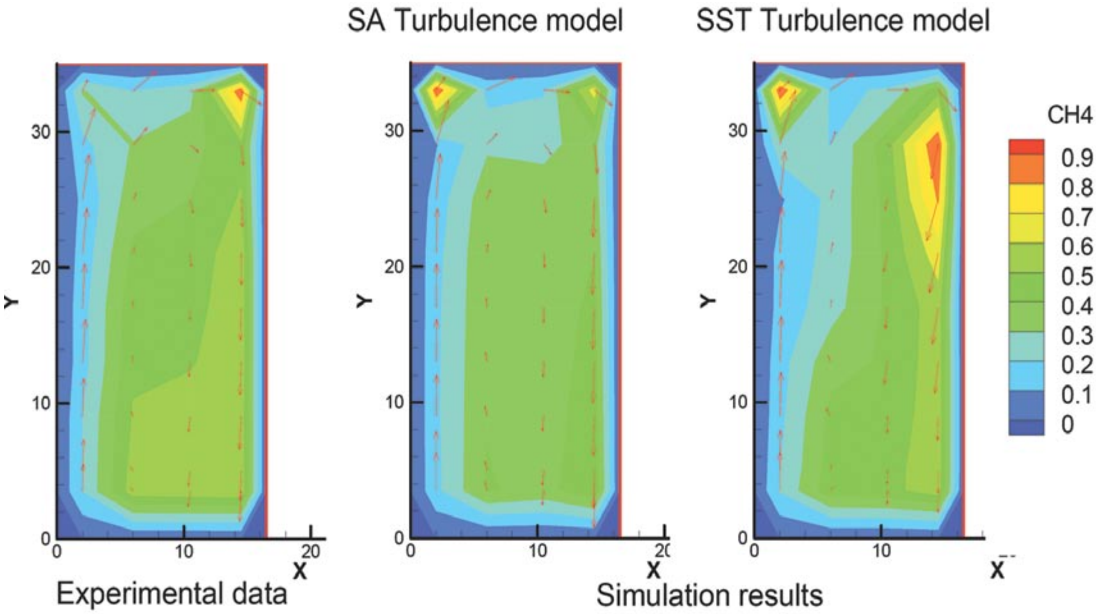
Conclusions

- As far as the authors know, this was the first validation test of the CFD simulation results for both flow and methane concentration in the face area against the full-scale, mine-related benchmark experiments.
- Because this was the first test at this scale, the arrangement at the face area was simplified by removing the equipment at the face to minimize the sources for flow interruption.
- During this study, it was shown that two turbulence models, namely, the Shear-Stress Transport model and the Spalart-Allmaras model, could be used to simulate the three-dimensional methane concentration along with the airflow distributions.
- In the box-cut scenario of the face ventilation system, the simulation results using the SST model show that the flow and the methane concentration are in good agreement with the experimental results.
- In the slab-cut scenario of the face ventilation system, the simulation results using the SA model show that the flow and the methane concentration are in good agreement with the experimental results.
- Based on these studies, it can be seen that there is a lot of potential for the FLUENT CFD software package for developing mine face-ventilation system designs
- It was observed in the experiments that the methane concentrations were oscillating at some frequency. This could be caused by the unsteady flow of methane, which was delivered by the commercial natural gas pipeline, or by air quantity, which was delivered by the gallery fan. To verify the source of these unstable behaviors, the authors suggest repeating one of the laboratory tests. During this test, all the measured (monitored)

parameters, i.e., airflow, amount of methane, air velocities and methane concentrations, at each location of the face area must be measured and recorded simultaneously
- To finally prove that the CFD code is the proper tool for the face-ventilation system analysis and the designs, similar studies with equipment must be performed. ■

FIGURE 10

Comparison of results from simulations using different turbulence model (slab-cut scenario).



References

Taylor, C.D., Timko, R.J., Thimons, E.D., and Mal, T., 2005, "Using ultrasonic anemometers to evaluate factors affecting face ventilation effectiveness," Paper number 05-080, presented at the 2005 SME Annual Meeting, Salt Lake City, UT.

Wala, M.A., Jacob, J., and Turner, D., 2002, "Experimental study of mine face ventilation system for validation of numerical models," Proceedings of the North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, Canada, June.

Wala, M.A., Jacob, J., Huang, P.G., and Brown, J.T., 2002, "A new approach to evaluate mine face ventilation," Paper number 02-177, presented at the 2002 SME Annual Meeting, Phoenix, AZ.

Wala, M.A., Jacob, J., Huang, P.G., and Brown, J.T., 2003, "A new approach to evaluate mine face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J., Huang, P.G., and Rangubhotla, L., 2004, "How scrubbers help ventilate the face during deep cut mining with a blowing curtain," Proceedings of the 10th U.S. Mine Ventilation Symposium, Anchorage, AK, May.

Wala, M.A., Stoltz, J.R., and Hassan, M.I., 2000, "Scaled modeling of a mine face ventilation system for CFD code validation," Proceedings of the Third International Symposium on Scale Modeling, Nagoya, Japan, September.

Wala, M.A., Stoltz, J.R., and Jacob, J., 2001, "Numerical and experimental study of a mine face ventilation system for CFD code validation," Proceedings of the 7th International Mine Ventilation Congress, Krakow, Poland, June.

FIGURE 11A

Methane concentration comparison at plane 1 m (3.5 ft) from the ground, for 2.8 m³/s (6,000 cfm) airflow.combination (Decanter, 2002; Falcon, 2002).

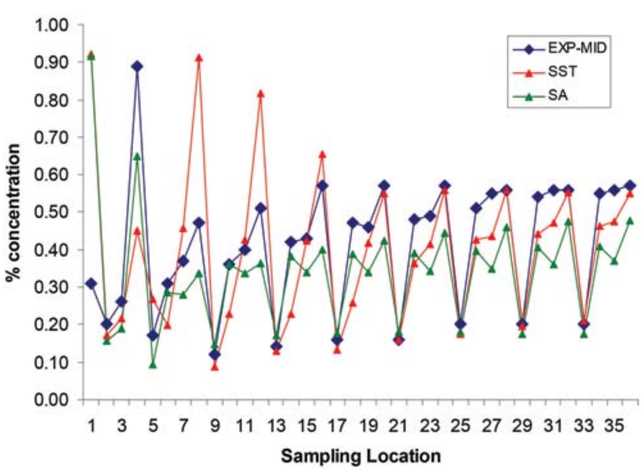


FIGURE 11B

Methane concentration comparison at plane 1 m (3.5 ft) from the ground, for 4.7 m³/s (10,000 cfm) airflow.