Study on the effects of scrubber operation on the face ventilation

A. M. Wala
University of Kentucky, Lexington, KY, USA

S. Vytla & G. Huang
Wright State University, Dayton, OH, USA

C. D. Taylor
NIOSH Pittsburgh Research Laboratory, Pittsburgh, PA, USA

ABSTRACT: For the past four years, the Department of Mining Engineering at the University of Kentucky has conducted a validation study of the CFD Fluent code, by comparing its results against mining-related benchmark experimental data provided by NIOSH's Pittsburgh Research Laboratory. This particular study is dedicated to the effect of machine-mounted dust scrubbers on the performance of face ventilation systems using extended-cut mining with a blowing curtain.

1 Introduction

One of the main concerns during coal extraction, when room and pillar mining method is used with a continuous miner, is the large amount of methane released in the face area. In such circumstances, it is necessary that the methane concentration be checked and maintained at safe levels. There are various ventilation techniques and systems that supply air used to dilute and carry-out accumulated methane (Goodman el at., 1990) to avoid ignition. However, due to the geometry of the face area, especially for a box cutting sequence with a blowing curtain, a very complicated flow patterns occurs (Wala et al., 2003).

During the earlier studies performed by the authors (Wala et al., 2007) in an empty (containing no equipment) face area, it was found that in order to maintain the same level of methane concentration at the face area, the quantity of air needed to ventilate the box cut was approximately five time higher than for the slab cut (Wala et al., 2007). This is caused by flow separation from the wall that results in an unsteady flow behavior, which is predominant during the box cutting mining sequence (Wala et al., 2005). In this case around 70 to 80% of the airflow delivered behind the curtain does not reach the face area. Therefore, due to the greater difficulty of ventilating the face area during the box cut mining sequence, the authors of this paper chose to focus their research on this particular scenario.

As mentioned above, this study was carried out, first to validate the Fluent as a CFD simulation tool, and secondly, to better understand how the use of a mining machine-mounted dust scrubber affects the airflow and methane distribution. These scrubbers, which are primarily used for dust collection, have been shown to be very useful ventilation devices in directing air into the immediate face area where ventilation is poor during the box cut mining sequence.

2 Experimental Studies at NIOSH Laboratory

Two independent studies were conducted at the NIOSH's Pittsburgh Research Laboratory to study the effect of scrubbers on airflow and methane distribution at the mining face. These tests were a combined effort of NIOSH and Department of Mining Engineering at the University of Kentucky because of their mutual interests. The results of the first test were presented during the 11th U.S./North American Mine Ventilation Symposium (Taylor et al., 2006). The results of the second test, performed by NIOSH, have been used for the research study presented in this paper.

2.1 Test Facility

The testing was conducted in the NIOSH's Pittsburgh Research Laboratory's Ventilation Test Gallery, shown in Figure 1. A part of the "L" shaped building is designated to model an underground mining entry, which is 5m (16.5ft) wide by 2.2m (7ft) high. The 1.2m (3.5ft) wide by 12.2m (40ft) long box was built along the right side of the entry to simulate an uncut slab of coal. The resulting entry simulated a 4m (13ft) wide box cut.

The exhaust fan draws approximately 5.9m^3/s (12,500cfm) of air through the gallery. A brattice and wood curtain constructed 0.6m (2ft) from the left side of the entry directed airflow toward the face. The curtain was positioned so that setback distance~~s~~ between the curtain and the face was 10.7m (35ft). Regulator doors were adjusted to provide intake flows behind the curtain of either 1.9 or 2.8m3/s (4,000 or 6,000cfm). A full-scale wooden model of the continuous miner, was located at the center of the entry to simulate a mining machine during the box cutting sequence (see Figure 2). The continuous miner model includes a simulated scrubber system that consists of the following:

- Two inlet openings 0.25m by 0.3m (10 by 14 inches), one on each side of the mining machine, and 2.7m (9ft) from the front edge of the cutting drum.
- An exhaust opening 0.38 by 0.38m (15 by 15 in.) at the right rear of the machine chassis.
- Ducting to provide flow between inlets and exhaust openings.
- An axial fan to move the air from the scrubber inlet opening to the exhaust.

Scrubber airflow is adjusted by inserting a different size ~~of the~~ orifice plate into the ducting. Airflow distribution was not measured during this test because of a space limitation and very complicated 3D flow patterns.

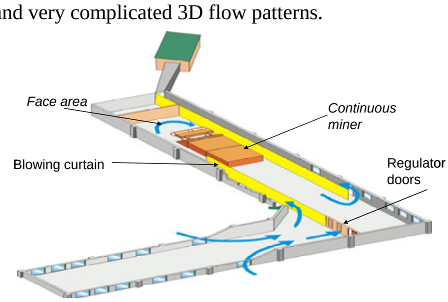

Figure 1. Ventilation test gallery

2.2 Methane Measurement

Methane measurements were made by drawing air samples through eight air sampling tubes that were suspended from an overhead support system. The ends of the upper four tubes were 0.4 (1.3 ft) from the roof while the lower four were 0.7 m (2.4 ft) from the roof. By moving the overhead support system either toward or away from the face, samples were collected for 28 locations at each of the two sampling heights. These 28 sampling locations were arranged in four columns and seven rows as shown in Figure 2. The total number of methane measuring points (upper and lower) was 56, see Figure 2.

To simulate liberation of methane from a mining face methane gas was released from a manifold made of four 3-m (10-ft) long horizontal copper pipes that were located 0.1 m (4 in) away from the face. The pipes were equally spaced horizontally to provide a relatively uniform release of gas. Two millimeters (1/16 in) diameter holes were drilled 6.5cm (2.5 in) apart on the top and bottom of each of the 10-ft long pipes. For the box cut mining sequence the methane flow rate into the gallery was regulated to the level of 0.0063m^3/s (13.4cfm) to prevent methane concentration in face area from exceeding 2.0 percent.

2.3 Experimental Results

The contours in Figures 3 and 4 show the distribution of the methane concentration in the face area for all four scenarios mentioned above. Some general observations can be made by looking at Figure 3 and 4: 1). The ratio between a maximum methane concentration at the methane elevated regions between scenario 1 and 2 is 2.0/0.5 (4.0), and

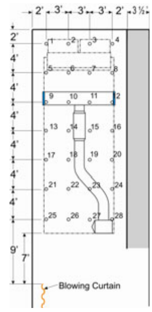

Figure 2: Sampling locations

2.4 Testing Scenario

Methane concentrations readings for the box cut mining sequence were taken for the following four scenarios:

1. Scrubber off, intake flow 1.9m^3/s (4,000ft^3/min)
2. Scrubber off, intake flow 2.8m^3/s (6,000ft^3/min)
3. Scrubber flow 1.9m3/s (4,000ft^3/min), intake flow 1.9m^3/s (4,000ft^3/min)
4. Scrubber flow 1.9m3/s (4,000ft^3/min), intake flow 2.8m^3/s (6,000ft^3/min)

between scenarios 3 and 4 is 2.6/0.5 (5.5). This means that the scrubber improved the ventilation at the face area. However, the remaining question is, how much? 2) Methane concentration along the intake (left-hand) side, between the rib and the miner is lower than on the opposite side of the miner. This means that although the majority of the intake air does not flow along the rib to the mining machine, due to separation from the rib, a small part of the intake air from the curtain does flow between the mining machine and the rib. This flow pattern varies a little from the flow seen during tests in the empty face area 3) Although more air is delivered during test scenario 2 (6,000 cfm) the methane zone is larger and the maximum concentration inside this zone is higher than for scenario 1 with an intake flow of 4.000cfm. These experimental data will be compared with the CFD simulation data later in this paper.

3 Computer Simulation Study

There are three major steps in any CFD solution process: (1) preprocessing (mesh generation), (2) processing (CFD simulation and refinement/adaptation of grid), and (3) post processing (visualization and analysis of results).

FLUENT 6.X, 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 the Fluent and graphically shown using the

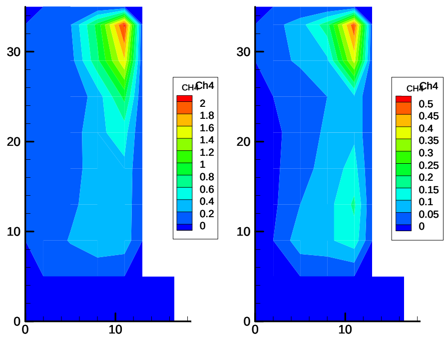

Figure 3: Methane concentration contour maps for intake flow equal 4,000 cfm cases: (left) no scrubber (scenario #1), (right) scrubber flow equal 4,000 cfm (scenario #3)

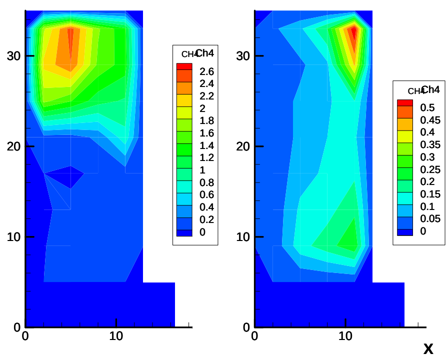

Figure 4: Methane concentration contour maps for intake flow being 6,000 cfm cases: (left) no scrubber (scenario #2, (right) scrubber flow equal 4,000 cfm (scenario #4)

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 the Fluent and graphically shown using the Excel plots capability. The results of these simulations were tested (compared) against the experimental data for CFD code validation. For the validation purposes the CFD simulation data were extracted at the same locations as the experimental data were collected.

Based on experiences gained during the previous CFD studies using fluent code were applied to simulate the 3-D methane concentration along with the air flow distributions.

3.1 Pre Processing

The configuration of the model for the box cutting sequence is shown in Figure 5. The geometry includes: (1) flow path between the rib and brattice from the Velocity Inlet (Air) to the discharge location at the end of the blowing curtain; (2) flow through the Interface zone; (3) flow in the face area; (4) flow return toward the Outlet and (5) Velocity Inlet (Methane). One ventilation arrangements for the box cutting mining sequence, with 35 ft setback blowing curtain was considered.

The computational mesh (grid) was generated using the GAMBIT 2.1., mesh generator. In the CFD model the methane boundary condition at the Velocity Inlet (Methane) represents 192 nozzles which 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. In order 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.

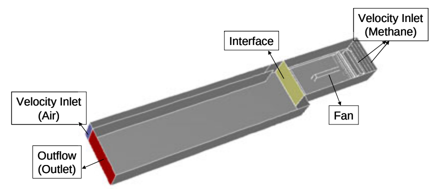

Figure 5: Boundary conditions (box cutting sequence)

The velocity inlet boundary condition was applied at the air and methane inlets. The out-flow boundary condition was applied at the outlet. All the other surfaces are treated as adiabatic walls with no slip boundary condition. Table 1, shows the experimental values used to calculate the boundary conditions. A correction was applied to the flow of methane delivered to the face area because the flow-meter (Rotometer) used during the NIOSH test was calibrated for airflow. The computational mesh of the test box cut configuration is shown in Figure 6. The mesh with around 1,830,000 cells was accepted based on previously performed studies concerning the grid independence results.

Scenario	Intake flow (cfm)	Scrubber flow (cfm)	Methane (cfm)
1	4,000	0	13.4
2	4,000	4,000	13.4
3	6,000	0	13.4
4	6,000	4,000	13.4

Table 1: Data used to calculate the boundary condition

3.2 Processing

A 3D 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,

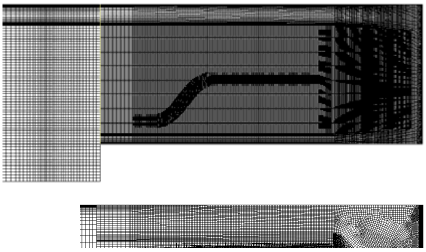

Figure 6: Computational mesh; top view and side view of the mesh

considering the conservation of mass, momentum, energy and species transport. The analysis was performed using different turbulence models to identify the model which can best predict both the flow and the methane distribution. In this study only the analysis results using SST (Shear-Stress Transport) turbulence model 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. Effect of buoyancy is also included by switching on the gravity. Further details will be discussed in the results section.

3.3 Post Processing

Based on the computer simulations, the methane distributions are presented as contour maps and the airflow distribution as flow lines (path lines). The results for all four scenarios are shown in Figures 7 and 8. In general the CFD simulation and experimental study results are similar and show that scrubber helps to ventilate the face area by bringing more air ~~in~~to the face.

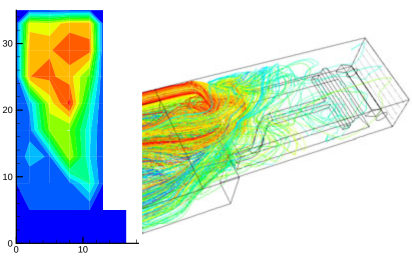

Figure 7(a): Methane concentration contour plots and path lines (colored by particle numbers) for intake flow 4,000 cfm and no scrubber flow, (scenario #1)

4 Comparison of Experimental and Simulation Results

In this section, the experimental results (NIOSH) are compared with the CFD simulation results. NIOSH's experimental results plotted in Figures 3 and 4 show methane concentration contour maps for all four scenarios

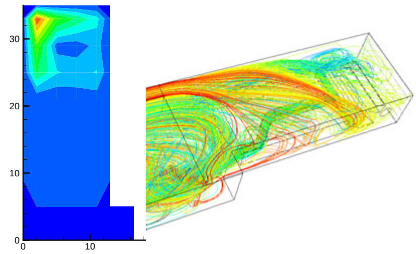

Figure 7(b): Methane concentration contour plots and path lines (colored by particle numbers) for intake flow 4,000 cfm and scrubber flow 4,000 cfm, (scenario #3)

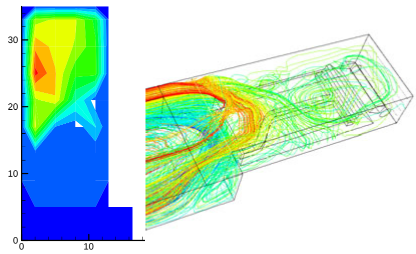

Figure 8(a): Methane concentration contour plots and path lines (colored by particle numbers) for intake flow 6,000 cfm and no scrubber flow, (scenario #2)

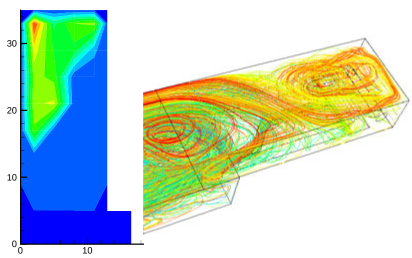

Figure 8(b): Methane concentration contour plots and path lines (colored by particle numbers) for intake flow 6,000 cfm and scrubber flow 4,000cfm (scenario #4)

tested in the NIOSH's ventilation gallery. These results are compared to the methane concentration contour maps shown in Figures 7 and 8, which were generated using data from the CFD simulation. The experimental data and CFD results are compared based on the scenarios given in Table 1.

Scenario #1: For this scenario the amount of air being delivered behind the blowing curtain, for ventilation, is 4,000cfm and continuous miner's scrubber is off. Figures 3 (left part) and 7a show that the maximum methane concentration in the methane elevated zone is around 2.0 percent. The shape and sizes of the methane elevated zones for both cases are different. Figure 3 shows that majority of air delivered to the face is flowing along the left-hand side of the entry, between the entry rib and continuous miner. This makes the methane concentration along this path lower. According to the simulation study the limited amount of air is reaching the face area because the majority of air is separating from wall (rib) and flowing back out of the face area. Even though the flow behavior predicted in the simulations is different to that of experiments it is observed to be consistent with our previous validation study (Wala et. al. 2007). This kind of flow behavior makes the methane elevated zone bigger and concentration higher. To better understand this flow behavior, please see the flow path lines plot, as a part of Figure 7a and Figure 9a.

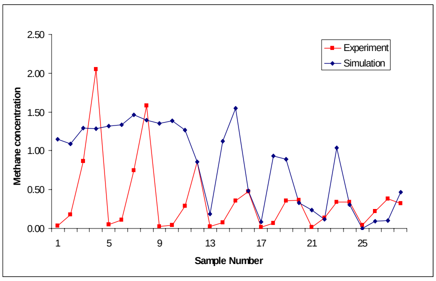

Figure 9(a): Methane concentration comparison 4,000 cfm intake airflow and no scrubber (scenario #1)

Scenario #2: For this scenario the amount of air being delivered behind the blowing curtain is 6,000cfm and continuous miner's scrubber is off. Figures 4(left) and 8a show that the maximum methane concentration in the methane elevated zone is around 2.0 percent. The sizes and the shape of the methane elevated zones for both cases are similar. Figure 4 shows the high methane concentration on the left side this can be explained by the fact that majority of air delivered to the face behind the curtain is separating from wall and flowing back, out of the face area. The rest of the intake air is flowing into the face along the right-hand side. The experimental data and simulated results are similar. The limited amount of air is reaching the face area because the majority of air is separating from wall (rib) and flowing back, out of the face area. This kind of flow behavior makes the methane elevated zone bigger. To better understand this flow behavior, please see the flow path lines plot, as a part of Figure 8a and Figure 10a.

Scenario #3: The difference between this scenario and scenario #1 is that the continuous miner's scrubber is on. Figures 3 (right) and 7b show that the maximum methane concentration in the methane elevated zone is relatively low and is around 0.5 percent. This means that scrubbers have helped improve the ventilation at the face area. The larger amount of air is reaching the face area because of 4,000cfm scrubber being in operation. This kind of flow behavior makes the methane elevated zone smaller. The shape and sizes of the methane elevated zones for both cases are different. Figure 3, right side, shows that majority of air delivered to the face is flowing along the left-hand

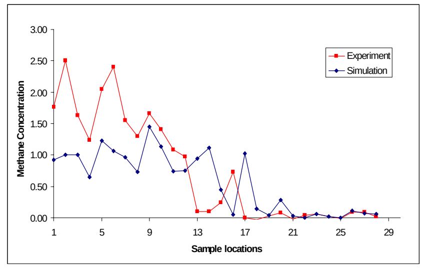

Figure 10(a): Methane concentration comparison 6,000 cfm intake airflow and no scrubber (scenario #2)

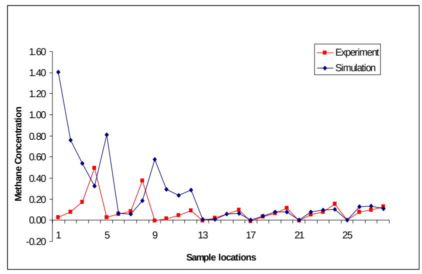

Figure 9(b): Methane concentration comparison 4,000 cfm intake airflow and scrubber flow is 4,000 cfm (scenario #3)

side, between the entry rib and continuous miner. However the simulation results show a different picture. Simulation results predict an increased airflow towards the face area but there still exists a separation and a recirculation region resulting in a different methane concentration in the face area to that of the experiments. With the lack of velocity measurements from experiments for comparison the simulation results are inconclusive. To better understand the flow behavior for this scenario, please see the flow path lines plot, as a part of Figure 7b and Figure 9b.

Scenario #4: In this scenario the amount of intake air delivered at the curtain is 6,000cfm and continuous miner's scrubber is on with flow of 4,000cfm. Figures 4 (right) and 8b show that the maximum methane concentration in the methane elevated zone is around 0.5 percent. This means that scrubbers have improved the ventilation at the face area. The shape and sizes of the methane elevated zones for both cases are different. Figure 3, right-side, shows that majority of air delivered to the face is flowing along the left-hand-side, between the entry rib and continuous miner. However the simulation results show a different picture. The same can be said for Scenario #4 as discussed in Scenario #3 the simulation results were inconclusive with the limited comparison with experimental data. To better understand the flow behavior, please see the flow path lines plot, as a part of Figure 8b and Figure 10b.

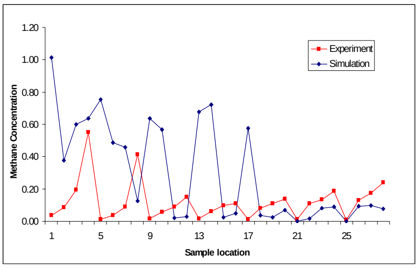

Figure 10(b): Methane concentration comparison 6,000 cfm intake airflow and scrubber flow 4,000 cfm (scenario #4)

5 Conclusions

- Based on these studies there is potential of using Fluent CFD code to develop stand-alone mine face ventilation system design simulation packages.
- Similar type of validation studies for various mining scenarios must be performed in the future in order to build confidence in the use of CFD as a tool for face ventilation analysis and design.
- Such studies had never before been tried to such an extent. By conducting these studies, a vast amount of knowledge is gained about the face mine ventilation process.
- Some differences between the experimental data and simulated data were found. The reasons for these differences must be determine and proved by performing additional tests.

References

Goodman G.V.R, Taylor, C.D. and Divers, E.D., 1990 “Ventilation schemes permit deep advance,” Coal October

Taylor, C.D., Chilton, E.F., Hall, E.E., and Timko, R.J., 2006, “Effect of scrubber on airflow and methane patterns at the mining face,” ”, Proceedings: 11th U.S./North American Mine Ventilation Symposium, University Park, PA, June.

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

Wala, M.A., Vytla, S., Taylor, C.D., and Huang, P.G., 2007, “Mine Face ventilation: a comparison of CFD results against benchmark experiments for the CFD code validation,” Mining Engineering, Vol 59, No 10, October.