

CFD ANALYSIS ON GAS DISTRIBUTION FOR DIFFERENT SCRUBBER REDIRECTION CONFIGURATIONS IN SUMP CUT

Y. Zheng, NIOSH, Pittsburgh, PA, USA
J. A. Organiscak, NIOSH, Pittsburgh, PA, USA
L. Zhou, NIOSH, Pittsburgh, PA, USA
T. W. Beck, NIOSH, Pittsburgh, PA, USA
J. P. Rider, NIOSH, Pittsburgh, PA, USA

ABSTRACT

The National Institute for Occupational Safety and Health (NIOSH), Office of Mine Safety and Health Research (OMSHR) has recently developed a series of models utilizing Computational Fluid Dynamics (CFD) to study gas distribution around a continuous mining machine with various fan-powered flooded bed scrubber discharge configurations. CFD models utilizing Species Transport Model without reactions in FLUENT were constructed to evaluate the redirection of scrubber discharge toward the mining face rather than behind the return curtain. The following scenarios are considered in this study: 100% of the discharge redirected back towards the face on the off-curtain side of the continuous miner; 100% of the discharge redirected back towards the face, but divided equally to both sides of the machine; and 15% of the discharge redirected toward the face on the off-curtain side of the machine, with 85% directed into the return. These models were compared to a model with a conventional scrubber discharge where air is directed away from the face into the return. The CFD models were calibrated and validated based on experimental data and accurately predicted sulfur hexafluoride (SF_6) gas levels at four gas monitoring locations. One additional prediction model was simulated to consider a different scrubber discharge angle for the 100% redirected, equally divided case. These models identified relatively high gassy areas around the continuous miner which may not warrant their use in coal mines with medium to high methane liberation rates. This paper describes the methodology used to develop the CFD models, and the validation of the models based on experimental data.

INTRODUCTION

Methane gas and respirable dust are two hazards frequently encountered at continuous miner (CM) operations. An accumulation of both methane gas and coal dust may lead to massive explosions underground [1], and overexposure to respirable coal dust can result in pneumoconiosis (commonly referred to as "Black Lung Disease") in coal workers [2].

Current U.S. federal coal mine regulations limit concentrations of methane gas to below 1% at the face area and require exposures to respirable coal dust to be at or less than 2.0 mg/m^3 for an 8 h shift average. Begin on August 1, 2016; the overall respirable dust standard in coal mines is reduced from 2.0 to 1.5 mg/m^3 for full shift sampling. If the quartz content of the dust exceeds 5% by weight, the dust standard becomes more stringent, reducing the allowable concentration to the value of 10 divided by the % quartz.

The most effective way to prevent high methane gas and dust concentrations is to deliver sufficient fresh air to the active face. In this region, operating machine-mounted water sprays and fan-powered, flooded bed scrubbers can significantly increase the amount of airflow reaching the face [3-5].

When an entry is large, especially in higher coal seams, the water spray configuration and the scrubber capacity may not be adequate to maintain sufficient forward airflow to confine the dust cloud to the face. Dust can roll back toward the machine operator as a result [6]. Several coal mines having negligible gas emissions in higher coal seams (2.4

to 3.0 m (8 to 10ft.) thick) with exhaust tubing systems redirect the scrubber exhaust back towards the face in an effort to improve dust collection [7]. Since the redirection of scrubber exhaust has not been approved by the Mine Safety and Health Administration (MSHA) for use in thinner and gassier coal seams with ventilation curtains, NIOSH studied the redirected scrubber exhaust techniques for dust and gas control in a full scale CM gallery to assess its potential for industry-wide application.

Previous studies [8-9] have concluded that the behavior of gas and respirable dust in a CM face is a complex process. The distribution/dispersion and transport of gas and respirable dust after they are liberated at the face are governed mainly by their mass property, movement pattern of the ventilation air, and interactions with the mining machine and water droplets created by the external water spray systems used for dust control [10-12]. To understand these behaviors in a complex CM environment and to evaluate the effectiveness of various control techniques, numerical modeling has become a necessary supplement to laboratory experiments and field studies.

CFD simulations have been successfully used in mining research to detect spontaneous combustion and apply inertization in gob areas [13-14], study airflow patterns and gas concentrations in common CM ventilation configurations [15-19], investigate scrubber intake designs for longwall dust control [20], visualize diesel emissions dissipation in underground metal/nonmetal mines [21], and estimate a mine's ventilation status after a disaster [22]. CFD modeling has become a powerful tool for understanding airflow movement and gas/dust behavior in a complicated three-dimensional environment. It can also provide useful information for initial concept testing of new ideas and equipment for gas and dust control.

This study used CFD techniques to reconstruct the experimental findings from prior OMSHR laboratory dust control research. The effects of different scrubber redirection configurations on gas distribution were evaluated for four experimental cases and one prediction case. The simulation procedure and results are revealed and discussed in the paper. Our study extends the knowledge of airflows at the CM face, identifying regions of high gas concentrations to guide future testing and design of control technology. The prediction of airflows for a potential redesign is demonstrated, showing the technique's ability to augment laboratory testing.

METHODS

Laboratory Experiments

Laboratory experiments were previously conducted by OMSHR to test the effect of scrubber redirection on gas and dust distribution using an exhaust curtain ventilation system [7]. The layout of this full-scale CM facility is shown in Figure 1 with an exhaust ventilation curtain extended to within 12.2 m (40 ft.) of the mining face [23]. A 12.2 m (40 ft.) distance represents the largest curtain setback typically encountered while mining extended (deep) cuts [24]. The dimensions of the gallery entry are 5.5 m (18 ft.) wide by 2.1 m (7 ft.) high, with a full-scale mockup of a Joy CM14 continuous mining machine

positioned at a simulated mining face. A 1.2 m (3.9 ft.) wide by 6.1 m (20 ft.) long slab is left on the off-curtain side of the mining face to simulate a block of coal in the slab portion of the cut. Face airflow was measured at the inlet of the curtain (0.91 m (3 ft.) width) with a vane anemometer (moving traverse) at the beginning and end of each experimental run to verify a flow rate of 5.66 m³/s (12,000 cfm).

The full-scale miner was equipped with an on-board scrubber with modified discharge ducts as shown in Figure 1. The scrubber was operated at a flow rate of 3.3 m³/s (7,000 cfm) for various scrubber configurations. The scrubber exhaust discharge configurations examined were: 1) 100% of the air directed outby on the curtain side of the face from scrubber exhaust location 1; 2) 15% of the air redirected to the off-curtain side of the mining machine from exhaust location 3, and 85% of the air directed outby on curtain side of the face from exhaust location 1; 3) 50% of the air directed up each side of the mining machine toward the face, 50% from exhaust location 2 and 3; and 4) 100% of the air directed toward the face on the off-curtain side of the mining machine from exhaust location 3 [7].

External sprays consisted of 15 top-mounted boom sprays (Sprays Bar in View 1 of Figure 1) directed at the top of the rotating drum, three under boom throat sprays (View 2 of Figure 1) directed at the loading pan, and three sprays on each side of the cutter boom directed at the drum's end rings (Figure 1 and View 1 of Figure 1). Spray groups were combined and simplified as rectangular based pyramid with flat top, as indicated by the detailed view of sprays on the right.

Sulfur hexafluoride (SF₆) gas was introduced in front of the rotating cutting drum from two tubes to the left and right side (View 1 of Figure 1). SF₆ gas measurements were recorded at the curtain and off-curtain side of the cutting boom (Gas 1 and Gas 2 as shown in Figure 1 and View 1 of Figure 1). Also, gas measurements were recorded inside the scrubber duct close to the duct junction (Gas 3 in Figure 1) and in the return air course (Gas 4 in Figure 1).

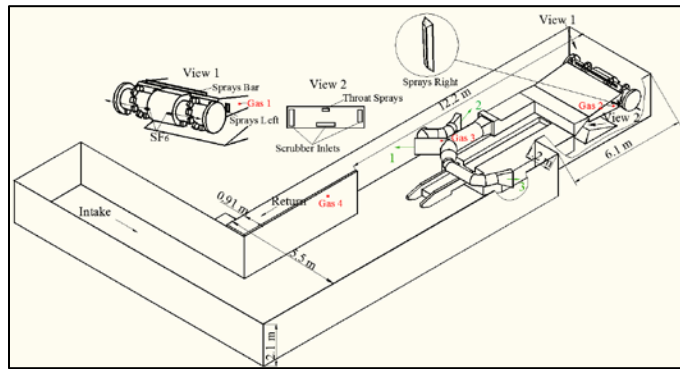


Figure 1. Layout and Dimension of the Full-scale CM Gallery for Scrubber Redirection Experiments.

CFD Modeling

CFD simulations were conducted with ANSYS FLUENT 13.0 (ANSYS Inc, Canonsburg, PA). Species Transport Model without reactions was used to study the gas distribution in the mining face area. Five simulation cases were considered in this paper with steady-state analysis, four of which were based on the laboratory experiment settings; the fifth was based on using modified scrubber discharge angles. All the cases are illustrated in Table 1 below. Geometric model representing the four cases was built according to the design measurements of the full-scale CM gallery. The simulation domain was discretized into about 1.4 million hexahedral and tetrahedral control volumes (cells).

The locations of the boundary conditions are illustrated in Figure 2. Fresh air entering the mining face/simulation domain is depicted by an entry inlet with a fixed air velocity to provide face ventilation. An outlet with zero gauge pressure was placed in behind the exhaust return curtain. The scrubber's airflow was controlled at four locations according to the different simulation scenarios. At the scrubber inlet, a

fan condition was assigned to draw the air into the system. In order to model each case, a wall, an interior plane, or a fan was placed at each of the three scrubber outlets. For example, in the 50/50 scenario, air was discharged evenly from outlet locations 2 and 3. The boundary condition for scrubber exhaust location 1 was a wall to prevent airflow. At outlet location 2, a fan was used to compensate the higher shock loss through its branch, while an interior plane was used to allow airflow through outlet location 3. To model the air-moving effects of the machine-mounted water sprays, a fan was placed at the four specified locations, representing the nozzle-induced airflows. For the gas feeders, a velocity inlet was created with a fixed velocity. All the other boundaries within the domain were defined as interior planes or walls.

Table 1. Simulation Cases with Different Scrubber Redirect Configurations

Experimental cases	Description
1) Conventional	100% airflow discharged from scrubber exhaust location 1
2) 85/15	85% airflow came out from scrubber exhaust location 1 and the other 15% from scrubber exhaust location 3
3) 50/50_23D	50% airflow discharged from scrubber exhaust location 2 and the other 50% airflow from scrubber exhaust location 3, the discharge angle was 23° from the body
4) 100% R	100% airflow discharged from scrubber exhaust location 3
Prediction case	Description
5) 50/50_45D	50% airflow discharged from scrubber exhaust location 2 and the other 50% airflow from scrubber exhaust location 3, the discharge angle was 45° from the body

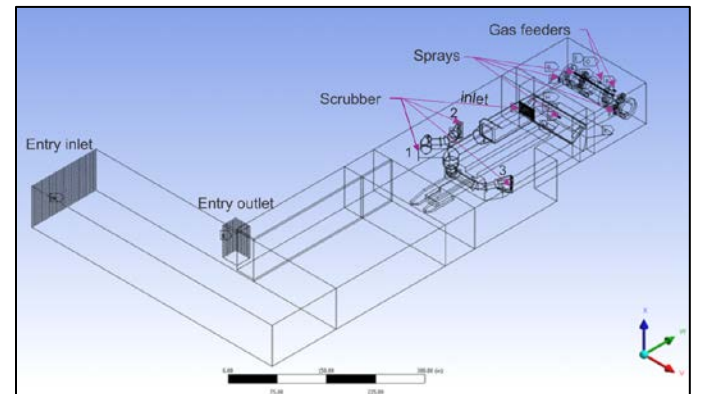


Figure 2. Name and Locations of Boundary Conditions for CFD Models

For the boundary conditions, entry inlet, entry outlet, and velocity speed of gas feeders are known from the experiment. The scrubber can be considered as known by adjusting the boundary conditions at the scrubber inlet and three discharge locations to provide the airflow according to the experimental configuration. The unknown boundaries are the fan condition at the four spray locations and the mass fraction of gas from the gas feeders. These were obtained by iterations of the conventional scrubber discharge case and confirmed for the other three experimental cases.

Of the four experimental cases, the conventional scrubber discharge case was selected to calibrate the model because this configuration results in less complex airflow patterns in the face region. During the calibration process, the turbulence model and solution method were evaluated and the unknown boundary conditions were adjusted to make the gas concentration at the four gas monitor locations agree with the experimental results.

The parameters of the CFD models are listed in Table 2. The same settings, except for the scrubber parameters, were used in all four experimental cases to validate the gas distribution study. The

repeated settings include the entry inlet velocity and outlet pressure, fan condition with the same pressure increase in all four spray locations, and the same gas feeder velocity and mass fraction number. For the scrubber inlet and exhaust outlets, the boundary conditions were adjusted to match the experimental airflow rates at the discharge outlets for different cases.

Table 2. Input Parameters for CFD Models

Description	Parameter
Simulation model	Species transport model without reactions
Turbulence model	k-ε, realizable, enhanced wall condition
Boundary conditions	Entry inlet: velocity inlet (0.98 m/s or 193 fpm)
	Entry outlet: pressure outlet (0 Pa)
	Sprays: fan (40 Pa)
	Scrubber: fan/wall/interior plane (depend on the case and location)
	Gas feeders: velocity inlet (3.86 m/s or 760 fpm), mass fraction of gas 0.0056
Solution method	Others: wall or interior plane
	Pressure-velocity coupling scheme: SIMPLE
	Spatial discretization for gradient: Green-Gauss Node Based; for pressure: PRESTO; others: 2 nd order upwind

For each case, the CFD-modeled airflow patterns were examined for agreement with air currents observed by smoke visualization for the corresponding laboratory experiment. This comparison was used to evaluate the ability of the geometric CFD model to represent the laboratory facility and the appropriateness of the boundary conditions. After the calibration and validation process, the prediction case used the same simulation parameters to study the gas distribution.

RESULTS

Comparison of CFD Modeling to Laboratory Experiments

Table 3 shows that the CFD simulation cases agreed well with the experimental data (within ±15% difference) at the four gas monitoring locations. Also, the airflows revealed by simulation agreed with the visually observed smoke tube flow patterns. Therefore, the simulation results shown below can be used with reasonable accuracy to represent the gas distribution in the CM gallery.

Table 3. Percent Difference (Diff.) between Experiment (Exp.) and CFD Simulation (CFD).

	gas 1	gas 2	gas 3	gas 4
Exp. (conv.) (SF ₆ ppm)	0.75	7.38	2.82	1.51
CFD (conv.) (SF ₆ ppm)	0.71	6.41	2.94	1.40
Diff. (%)	-5.33	-13.14	4.26	-7.28
Exp. (85/15) (SF ₆ ppm)	1.19	7.26	3.23	1.48
CFD (85/15) (SF ₆ ppm)	1.24	6.28	3.62	1.40
Diff. (%)	4.20	-13.50	12.07	-5.41
Exp. (50/50) (SF ₆ ppm)	7.26	10.03	7.74	1.62
CFD (50/50) (SF ₆ ppm)	6.70	9.19	8.14	1.65
Diff. (%)	-7.63	-8.41	5.25	2.12
Exp. (100% R) (SF ₆ ppm)	8.28	4.73	5.85	1.34
CFD (100% R) (SF ₆ ppm)	7.11	4.75	6.26	1.41
Diff. (%)	-14.13	0.42	7.01	5.22

* gas 1, 2, 3, and 4 are the four SF₆ monitors, located as shown in Figure 1.

Conventional Scrubber, 100% Exhaust Discharge. In the conventional case, 100% of scrubber exhaust airflow came from scrubber outlet 1 (Figure 1), which is the normal discharge pattern of the scrubber. The fresh airflow pattern shown in Figure 3 revealed that as the air flowed toward the face, part of the airflow went directly to the return, while a portion was drawn toward the scrubber inlets. Most of the fresh air that flowed toward the face went to the left side (curtain side) of the entry through the left scrubber inlet. The right side of the entry was obstructed by the simulated block of coal in the slab portion of the cut. This fresh airflow pattern reduced the amount of gas in the left side of the face area (0.75 ppm from the experiment, 0.71 ppm from CFD) as shown in Figure 4. The gas accumulation was

approximately 10 times higher on the right side of the face area (off-curtain side, 7.38 ppm experiment, and 6.41 ppm CFD). The air and gas mixture drawn into the scrubber was discharged from exhaust location 1 and impacted the wall close to the curtain creating turbulence in that area.

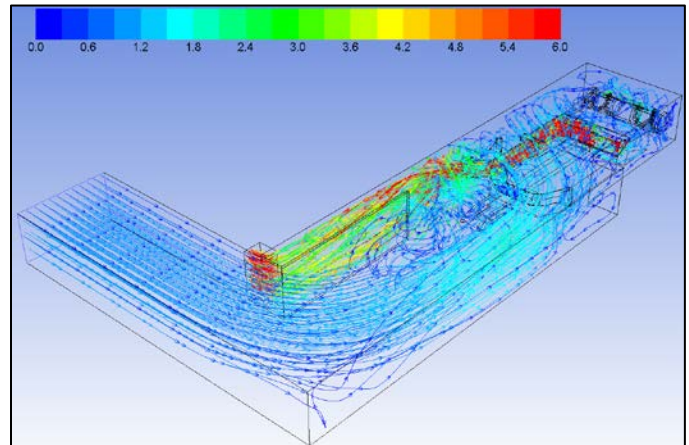


Figure 3. Pathline of Fresh Airflow (conventional), Different Colored Arrow Lines Representing the Velocity (0.0-6.0 m/s) of Different Particles Released from Entry Inlet.

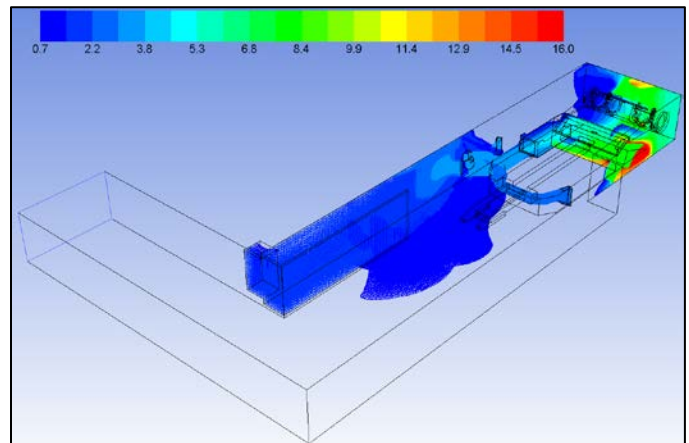


Figure 4. Gas Concentrations for Simulation of Conventional Scrubber Discharge, Gas Level from 0.7 to 16.0 ppm.

85% Conventional Exhaust Discharge with 15% Scrubber Redirection. In the 85/15 case, 85% of airflow (6,000 cfm) was exhausted from scrubber outlet 1 and the other 15% (1,000 cfm) from outlet 3 (off-curtain side). Compared to the conventional scrubber flow, the 85/15 configuration induced more airflow to the right side of the face area as shown in Figure 5. The gas concentration near the front right corner of the machine (off-curtain side) was reduced by this arrangement as shown in Figure 6. However, the airflow from the curtain and off-curtain side of the machine flowed in opposite directions near the face region—a scenario that indicated recirculation in the central face region. As a result, it can be observed from Figure 6 that the gas level could be relatively high both above and in front of the drum.

When compared to the conventional case, this exhaust redirect configuration may result in higher gas levels in the central region above the cutting drum, where frictional ignition sources are known to exist.

50% Redirection to Curtain Side and 50% to Off-curtain Side. In this 100% scrubber redirection case, with 50/50 distribution of the scrubber discharge, 50% of airflow was discharged from scrubber outlet 2, and 50% from scrubber outlet 3. The discharge angles of both outlets were approximately 23 degrees away from the body of the CM.

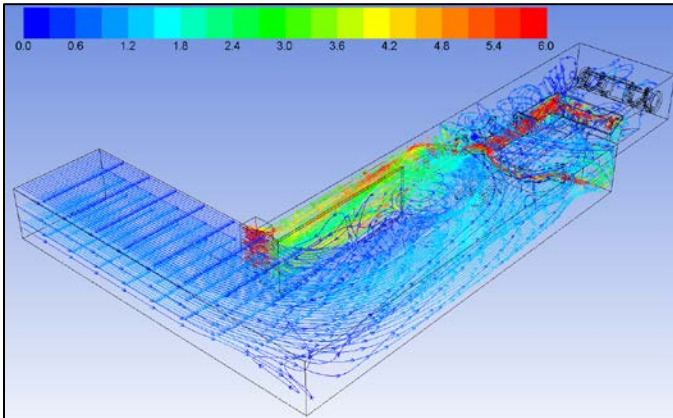


Figure 5. Pathline of Fresh Airflow (85/15), Different Colored Arrow Lines Representing the Velocity (0.0-6.0 m/s) of Different Particles Released from Entry Inlet

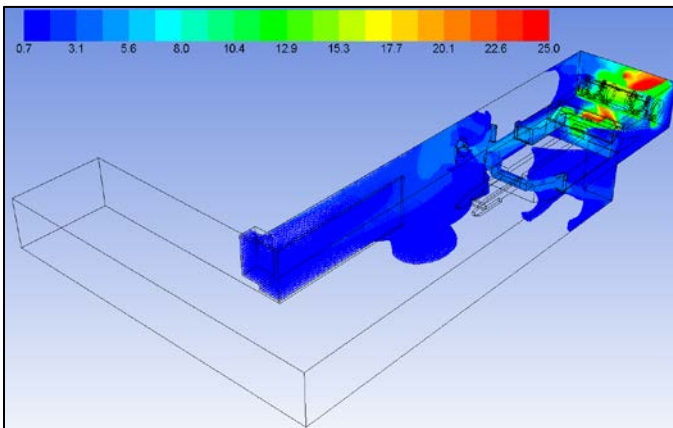


Figure 6. Gas Concentrations for Simulation of 15% Scrubber Redirection and 85% Conventional Scrubber Discharge, Gas level from 0.7 to 25.0 ppm

The outlet area is 0.093 m^2 (1 ft^2) for both the left and right scrubber outlets, with an average airflow speed of 17.78 m/s ($3,500 \text{ fpm}$) at the outlet opening. This configuration created a large area of turbulence in the front of the CM, as the airflow discharged from the left outlet caused the air to swirl toward the off-curtain side of the entry in the face area. Due to the obstruction caused by the block of coal in the slab portion of the cut, only a small part of the air was directed from the right scrubber outlet toward the face. The airflow from the left scrubber outlet dominated the ventilation of the face area as shown in Figure 7.

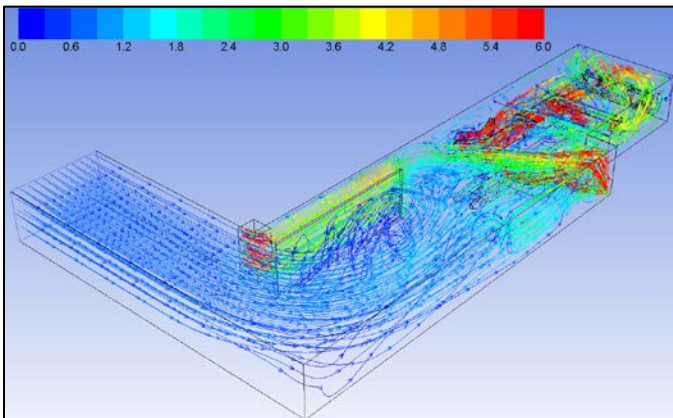


Figure 7. Pathline of Fresh Airflow (50/50), Different Colored Arrow Lines Representing the Velocity (0.0-6.0 m/s) of Different Particles Released from Entry Inlet

In the rear of the CM, airflow from the right scrubber outlet hit the slab, right wall, and floor. Most of the airflow went directly toward the return at the roof level. Part of the fresh air from the entry inlet flowed toward the face close to the floor level while a portion of fresh air was directed toward the return without ventilating the face (Figure 7).

Recirculation in the face area, caused mostly by the left scrubber outlet discharge, resulted in higher and more uniform gas levels in the face region compared to the previous two simulation cases as shown in Figure 8. The highest gas level was observed on the off-curtain corner due to less airflow reaching the face from the right scrubber outlet discharge. The CFD drawing shows that the scrubber reused some of the airflow from its exhaust, reducing the amount of fresh air reaching the face.

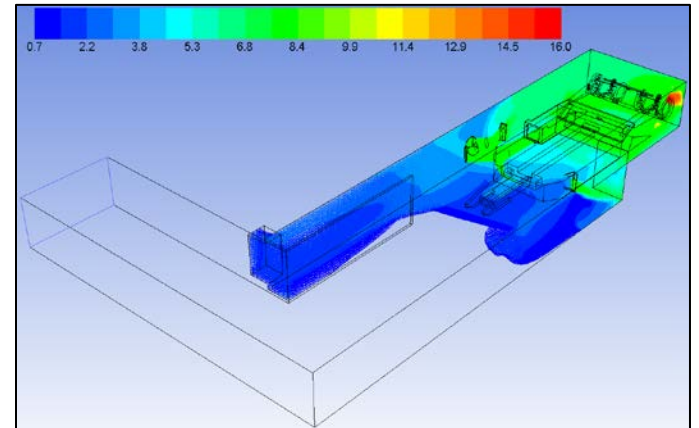


Figure 8. Gas Concentrations for Simulation of Evenly Split 100% Scrubber Redirection, Gas Level from 0.7 to 16.0 ppm

100% Redirection to Off-curtain Side . In this 100% scrubber redirection case, all of the scrubber discharge was from the right outlet (off-curtain side outlet 3) with a velocity of 35.56 m/s ($7,000 \text{ fpm}$).

It can be observed from Figure 9 that airflow from the right scrubber outlet hit the slab, right wall, and floor. Most of the exhaust flow traveled toward the return at the roof level and some of it flowed toward face after hitting the left wall, while a quantity of the fresh air from the entry inlet went toward the face underneath that flow. A portion of the discharge from the right scrubber outlet combined with the fresh air traveling toward the face, and flowed from the off-curtain to curtain side of the entry.

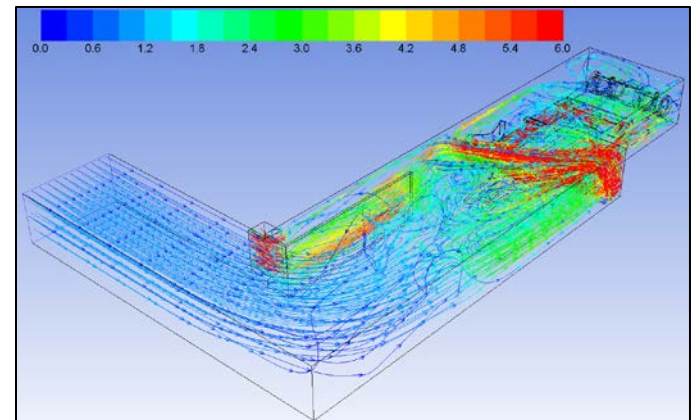


Figure 9. Pathline of Fresh Airflow (100% R), Different Colored Arrow Lines Representing the Velocity (0.0-6.0 m/s) of Different Particles Released from Entry Inlet

Since the airflow in the face area flowed from right to left, gas was pushed toward the front left side of the face area as shown in Figure 10. Overall, this redirected configuration recirculated the scrubber exhaust resulting in gas levels that were uniformly higher than the conventional and 85/15 simulation scenarios. Similar to the 50/50 case

above, the scrubber reused some of the airflow from its exhaust, reducing the amount of fresh air reaching the face.

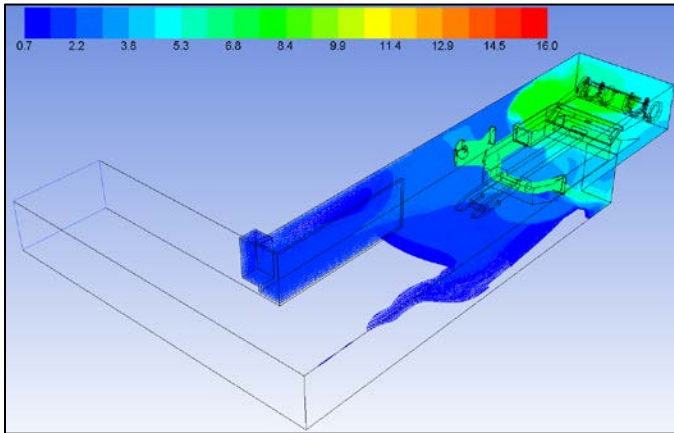


Figure 10. Gas Concentrations for Simulation of 100% Redirected Scrubber Discharge to Off-curtain Side, Gas Level from 0.7 to 16.0 ppm

Prediction of Face Conditions Using CFD

By calibrating and validating the CFD models, it is possible to predict the gas distributions for some interesting simulation cases that may not be practical for laboratory experimentation. For example, an underground U.S. mine with 10 ft. height (3.0 m) redirects the discharge from the scrubber, as in the 50/50 experimental case, but at an angle of 45 degrees from the body of the CM. Though this condition was not evaluated in the previous laboratory experiments, CFD may be used to predict the possible outcome when implemented in a 7 ft. (2.1 m) coal seam face. These results can be further compared to those obtained in the prior 50/50 CFD simulation with scrubber outlets angled at 23 degrees to identify regions of interest and estimate any possible changes in gas concentrations.

Figure 11 shows the 45 degree scrubber outlets for this 50/50 prediction case and the 23 degree outlets for the prior 50/50 simulation. Due to the confined space, the left scrubber outlet had to be shortened. At the larger angle of 45 degrees, airflow directed toward the face from the right scrubber outlet was totally blocked by the right wall and slab (Figure 12A). The airflow from the left scrubber outlet dominated the ventilation of the face area and the airflow swept the face from left to right. As shown in Figure 13A, this flow pattern pushed the gas toward the off-curtain side of the entry. For the 23 degree simulation case shown in Figure 12B, a portion of the airflow from the right scrubber outlet traveled toward the face and ventilated the off-curtain corner. As a result, Figure 13A shows that gas levels in the right corner of the face area were higher and covered a larger area in 45 degree prediction case than in the 23 degree case in Figure 13B.

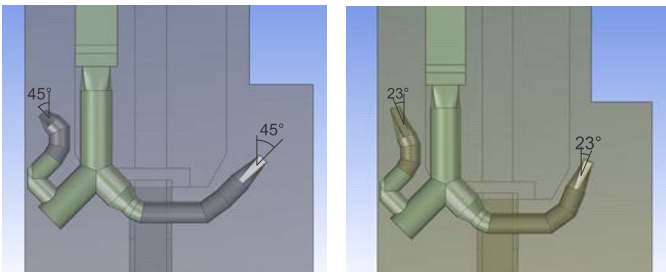
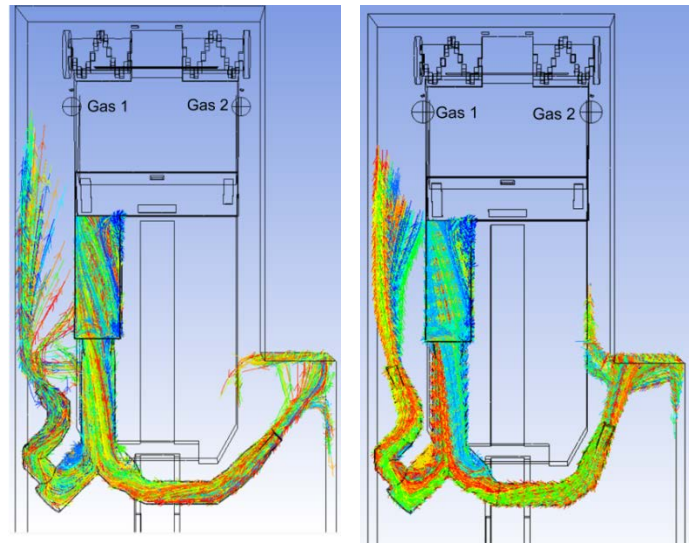


Figure 11. Angle and Length Change of Scrubber Outlets.

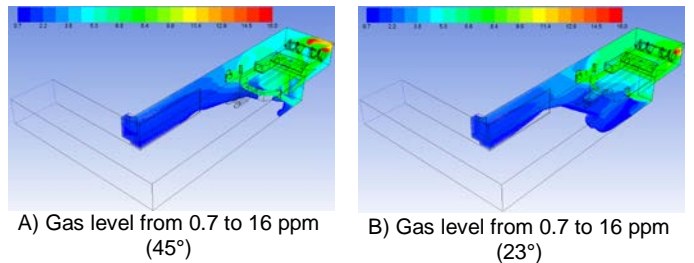
The gas concentrations for all four sampling locations are listed in Table 4. In the table, the simulated gas levels for 45 degree case are listed on the second column. The next two columns list the simulated gas concentrations for the 23 degree discharge angles and the difference between the 45 degree and 23 degree simulations for each sampling location. The simulated gas concentration at location 2 is

predicted to be nearly 40 percent higher when using the 45 degree scrubber outlets.



A) Pathline of airflow (45°), different colored arrow lines representing the course of different particles released from middle of scrubber
B) Pathline of airflow (23°), different colored arrow lines representing the course of different particles released from middle of scrubber

Figure 12. Comparison of Pathline and Gas Concentration for 45° and 23° Outlets (50/50).



A) Gas level from 0.7 to 16 ppm (45°)
B) Gas level from 0.7 to 16 ppm (23°)

Figure 13. Comparison of Gas Concentration for 45° and 23° Outlets (50/50).

Table 4. Prediction of Gas Level for 50/50_45D and Comparison with 50/50_23D.

Gas (ppm)	50 / 50 (45°) CFD	50 / 50 (23°) CFD	Diff. (%)
gas 1	5.56	6.70	-17.01
gas 2	12.91	9.19	40.48
gas 3	7.1	8.14	-12.78
gas 4	1.45	1.65	-12.12

DISCUSSION

A set of cross-sectional planes (sweep surfaces) were built to examine the gas levels on various sections of the simulation domain. Five of the sweep surfaces are illustrated in Figure 14. They divide the domain into five different regions which include: front-most region, between face and gas feeder (0.0 – 0.4 m); drum region, between gas feeder and end of drum (0.4 – 1.4 m); cutter boom region, between end of drum and scrubber inlet (1.4 – 3.0 m); scrubber region, between scrubber inlet and end of scrubber (3.0 – 8.9 m); and remote control region, between end of scrubber and gas monitor in return (8.9 – 14.4 m).

Figure 15 illustrates the highest gas level in the different cross-sectional planes for the five simulation cases within these regions. The vertical axis shows the gas concentration in parts per million (ppm). The horizontal axis is the distance between the sweep surfaces and the face, with zero being the location of the face.

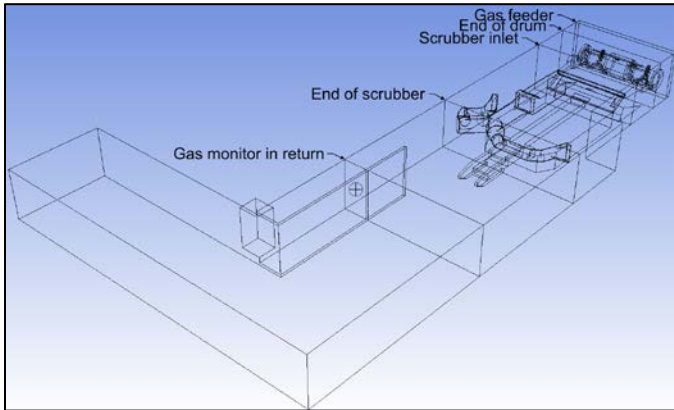


Figure 14. Representing Sweep Surfaces in the Simulation Domain.

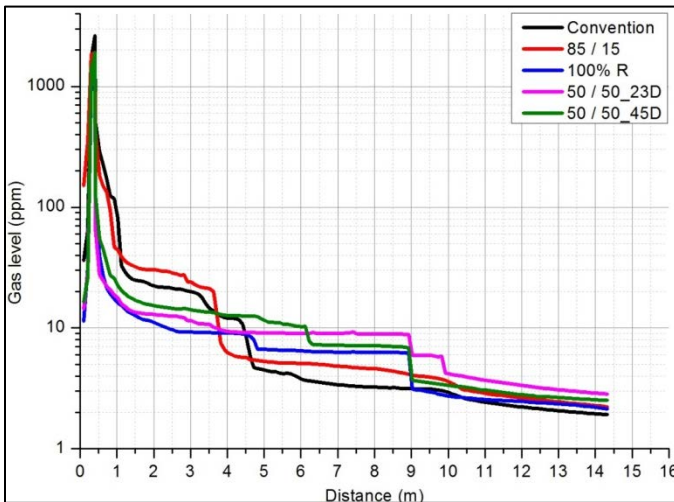


Figure 15. Highest Gas Level from Each Sweep Surfaces at the CM Gallery.

It can be observed from Figure 15 that the gas level is the highest at the gas feeder plane (about 0.4 m in horizontal axes), ranging from 1600 to about 3000 ppm. Although the gas mass fractions applied at the gas feeder are the same for different simulation cases, the different values indicate how well the ventilation diluted the plane. The conventional airflow case had the highest gas level, indicating the poorest ventilation in this particular plane. However, the actual gas levels of this region may not be predicted accurately by the current study since the rotation of the cutting drum was disregarded in the simulation. The rotation of the drum may help mix the gas and air more uniformly in the face area, which could lower gas levels.

From 0.0 to 0.4 m is the front-most region; the worst case is the 85/15 simulation with a gas level above 100 ppm at 0.1 m plane and above 300 ppm at 0.2 m plane. The second worst case is the conventional simulation, while the other three simulations are similar.

From 0.4 to about 1.4 m is the drum region; the ventilation in this area for the conventional and 85/15 simulation cases was worse than the other three simulations. The same trend can be observed in the cutter boom region (from 1.4 to 3.0 m).

From 3.0 to 8.9 m is the scrubber region with the scrubber inlets located from 3.0 to 3.6 m. It can be observed that the gas levels in the 85/15 simulation dropped sharply after the end of the scrubber inlet. As shown in Figure 6, it can be seen that the entire high level gas plume was drawn into the scrubber inlet. After that, the gas was mixed in the scrubber duct and gradually dropped to about 4 ppm at the scrubber outlets. For the conventional case, a portion of the high level gas plume missed the scrubber inlets on the off-curtain side and was pushed back to about 4.7 m outby the gas feeder. The other three simulations redirected all exhaust airflow toward the face. This caused

recirculation of gas in the face area, and as a result the gas levels were relatively more uniform and higher compared to the conventional and 85/15 simulations.

From 8.9 to 14.4 m is the remote control region. All five simulations had low gas levels and dropped gradually as the exhaust flowed toward the entry outlet.

CONCLUSIONS

CFD simulations have proven to be a valuable tool in understanding air and gas flow patterns on a CM face and in studying the effect of various scrubber redirection configurations on gas distribution and on detecting potential high gas level regions.

Based on current experimental settings, simulation results showed that for scrubber redirection on a 7 ft. (2.1 m) coal seam CM face, the 50/50 and 100% R configurations produced uniformly higher gas levels in the face area when compared to the conventional and 85/15 partial redirect configurations. However, the conventional and 85/15 arrangements had a tendency to produce gas hot spots with higher relative gas concentrations in the front-most, drum and cutter boom regions. Based on the results seen with the tested scenarios, the presence of high gas regions could not warrant the usage of redirected scrubber configurations towards the face in coal mines with medium to high methane liberation rates. These simulations predicted the gas distribution with reasonable accuracy and indicate that it may be beneficial to use the same geometric model to evaluate dust levels during scrubber redirection in low/medium coal seam mines. The CFD model and boundary conditions obtained in the study may be further applied to evaluations of future dust and gas control technologies involving water sprays and other airflow devices, ventilation plans, or work practices.

ACKNOWLEDGEMENTS

The authors of this paper sincerely acknowledge Jay F. Colinet, J. Drew Potts, William Reed, and Liming Yuan for their technical support. The authors also thank Jason S. Driscoll and Milan Yekich for their help with experiment and facility measurement.

DISCLOSURE

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.

REFERENCES

1. Dubinski, J., Krause, E., and Skiba, J. (2011), "Global Technical and Environmental Problems Connected with the Coal Mine Methane", *Proceedings of the 22nd World Mining Congress and Expo 2011*, September 11-16, Istanbul, Turkey.
2. NIOSH (2008), "Work-related Lung Disease Surveillance Report, 2007", U.S. Department of Health and Human Services, Centers for Disease Control and Prevention, National Institute for Occupational Safety and Health, DHHS (NIOSH) Publication No. 2008-143a, 378 pp, <http://www.cdc.gov/niosh/docs/2008-143/pdfs/2008-143.pdf>.
3. Taylor, C.D., Chilton J.E., Hall, E., and Timko, R.J. (2006), "Effect of Scrubber Operation on Airflow and Methane Patterns at the Mining Face", *Proceedings of the 11th U.S./North American Mine Ventilation Symposium*, June 5-7, University Park, PA, pp. 393-399.
4. Taylor, C.D., Chilton, J.E., and Goodman, G.V.R. (2010), "Guidelines for the Control and Monitoring of Methane Gas on Continuous Mining Operations", U.S. Department of Health and Human Services, Centers for Disease Control and Prevention, National Institute for Occupational Safety and Health, DHHS (NIOSH), publication No. 2010-141, Information Circular 9523, 75 pp, <http://www.cdc.gov/niosh/mining/UserFiles/works/pdfs/2010-141.pdf>.

5. Colinet, J.F., Rider, J.P., Listak, J.M., Organiscak, J.A., and Wolfe, A.L. (2010), "Best Practices for Dust Control in Coal Mining", U.S. Department of Health and Human Services, Centers for Disease Control and Prevention, National Institute for Occupational Safety and Health, DHHS (NIOSH) Publication No. 2010-110, Information Circular 9517, 76 pp, <http://www.cdc.gov/niosh/mining/UserFiles/works/pdfs/2010-110.pdf>.
6. Kissell, F.N. (2003), "Handbook for Dust Control in Mining", Pittsburgh, PA: U.S. Department of Health and Human Services, Centers for Disease Control and Prevention, National Institute for Occupational Safety and Health, DHHS (NIOSH) publication No. 2003-147, Information Circular 9465, 131 pp, <http://www.cdc.gov/niosh/mining/UserFiles/works/pdfs/2003-147.pdf>.
7. Organiscak, J. and Beck, T. (2013), "Examination of Redirected Continuous Miner Scrubber Discharge Configurations for Exhaust Face Ventilation Systems", *Transactions of the Society for Mining, Metallurgy and Exploration*, Vol. 332, No. 1, pp. 435-443.
8. Goodman, G.V.R., Taylor, C.D., Colinet, J.F., and Thimons, E.D. (2001), "Research by NIOSH for Controlling Respirable Dust and Methane Gas on Continuous Miner Faces", *Proceedings of the 7th International Mine Ventilation Congress*, June 17-22, Krakow, Poland. pp. 237-241.
9. Goodman, G.V.R., Beck, T.W., and Pollock, D.E. (2006), "The Effects of Water Spray Placement for Controlling Respirable Dust and Face Methane Concentrations", *Journal of the Mine Ventilation Society of South Africa*, Vol. 20, No. 2, pp. 56-63.
10. Jayaraman, N.I., Kissell, F.N., and Schroeder, W. (1984), "Modify spray heads to reduce dust rollback on miners", *Coal Age*, Vol. 89, No. 6, pp.56-57.
11. Pollock, D. and Organiscak, J. (2007), "Airborne Dust Capture and Induced Airflow of Various Spray Nozzle Designs", *Aerosol Science and Technology*, Vol. 41, No.7, pp. 711-720.
12. Chilton, J.E., Taylor C.D., Hall, E.E., and Timko, R.J. (2006), "Effect of Water Sprays on Airflow Movement and Methane Dilution at the Working Face", *Proceedings of the 11th U.S./North American Mine Ventilation Symposium*, June 5-7, University Park, PA, pp. 401-406.
13. Yuan, L. and Smith, A.C. (2007), "Computational Fluid Dynamics Modeling of Spontaneous Heating in Longwall Gob Areas", *Transactions of the Society for Mining, Metallurgy and Exploration*, Vol. 322, pp. 37-44.
14. Ren, T.X., Balusu, R., and Humphries, P. (2005), "Development of Innovative Goaf Inertisation Practices to Improve Coal Mine Safety", Aziz, N(ed), *Coal 2005: Coal Operators' Conference*, University of Wollongong and the Australiasian Institute of Mining and Metallurgy, pp. 315-322.
15. Sullivan, P. and Heerden, J.V. (1993), "The Simulation of Environmental Conditions in Continuous Miner Developments Using Computational Fluid Dynamics", *Journal of the Mine Ventilation Society of South Africa*, January, pp. 2-11.
16. Hargreaves, D.M. and Lowndes, I.S. (2007), "The Computational Modeling of the Ventilation Flows within a Rapid Development Drivage", *Tunnelling and Underground Space Technology*, Vol. 22, No. 2, pp. 150-160.
17. Wala, A.M., Vytla, S., Huang, G., and Taylor, C.D. (2007), "Mine Face Ventilation: a Comparison against Benchmark Experiments for the CFD Code Validation", *Mining Engineering*, Vol. 59, pp. 49-55.
18. Kollipara, V.K., Chugh, Y. P., and Relangi, D.D. (2012), "A CFD Analysis of Airflow Patterns in Face Area for Continuous Miner Making a Right Turn Cut", *Proceedings of the SME annual meeting*, February 19-22, Seattle, WA, preprint, pp. 12-132.
19. Torno, S., Toraño, J., Ulecia, M., and Allende, C. (2013), "Conventional and Numerical Models of Blasting Gas Behavior in Auxiliary Ventilation of Mining Headings", *Tunnelling and Underground Space Technology*, Vol. 34, pp.73-81.
20. Ren, T.X. and Balusu, R. (2008), "Innovative CFD Modeling to Improve Dust Control in Longwalls", Aziz, N(ed), *Coal 2008: Coal Operators' Conference*, University of Wollongong and the Australiasian Institute of Mining and Metallurgy, pp. 137-142.
21. Zheng, Y. (2011), "Diesel Particulate Matter Dispersion Analysis in Underground Metal/Non-metal Mines Sing Computational Fluid Dynamics," *PhD Dissertation*, Missouri University of Science and Technology, Rolla, MO., 261 pp.
22. Xu, G., Luxbacher, K.D., Ragab, S., and Schafrik, S. (2013), "Development of a Remote Analysis Method for Underground Ventilation Systems Using Tracer Gas and CFD in a Simplified Laboratory Apparatus", *Tunnelling and Underground Space Technology*, Vol. 33, pp. 1-11.
23. OMSHR (2013), "Full Scale Continuous Miner Dust Gallery", Office of Mine Safety and Health Research Facilities in Pittsburgh, PA, <http://www.cdc.gov/niosh/mining/content/facilities.html>.
24. MSHA (2012), "Re-issue of 110-V-09 – Procedures for Evaluation of Requests to Make Extended Cuts with Remote Controlled Continuous Mining Machines", U.S. Department of Labor, Mine Safety and Health Administration, Procedure Instruction Letter No. 112-V-11, 11 pp, <http://www.msha.gov/regs/complian/PILS/2012/PIL12-V-11.pdf>.