DEPARTMENT OF HEALTH & HUMAN SERVICES

Public Health Service
Centers for Disease Control
and Prevention (CDC)

Memorandum

Date: February 22, 2005

NOT TO NTIS. Contains copyrighted material.

To: William D. Bennett
Data Systems Team, Information Resources Branch, EID, NIOSH, P03/C18

From: Susan Board
Office of Extramural Programs, NIOSH, E74

Subject: Final Report Submitted for Entry into NTIS, Grant Number: 5R01CCR415822-03

The attached final report has been received from the principal investigator on the subject NIOSH grant. When the document is forwarded to the National Technical Information Service, please inform us of the document number. This will allow us to respond to inquiries received about the final report. Publications are highlighted on the attached final report summary.

Attachments:
- Final Report
- Summary

cc: Sherri Diana, EID, P03/C18

Title: CFD Analysis of Mine Face Ventilation Systems
Investigator: Dr. Andrzej Wala
Affiliation: University of Kentucky
City & State: Lexington,
Telephone: 6062572959
Award Number: 5R01CC415822-03
Start & End Dates: 7/1/1999-6/30/2003
Total Project Funding: $399,682
Program Area : Control Technology & Personal Protective Equipment

Final Report Abstract:
The use of extended (deep) cut mining with remotely controlled continuous miners is commonly utilized across 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 that higher levels of methane are liberated at the lace during coal extraction. This requires an increase in the air supply to the face of the heading for adequate methane dilution. Higher air velocities are a consequence, which, in turn, lead 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 recalculate and filter air in the immediate face area. These circumstances give rise to many concerns concerning miners' health and safety.
The former u.s. Bureau of Mines (currently NIOSH), Mine Safety and Health Administration, 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 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 potential to generate face ventilation designs without the previously mentioned disadvantages.
The major objective of this proposed project was to investigate the feasibility of developing a tool that facilitates the design of face ventilation systems to provide healthier and safer working environments for underground miners. The potential ability of this approach to reliably, quickly, and inexpensively test alternative face ventilation configurations will reduce the time between the generation of a design concept and its implementation by minimizing the tedious and costly experimental work required with existing design methodologies. If feasible, much improved designs for face ventilation systems should result

because it will be possible, for the first time, to screen a wide range of alternative configurations and quickly identify the most promising design concepts for a particular field application.

To achieve these objectives mentioned above, the authors conducted the following studies: (1) Development of a scaled physical model for examination of various ventilation scenerios and measurement of the airflow patterns using PIV (particle image velocimetry). These will be compared with simulations using commercial CFD software packages (CFD-2000, FLUENT, and CFX) with the objective of validating the CFD approach; (2) To refine modeling (numerical and scaled-physical) approaches, and to confirm that mathematical models and numerical method (CFD) can be used for design purposes of the face ventilation systems the full scale laboratory (gallery) tests will be performed at the Pittsburgh Research Center by the authors in conjunction with the MOSH, Dust and Ventilation Group; (3) based on the study performed the authors will down select a CFD software package, which could be, in cooperation with the software development company, converted to the specialized design tool utilized by the industry . It was determined that at the present time, CFD is not mature enough to be reliably used for ventilation design, though it can be used during the design process should sufficient validation be available.

Publications:

Wala AM, Stoltz JR, Hassan MI: Scale Modeling of A Mine Face Ventilation System for CFD Code Validation. Proceedings of the Third International Symposium on Scale Modeling, Nagoya, Japan, Septermber 10-13, 2000

Wala AM, Stoltz JR, Jacob JD: Numerical and Experimental Study of A Mine Face Ventilation System for CFD Code Validation. Proceedings of the Seventh International Mine Ventilatio Congress, Krakow, Poland, June 17-22, 2001

Wala MA, Jacob JD, Huang PG, Brown JT: 2002, A new approach to evaluate main face ventilation, Preprint 02-177:2002 SME Annual Meeting, Phoenix, AR February.Turner D, Wala MA, Jacob JD: 2002, Particle image velocimetry (PIV) used for mine face ventilation study, Proceedings: North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, June.

Wala MA, Jacob JD, Rangubhotla L, Watkins T: 2003, Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model, Perprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH February.

Wala MA, Jacod JD, Huang PG, Brown JT: 2003, New approaches to mine-face ventilation, Mining Engineering, Vol. 55, No. 3, March.

Wala MA, Jacob JD, Huang G, Rangubhotla L: 2004 How scrubbers help ventilate the face suring deep cut mining with a blowing curtain, Proceedings: 10th U.S. Mine Ventilation Symposium/North American, Anchorage, AL, May.

Mining Engineering Department, University of Kentucky,

230 M.M.R.B., Lexington, KY 40506

CFD Analysis of Mine Face Ventilation Systems

Final Report

January 8, 2005

Principal Investigator - Dr. Andrew M. Wala,

Mining Engineering Department

Co-Principal Investigators - Drs. Jamey D. Jacob, George Huang

Mechanical Engineering Department

NIOSH Grant R01/CCR 415822

Table of Contents

The major purpose of this proposed project is to examine CFD (computational fluid dynamics) as a technique for evaluation of modern underground coal mine face ventilation systems. This requires construction of numerical models of these systems, establishment of procedures to reliably obtain their computational solution, and, especially, validation of the numerical results with experimental data of the airflow.

The long-term objective of this work is to develop a reliable procedure utilizing CFD for analysis and design of face ventilation systems that are capable of protecting the health and safety of underground miners, particularly for the demanding situation of deep cut continuous miner sections with developed and used enhanced systems as water sprays and scrubbers.

Abstract

The use of extended (deep) cut mining with remotely controlled continuous miners is commonly utilized across 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 that higher levels of methane are liberated at the face during coal extraction. This requires an increase in the air supply to the face of the heading for adequate methane dilution. Higher air velocities are a consequence, which, in turn, lead 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 concerning miners' health and safety.

The former U.S. Bureau of Mines (currently NIOSH), Mine Safety and Health Administration, 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 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 potential to generate face ventilation designs without the previously mentioned disadvantages.

The major objective of this proposed project was to investigate the feasibility of developing a tool that facilitates the design of face ventilation systems to provide healthier and safer working environments for underground miners. The potential ability of this approach to reliably, quickly, and inexpensively test alternative face ventilation configurations will reduce the time between the generation of a design concept and its implementation by minimizing the tedious and costly experimental work required with existing design methodologies. If feasible, much improved designs for face ventilation systems should result because it will be possible, for the first time, to screen a wide range of alternative configurations and quickly identify the most promising design concepts for a particular field application.

To achieve these objectives mentioned above, the authors conducted the following studies: (1) Development of a scaled physical model for examination of various ventilation scenerios and measurement of the airflow patterns using PIV (particle image velocimetry). These will be compared with simulations using commercial CFD software packages (CFD-2000, FLUENT, and CFX) with the objective of validating the CFD approach; (2) To refine modeling (numerical and scaled-physical) approaches, and to confirm that mathematical models and numerical method (CFD) can be used for design purposes of the face ventilation systems the full scale laboratory (gallery) tests will be performed at the Pittsburgh Research Center by the authors in conjunction with the NIOSH, Dust and Ventilation Group; (3) based on the study performed the authors will down select a CFD software package, which could be, in cooperation with the software development company, converted to the specialized design tool utilized by the industry. It was determined that at the present time, CFD is not mature enough to be reliably used for ventilation design, though it can be used during the design process should sufficient validation be available.

Scientific Findings

The scientific findings of the grant are summarized below. Detailed findings and supporting data are presented in the papers included with this report.

1. The authors firmly proved that unsteady flow behavior (flow separation from the wall) is predominant during the box cut mining sequence, when a blowing curtain for face ventilation is used. This separation is driven by an adverse pressure gradient that builds up as the width of the entry decreases; a critical width of the entry of 16 feet was determined. This fact was first observed during the scaled physical model tests and next was verified during the full-scale tests. In such a case single pass continuous miners with wide-head (head extension) could be considered as a solution to avoid the separation problem.

   The flow studies in the face area, performed in the 1980s and 1990s for full scale models, were not fully understood because of limited opportunity to monitor the flow patterns in detail. The only source of information concerning similar flow behaviors were described by Luxner (1969) and Taylor et.al., (1997). Most of the researchers, who followed the Luxner (1969) comments concerning the higher effectiveness of the blowing systems versus exhaust systems, recommended that the blowing system was a better suited ventilation system for scrubber applications, particularly since the free jet with high initial momentum leaving the end of the line curtain should penetrate into the immediate face area and sweep the face and flow into the return. According to these studies, this is true as long as the entry width is more than 16 foot. This means that only during the slab cutting stage does the advantage of the blowing curtain prevail. For the rest of the mining phases, the ventilation air does not tend to flow to the immediate face area where it is mostly needed in the gassy mines for methane dilution. The width of the entry during the Luxner test was 10 ft. The width of entry for most of the studies performed at the NIOSH testing gallery was 16.5 ft.

2. The flow separation from the wall is a complex fluid dynamic phenomenon, difficult for many CFD packages to simulate due to their treatment of wall boundary conditions. During the early stage of the project, the CFD-2000 simulation package was used to simulate some of the scenario tested using the scaled physical model. At this time it was observed that the CFD-2000 results did not predict similar flow patterns as measured and seen in the physical model. A lot of effort was expended trying to create the boundary layer grid in CFD-2000 to get efficient and effective solutions, however there was a deficiency in the mathematical model used in this software and separation was never predicted. Since one of the goals of the project was to determine a reliable CFD package to analyze the mine face ventilation systems, a different CFD package was chosen. At the time the Fluent CFD software package was chosen due to its credibility. However, a few months later another commercially available CFD program, CFX, was tested too.

3. Fluent and CFX are both simulation software packages which were tested and have the ability to predict the flow within the face area of the continuous miner section. During these studies, four basic mining scenarios were explored to compare the CFD results with experimental data taken by PIV. Each used different mathematical models for boundary conditions and different size grids. Proper model generating procedures are described in the Annual Progress Report 2003, Brown (2003). This report will be a part of the Masters Thesis, which is still pending. Results of this study shows that both of these CFD software programs have difficulty predicting the flow separation.

4. To the authors' knowledge, this is the first time that a flow pattern of this detail has ever been measured and visualized for a mine opening. The figure 8 recirculation pattern in the narrow opening scenario, while previously predicted but ignored, has been confirmed in scaled, full-scale, and numerical tests, providing sufficient evidence that this geometry should be avoided during mine cut sequences.

5. The flow patterns measured for the exhaust ventilation (at the face) provides detail and valuable information concerning the effectiveness of this system. Ventilation engineers designing face ventilation systems for exhaust arrangements can use the data collected during these studies to improve their designs.

6. The study examined the effect of machine-mounted scrubbers on the performance of a face ventilation system for deep cut mining, partially proving the earlier results obtained by a study performed by the USBM, concerning the ratio between the scrubber flow $\mathbf{Q_s}$ and the ventilation flow $\mathbf{Q_{in}}$. According to the USBM studies this ratio should be equal to one. The authors demonstrated and discussed the flow behaviors in the face area for the different values of the $\mathbf{Q_s/Q_{in}}$ ratio.

7. In comparing the final numerical (CFD) results with the experimental (PIV) results, one will see good similarity in the flow behavior. However, some flow pattern discrepancies can be noticed. The experimental results show the location of the separation point to be more sensitive to smaller values of the scrubber flow ratio. This flow behavior could be explained by existence of the unsteady flow along the wall (rib). However, once the scrubber flow ratio reaches a level near 1.0 the separation is removed and the flow carries to the immediate face area. This is notable since most scrubbers in actual mining applications are required to have a flow ratio equivalent to 1.0. Also, it can be seen that as the flow ratio approaches one the recirculation (eddies) in the immediate face area increased.

8. The numerical simulations for these scenarios with the scrubber were examined using CFX, a commercial CFD package. Based on the intensive validation studies performed as a part of the entire project, the CFX software package along with the SST (Shear Stress Transport) turbulence model was selected for its ability to accurately predict separation due to adverse pressure gradients, as opposed to the k-ε model.

9. The scrubbers were shown to be very useful tools in moving air into the immediate face area where ventilation is poor, especially during the box cutting sequences.

10. Scrubbers introduced as dust collection devices help bring the air to the immediate face area and should be termed face ventilation devices, as well. Without a dust scrubber, the blowing curtain is ineffective for controlling respirable dust and methane.

11. With scrubbers, more efficient ventilation can be obtained in the face areas without the need for miners to extend ventilation controls inby permanent roof supports.

12. The flow ratio between the intake and scrubber is an important factor, which could be controlled for optimization of the face ventilation system.

13. The scrubber's performance was tested using the scaled physical model and simultaneously examined using developing CFD models of these scenarios; the CFD results very close to the experimental results because the separation process was eliminated.

14. The authors designed and built the scaled physical model of the face ventilation systems and developed its operating procedure. This model is equipped with a PIV measuring system. The testing facility proved to be essential for validation studies, such as the ones performed for this project. It can also be used for assessment and approval of face ventilation plans submitted by a mining company to MSHA. Its usefulness was demonstrated multiple times by the authors (Wala et. al., 2003).

15. The authors validated the scaled model results by using a full-scale testing gallery at the NIOSH Pittsburgh Research Center. The comparison of these experimental results proved that the scale physical model is a reliable tool to perform modeling studies for validation of CFD results.

16. Problems getting accurate CFD results and the experimental flow study taking more time than anticipated both caused the third task of the project to not be completed. Development of a custom (specialized) and reliable simulation CFD software package, which could be used by industry, is still being questioned at this stage.

17. The results of these studies uncovered a significant amount of knowledge about face ventilation in the case of extended (deep cuts) mining. These findings will add to the safety and health of miners.

Scientific Report

Since the advent of remote controlled continuous miners and extended cuts for room and pillar mining operations in the last two decades, the efficiency and productivity of mining operations has increased significantly. However, this high production presents environmental problems in that, per unit time, more dust is generated and higher levels of methane are liberated at the face during coal extraction. These conditions require the supply of more air into the face of the excavation heading for methane dilution. This must be done with care since more air means higher air velocity, which typically leads to the entrainment of more dust. Increased dust entrainment has been, at many operations employing these technologies, countered by the use of large capacity scrubbers mounted on the continuous miners. These scrubbers have a significant influence on the airflow distribution (flow patterns) in the heading and the design of their air handling capacities as well as the location of their inlet and outlet ports must be done with great care if the quality of the face environment for mine personnel is to be satisfactory. Thus, these circumstances give rise to many questions concerning miner health and safety.

The design of a balanced ventilation system for the scenarios described above requires consideration of the system in a three-dimensional, rather than in 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 numerical technique by which many complicated fluid flow problems can be solved. CFD embraces a variety of technologies, including mathematics, computer science, engineering, and physics, and these discipline have to be brought together to provide the means of modeling fluid flows, and it possibly has potential to generate face ventilation designs without the previously mentioned disadvantages of the theoretical and experimental methods.

The objective of this project is to attempt to develop a reliable procedure for analyzing and designing mine face ventilation systems, including an ability to assess novel designs, using CFD. Due to the complexity of face ventilation systems and increased performance demands on such systems through practices such as deep cut mining, there is a need to apply sophisticated engineering tools to study and design these systems before mine implementation.

A variety of face ventilation plans must be designed by the mine operators and evaluated by Mine Safety and Health Administration (MSHA) authorities, for critical factors such as an adequate air sweep of the face, compliance with health and safety regulations pertaining to acceptable methane concentration levels and worker exposure to dust. Existing theoretical and experimental approaches to this problem limit the ability of the mining engineers to design proper face ventilation plans and MSHA's authorities to evaluate these plans.

Using CFD, a ventilation system can potentially be designed and assessed quickly and inexpensively, avoiding costly underground or laboratory trials to test the alternative arrangements. CFD would enable the designer to perform studies of a much wider range of configurations than could ever be considered through experimental means. CFD has been attempted in South Africa, England, Japan and the United States in order to study the flow of air and contaminants in underground mine openings; however, this work has not yet been developed to the extent that could really benefit the health and safety of miners.

CFD could be used to predict flow fields and the dust and methane distributions on continuous miner faces. However, before such numerical modeling can be confidently used for the design of face ventilation systems, the results must be proven correct and effective. Thus, the following research was proposed by the authors: (1) construction of sufficiently detailed and realistic numerical models of the face ventilation systems; (2) establishment of procedures to reliably obtain the numerical solution of these models, and, especially, (3) validate the numerical results with experimental data to ensure that the model assumptions and numerical results are accurate.

To be able to achieve these objectives mentioned above, the authors proposed the following studies: (1) in cooperation with the Dust and Ventilation Group, NIOSH - Pittsburgh Research Center, review past data from experimental studies examining the performance of face ventilation systems and establish the baseline face ventilation scenarios to be tested. The numerical models for these basic scenarios will be developed and computer simulations, using commercially available CFD software packages (CFD2000, FLUENT, and CFX), would be performed. The simulation results, in the form of plots of air flow vectors, will be qualitatively and quantitatively compared with experimental data collected during the scaled-physical model tests with the objective of validating the CFD approach; (2) to refine modeling (numerical and scaled-physical) approaches, and to confirm that mathematical models and numerical methods such as CFD can be used for design purposes of the face ventilation systems, full scale laboratory (gallery) tests will be performed at the Pittsburgh Research Center by the authors in conjunction with the NIOSH, Dust and Ventilation Group; (3) based on the results of the study the authors will choose an effective and reliable CFD software package, which could be, in cooperation with software development company, converted to a specialized design tool for use by the mining industry.

During the first year of the project, the authors concentrated on numerical model development for chosen face ventilation scenarios, using, at that time, the commercial CFD software package named CFD2000. In the same time, the authors designed and built a scaled physical model of the face ventilation system to study the flow using the PIV facility of the TSI Company, at St. Paul, Minnesota. In the mean time the opportunity for PIV testing developed at the University of Kentucky. The Mechanical Engineering Department arranged the laboratory for PIV testing. The first test of the scaled physical model of the face ventilation system was performed using this facility. Results of these studies lead to the changes of the existing plan of the project. The modified plan for the project was submitted with the Progress Report (7/1/2000 to 4/16/2001).

The preliminary results, obtained from the flow studies using the numerical and physical simulation of the flow behavior in the face area using a blowing curtain show significant discrepancies. Unexpected flow patterns were seen during the experimental studies of the flow distribution for the 30 ft box cut scenario using the blowing curtain. This was the first time that a flow pattern of this detail has ever been measured for a mine opening to the authors' knowledge. The flow in the study area has complicated features, including separation and recirculation, necessitating the need for a proper boundary model for the numerical simulations. However, the CFD simulation results, using CFD2000, did not show this flow behavior. At that time the decision was made that CFD2000 was not going to be used for any further testing. The important characteristic of the flow pattern are discussed by Wala et al. (2000 and 2001) specifically for the box cutting sequences.

To be able to understand better the complex flow behaviors in the face area for varied mining scenarios and to see the effects of various parameters on the ventilation arrangements, the plan for the scaled physical modeling studies was expended and a custom designed Particle Image Velocimetry (PIV) measuring system (Turner et al., 2002) was added to the scaled physical model. At the beginning series of studies concerning the effect of various geometric and flow parameters on the air flow behaviors were extensively examined (Wala et al., 2002 and 2003). The important characteristic of the flow was that 70% to 80% of the air leaving the end of the blowing curtain almost immediately separates from the rib (wall) and a major part of the primary flow turns back towards the return and the rest of the curtains airflow, approximately 20% to 30%, called the secondary flow, continues to flow towards the immediate face area in the opposite direction. As can be seen in general, the ventilation air delivered by the blowing curtain does not flow into the immediate face area.

The Graduate Student, Lavanya Rangubhotla, carried out the continuation of this parametric study as part of her Master Thesis. Most of these experimental data utilizing the scaled physical model were used for the validation of the CFD simulations. For the further studies of the flow behavior, the experiments were conducted for different scenarios with different combinations of parameters such as curtain setback, intake air quantity delivered by the curtain, tight-rib distance (curtain distance from the rib), and mining sequence. The data collected during these studies allows making conclusions concerning the flow behavior to assist the design process of the face ventilation system. The results for the above studies are presented and discussed in the Master Thesis included with this report (Rangubhotla , 2004).

During these studies it was found that primarily for the box cutting sequences, a certain width of the face zone was unable to sustain ventilation air in the immediate face area. Therefore, the effect of a scrubber system on face ventilation was studied. In reverse, some situations exist when the miner machine is not equipped with the scrubber and the exhaust ventilation system is preferred. The results of these studies are presented and discussed by Wala et.al., (2003) and Rangubhotla (2004).

As this unsteady flow behavior (flow separation from the wall) was predominant and unexpected, full-scale measurements were needed to validate this flow behavior. To ensure that scaled model test results interpret conditions reliably in the underground face area cooperative validation testing has been performed by the Mining Engineering Department, University of Kentucky and Pittsburgh Research Center NIOSH, using full-scale testing gallery at the NIOSH Research Center. These studies proved that a 1:15 scaled laboratory model used for the project is a valid tool and realistically simulated underground conditions for the face ventilation systems. Observations showed that the flow patterns for one of the scenarios tested using the laboratory scaled model and the full-scale model are similar. These discussions can be seen in Chapter 6 of the Master Thesis (Rangubhotla, 2004).

The Graduate Student, Justin Brown, performed the tasks dedicated to the numerical model development and analysis of the CFD results. The numerical models for four basic mining scenarios including (a) 30 foot box cut, (b) 30 foot slab cut, (c) 60 foot box cut and (d) 60 foot slab cut were developed. In all four scenarios the geometry of the mining equipment was excluded for the purpose of simplifying the modeling procedure.

Because the solution of an analysis can be dependent on how the model is discretized, it is proper procedure to seek a grid-independent solution before accepting any results with confidence. As mentioned before, problems were encountered in obtaining a proper solution using simulation software CFD2000 in matching the experimental results from PIV. There are many different methods to model turbulence and some ways are more effective than others depending on the application. Also, some turbulence models are more easily converged than others. The Reynolds Average Numerical Solutions (RANS) models are usually faster but are only accurate in certain applications and geometries. Three RANS turbulence models including the Standard $k - \varepsilon$, Spalart-Allmaras and Shear-Stress Transport $k - \omega$ models were used and the results were compared to find the most robust and most accurate model. During this part of the study some problems with using the commercial software package Fluent were encountered in trying to predict the onset of separation that occurs for certain mining scenarios. This is one of the reasons that more studies are needed concerning the validation of the CFD simulation results before being used for of predicting the flow within the face area.

Additional studies for different grids (coarse, moderate and fine) using both Fluent and CFX were performed to find a grid-independent solution. From this series of simulations the $k - \omega$ SST turbulence model was selected for its ability to accurately predict separation due to adverse pressure gradients. Some part of this work is documented in two Progress Reports from 2002 and 2003. These two reports are included with this document.

The second part of the study concerning numerical model development and analysis of the CFD results was dedicated to the exploring the effects of the introduction of the scrubber at the face area. When the scrubbers were added to the face ventilation system of coal mines, the existing arrangements of the face ventilation system were altered to accommodate these machines.

Many studies concerning face ventilation with scrubbers were carried out in United States in the last twenty years. Some of them which have contributed greatly to this topic include Gillies (1982), Volkwein and Thimons (1985). Jayaraman et al., (1992), and Kissell (2003). Most of the researchers, followed the Luxner comments (1968) concerning the higher effectiveness of the blowing system versus the exhaust system, and thus recommended the blowing system as the "best" ventilation systems for scrubber applications, particularly because the free jet with high initial momentum leaving the end of the line curtain should penetrate into the immediate face area and sweep the face and flow into the return. This is true as long as the entry width is more than 16 foot, as was discovered during this study (Wala et al., 2001). This means that only during the slab cutting scenario does the advantage of the blowing curtain prevail. For the rest of the mining scenarios the ventilation air does not tend to flow to the immediate face area where it is mostly needed in gassy mines for methane dilution. The results showing these behaviors are presented in the attachments to this report.

The effects of machine-mounted scrubber system on levels of methane and respirable dust were tested first by Ingersoll Rand Research Inc., (Clouse, 1982). These studies have shown that properly designed machine-mounted scrubber systems can allow curtain setbacks of 12.2m (40 ft), while still controlling respirable dust and methane levels. At that time, it was shown that the application of the machine-mounted scrubber primarily helped to collect respirable dust at the face (dust source location) and improved the visibility. The experiments were performed for different ventilation ratios *(Qs/Qin)*, where *Qs* and *Qin* are the scrubber flow rate and curtain inlet flow rate, respectively. Based on the Ingersoll Rand studies, MSHA recommends a ratio between the scrubber flow *Qs* and the intake flow *Qin* of close to one. A study performed by Jayaraman, 1992, supports these finding, however, latest studies performed by Taylor et al., 1997 and by Thimons et al., 1999 show that the relations between face ventilation performance and scrubber/ventilation flow ratios are more complicated (due to recirculation, etc.) than was previously thought. From the experimental studies performed during this project it can be seen that different ratios between these two flows could be used to optimize the performance of the ventilation system for the extended cuts. The authors conclude that Qs/Qin could be optimized depending on miner location (depth of the cut). The preliminary results concerning scrubbers performance were presented by Rangubhotla (2004) (experimental data) and by Brown (2003) (numerical data) and by Wala et.al., (2004) (combined).

Acknowledgments

This study was supported by the National Institute of Occupational Safety and Health (NIOSH), under Grant No. R01/CCR415822. The authors would like to express their gratitude to the Joy Mining Machinery, Inc. for providing the scale of the continuous miner used during this study.Special thanks are also extended to Drs. K. Saito and M. Hassan, from the Mechanical Engineering Department, University of Kentucky, for their help in collecting preliminary PIV data.

Personnel

Personnel from the University of Kentucky supported on this project included the Principal Investigator, Dr. Andrew M. Wala, (Mining Engineering Department) and the Co-Principal Investigators, Drs. Jamey D. Jacob and George Huang (Mechanical Engineering Department). Graduate students included Lavanya Rangubhotla and Justin Brown (Mechanical Engineering Department) and Jason Stoltz and David Taylor (Mining Engineering Department).

References

Brown, J. T., 2003, "Prediction of mine face ventilation using Computational Fluid Dynamics, Progress Report, October.

Clouse, B.E., 1982, "Assess Feasibility of Machine Mounted Scrubber System for Ventilation Coal Mine Working Face. Phase 1, Interim Report," Ingersoll-Rand Research Co., New Jersey.

Gillies, A.D.S., 1982, "Improvement Coal Face Ventilation Through Use of Dust Scrubber Systems Leads to Greater Production Efficiency." Proceedings, First Mine Ventilation Symposium, Tuscaloosa, AL.

Jayaraman, N.I., R.A.,Jankowski, and K.L. Whitehead, 1992, "Optimizing Continuous Miner Scrubbers for Dust Control in High Coal Seams, Proceedings of the 121st Annual SME Meeting, Phoenix, AZ.

Kissell, F.N., 2003,"Handbook for Dust Control in Mining", Information Circular IC 9465, U.S. Department of Health and Human Services, Public Health Service, Centers for Disease Control and Prevention, National Institute for Occupational Safety and Health, Pittsburgh Research Laboratory, Pittsburgh, PA.

Luxner, J.V., 1969, "Face Ventilation in Underground Bituminous Coal Mines." Bureau of Mines RI 7223.

Rangubhotla, L., 2004, "Investigation of Characteristics of Bounded Wall Jets in Dead End Mine Headings", MS Thesis, December.

Taylor, C.D., 2000, "Effect of Water Sprays and Scrubbers Exhaust on Face Methane Concentrations." Proceedings: Seventh International Mining Ventilation Congress, Crakow, Poland.

Taylor, C.D., 1997, "Impact of Unbalanced Intake and Scrubber Flows on Face Methane Concentrations," Proceedings: Seventh International Mine Ventilation Congress, Pittsburgh, PA.

Thimons, E.D., Taylor, C.D., and Zimmer, A., 1999, "Ventilating the Box Cut of a Two-Pass 40-ft Extended Cut," Proceedings: Eight US Mine Ventilation Symposium, Kingston, Canada.

Turner, D., Wala, M.A., Jacob, J.D., 2002, "Particle image velocimetry (PIV) used for mine face ventilation study," Proceedings: North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, June.

Volkwein, J.C., E.D. Thimons, and G. Halfinger, 1985 "Extended Advance of Continuous Miner Successfully Ventilated with a Scrubber in a Blowing Section, Proceedings of the 2nd U.S. Mine Ventilation Symposium, Reno, NV.

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

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2002, "A new approach to evaluate mine face ventilation," Preprint 02-177: 2002 SME Annual Meeting, Phoenix, AR, February.

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

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2003, "New approaches to mine-face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J.D., Huang, 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/ North American, Anchorage, AL, May.

Publications

Journal and Conference Publications

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

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2002, "A new approach to evaluate mine face ventilation," Preprint 02-177: 2002 SME Annual Meeting, Phoenix, AR, February.

Turner, D., Wala, M.A., Jacob, J.D., 2002, "Particle image velocimetry (PIV) used for mine face ventilation study," Proceedings: North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, June.

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

Wala, M.A., Jacob, J. D., Rangubhotla, L., and Watkins, T., 2003, "Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model," Preprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH, February. (Accepted for publication in Mining Engineering.)

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2003, "New approaches to mine-face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J.D., Huang, 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/ North American, Anchorage, AL, May.

Reports

Brown, J. T., 2002, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, June.

Brown, J. T., 2003, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, October.

Theses

Rangubhotla, Lavanya, 2004, "Investigation of Characteristics of Bounded Wall Jets in Dead End Mine Headings", MS Thesis, December.

Publications Included in the Report

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2003, "New approaches to mine-face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J. D., Rangubhotla, L., and Watkins, T., 2003, "Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model," Preprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH, February. (Accepted for publication in Mining Engineering.)

Wala, M.A., Jacob, J.D., Huang, 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/ North American, Anchorage, AL, May.

Brown, J. T., 2002, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, June.

Brown, J. T., 2003, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, October.

New approaches to mine-face ventilation

A. WALA, J. JACOB, J. BROWN AND G. HUANG

A. Wala, member SME, is professor with the Department of Mining Engineering, University of Kentucky, Lexington, KY. J. Jacob, J. Brown and G. Huang are associate professor, graduate student and associate professor, respectively, with the Department of Mechanical Engineering, University of Kentucky, Lexington, KY. Preprint number 02-177, presented at the SME Annual Meeting, Feb. 25-27, 2002, Phoenix, AZ. Original manuscript accepted for publication August 2002. Discussion of this peer-reviewed and approved paper is invited and must be submitted to SME Publications Dept. prior to June 30, 2003.

Introduction

The enhancement of continuous-miner technology in underground coal mines in the United States, including the use of remote-control miners, deep cuts and continuous haulage, has led to highly productive and efficient operations. However, this high production presents environmental problems in that, on a per-unit-of-time basis, more dust is generated and higher levels of methane are liberated at the face during the extraction of the coal. These conditions require the supply of more air into the face of the excavation heading for dilution. Several ventilation techniques have been used in conjunction with this modern mining methods (Goodman et al., 1990). These include machine-mounted scrubbers, extensible line curtains/flexible tubing, jet fans and combinations of these techniques. Implementation of these techniques has had an influence on the airflow distribution (flow patterns) in the mining heading and the design of their air-handling capacities.

The design of a balanced ventilation system for the above scenarios requires the consideration of the system in three, rather than two, dimensions. With computational fluid dynamics (CFD), many design alternatives can be explored in a short period of time, and the detailed three-dimensional results can be viewed graphically. This graphical type of information concerning airflow, gas and dust distribution established by CFD are readily rationalized and may lead to confidence in the final design of the mine face-ventilation system. However, to develop this confidence, extensive validation studies are required involving comparisons between physical and numerical (CFD) modeling (Hargreaves and Lowndes, 2001).

Computational fluid dynamics

CFD is a numerical technique by which complex fluid-flow problems can be solved economically within a relatively short time. CFD embraces a variety of sciences, including mathematics, computer science, engineering and physics. These disciplines have to be brought together to provide the means of modeling fluid flows, which by their nature are nonlinear in behavior. CFD is now a standard tool for design in the aircraft, automobile and turbo-machinery industry. The application of CFD for mine ventilation (Sullivan and Van Heerden, 1993; Brunner; 1995; Moloney et al., 1999; Hargreaves and Lowndes, 2001) may help engineers analyze complex airflow fields and contaminant transport. This would allow them to design more economical and safer systems.

Development of the numerical model for the flowfield study requires an analyst to provide a substantial amount of information. The quality of this information, in terms of suitability and accuracy, determines the overall value of the simulation. In particular, the geometry specification is important for all CFD simulations. It is desirable to describe the physical boundaries that contain the fluid as accurately as possible. This is true for engineering problems, where usually the effect of one or more man-made objects is to be predicted. Any inaccuracy in the specification of the bounding surfaces may produce a misleading result.

Abstract

Computational fluid dynamics (CFD) is a continually developing numerical technique by which complex fluid-flow problems can be solved. This technique can be an attractive approach to predict, evaluate and design proper mine face-ventilation systems. However, to develop confidence in the accuracy of CFD, the verification of numerical and experimental results is required for validation purposes.

During the last two years, such a validation study was funded by the National Institute for Occupational Safety and Health under Grant R01 CCR415822. It was carried out in the Department of Mining Engineering at the University of Kentucky. In this paper, the authors present, compare and discuss the experimental and numerical data collected during the period of the project. It was found that, in addition to the needs of establishing a grid-independent solution, the proper choice of codes and models may have a profound impact on the accuracy of the solution.

Therefore, comprehensive validation work of the current CFD tools against mining-related benchmark experiments will become an important issue in the transfer of the CFD technique in designing mine face-ventilation systems. In addition, nonintuitive flow field behaviors were discovered in the face-ventilation system in the experimental portion of the project. It was discovered in several geometries that the face was not properly ventilated.

FIGURE 1

Picture of the scaled physical model.

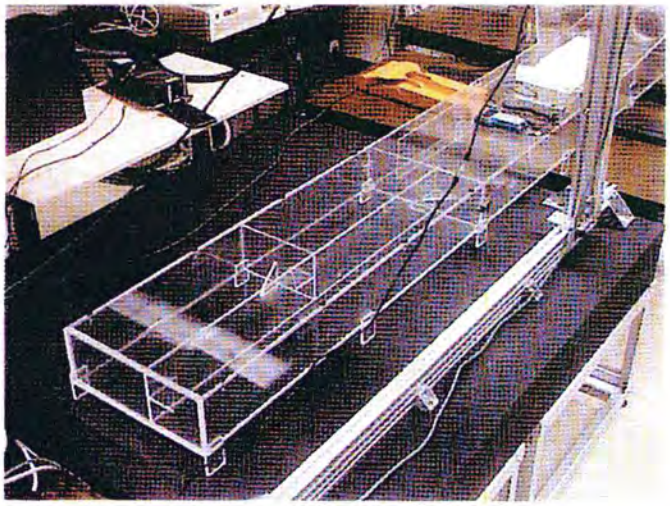


FIGURE 2

Schematic of the integrated PIV system.

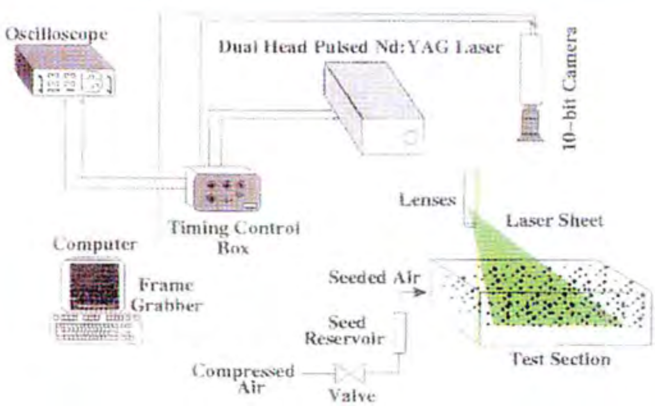
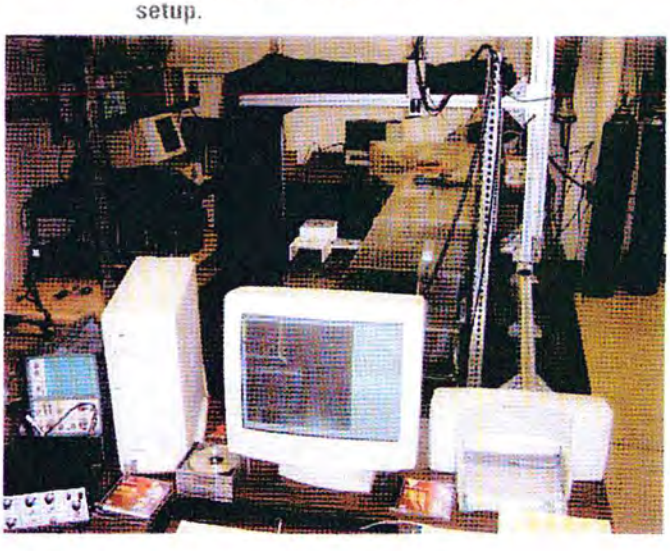


FIGURE 3

Overview of the experimental setup.



To make discrete the partial differential equations over a computational domain, the domain must be gridded, regardless of the discretization scheme used. The values of the variables at these grid points are used to develop equations for a numerical solution. Geometry and grid generation used to take a month or longer to perform. Today, the geometry set-up and grid-generation processes have been made much easier, albeit still time consuming. This is because most of the software packages have computer-aided drafting (CAD) or similar systems for automatic grid generation. Although CFD has many advantages, it does not eliminate the need for expensive and time-consuming experimental (scale-modeling) studies (Gillies, 1982; Wala et al., 2000; Wala et al., 2001) that are still required for verification of the numerical results.

Validation study at the University of Kentucky

The validation study was funded by National Institute for Occupational Safety and Health Grant R01 CCR415822 (Wala et al., 2000; Wala et al., 2001). It is being performed in the Department of Mining Engineering at the University of Kentucky. The goal of this study is to determine CFD's ability to predict, evaluate and design effective face-ventilation systems. During this effort, the authors performed the following tasks:

- Designed and built a physical model of selected face-ventilation systems.
- Performed a series of measurements, using particle image velocimetry (PIV), to determine the velocity fields in the model.
- Developed three-dimensional numerical models of the tested systems and performed computer simulations using CFD codes.
- Compared experimental and numerical simulation data for the purpose of CFD code validation.

Description of the scaled physical model. A 1:15 scale model of a 2.1-m- (7-ft-) high, 6.1-m- (20-ft-) wide mine face area was designed and built out of transparent Plexiglas (Fig. 1). Scaling of the geometry was performed using dimensional analysis based on the Reynolds Number, $Re=UL/n$, where U and L are the velocity at the brattice line mouth and width of the inflow (distance between the brattice line and entry rib), respectively. The model consists of three significant parts:

- The middle section, which represents an ordinary mining entry with a partition (brattice line), divides the entry into a narrow 0.6-m- (2-ft-) wide intake and an 5.5-m- (18-ft-) wide return airway.
- The working section, which represents a face area, where movable Plexiglas walls can be used to replicate many different ventilation configurations such as a box-cut or a slab-cut arrangement with brattice setback distances varying from zero to 18.3 m (60 ft).
- The intake/return section, where the ventilation system can be arranged as either blowing or exhausting by changing the fan location.

During this study, the model was set up as a blowing ventilation system with a setback distance of 10.6 and 18.3 m (35 and 60 ft). Description of the particle image velocimetry system.

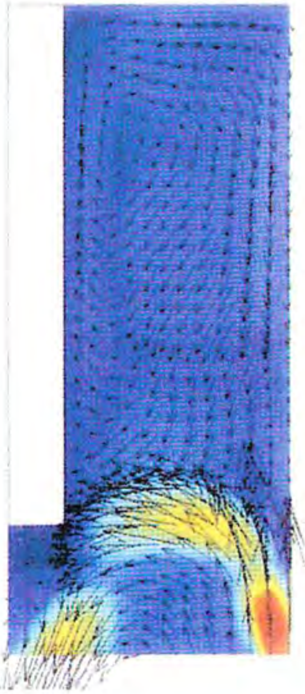


FIGURE 4

Velocity vectors from PIV measurements, Scenario 1.

The PIV technique was proposed to measure instantaneous velocity fields of the time dependent flows in the scaled model. Due to the relatively high air velocity in the model, a pulsed laser system was required. To perform these measurements, tracer particles must be added to the flow field: highly reflective ~10-μm (1,250-mesh) diam talc particles were used in the experiments discussed herein. The laser-light sheet illuminates these particles twice within a short, controlled time interval. The reflected (scattered) light from each laser pulse is sequentially imaged by a CCD camera and recorded digitally to a computer using a frame grabber. To create one velocity field, approximately 50 pairs of instantaneous images of velocity fields are recorded for digital analysis.

The schematic and overall picture of the integrated PIV system used in this study are shown in Figs. 2 and 3, respectively. This system is comprised of the following components:

- Laser and light sheet optics.
- Image capture.
- Synchronizer.
- Smoke particle generator and
- Image analysis and display.

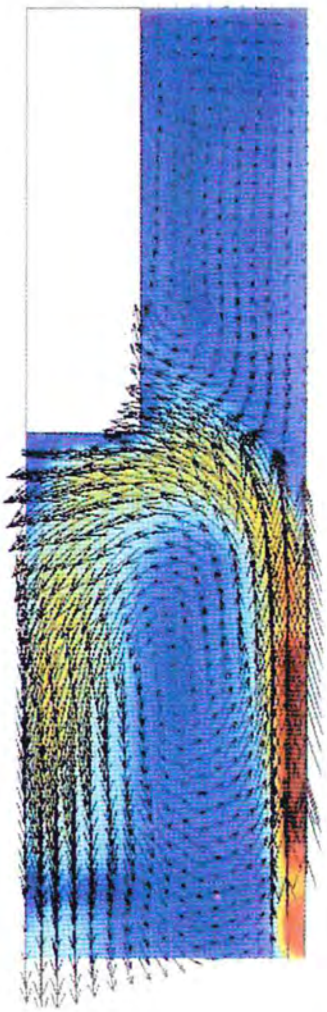


FIGURE 6

Velocity vectors from PIV measurements, Scenario 3.

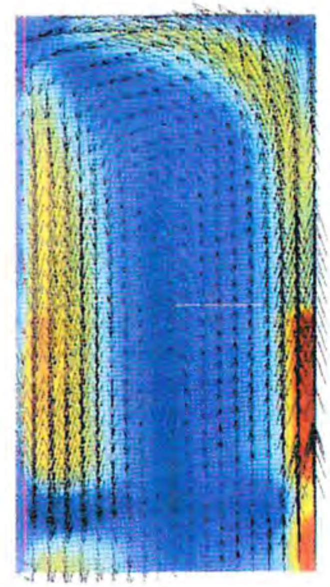


FIGURE 5

Velocity vectors from PIV measurements, Scenario 2.

The laser is a double-pulsed 50 mJ Nd:YAG laser. The tracer particle motions are recorded using a Kodak Megaplus 10-bit monochrome charge coupled device camera. Images are stored to a personal computer (PC) using an Epix PIXCI-D digital frame grabber. The frame grabber/camera combination has a double-trigger feature that allows two images to be acquired back-to-back within an extremely small time (~10 μm or 1,250 mesh). Timing is controlled by a Taitech DG-100 timing control unit, which is monitored using an oscilloscope.

Results

Experimental data from scale modeling. By using the physical model of the face ventilation system (described above), five mining scenarios were tested. They represented possible coal mining sequences for 18.3-m (60-ft) depth extended cuts. Deep cuts of this length can be taken as two 18.3-m (60-ft) cuts, four 9.15-m (30-ft) cuts or some other cutting sequence combination. For this study, a four-cut sequence operation was adopted:

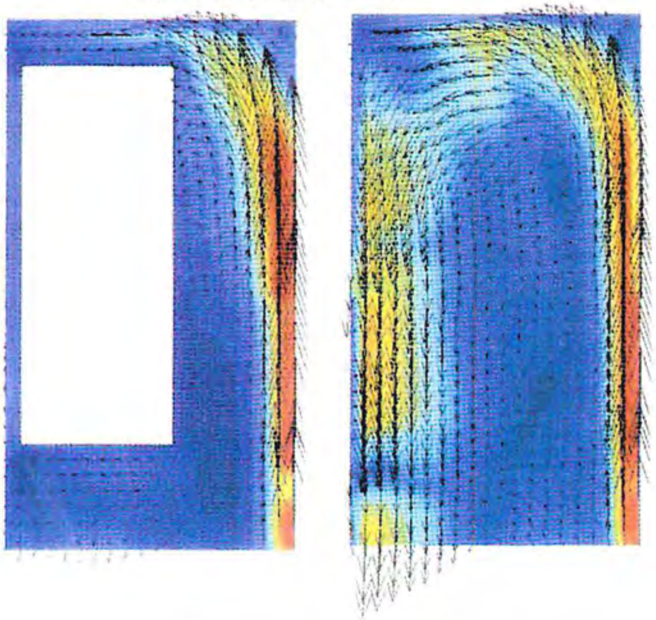
- Scenario 1: First mining sequence — completion of the 9.15-m- (30-ft-) deep box cut.
- Scenario 2: Second mining sequence — completion of the 9.15-m- (30-ft-) deep slab cut

FIGURE 7

Velocity vectors from PIV measurements, Scenario 4.

FIGURES 8A AND 8B

8A, left: Velocity vectors from PIV measurements, Scenario 5, mid-plane of miner.8 B, right: Velocity vectors from PIV measurements, Scenario 5, above miner.



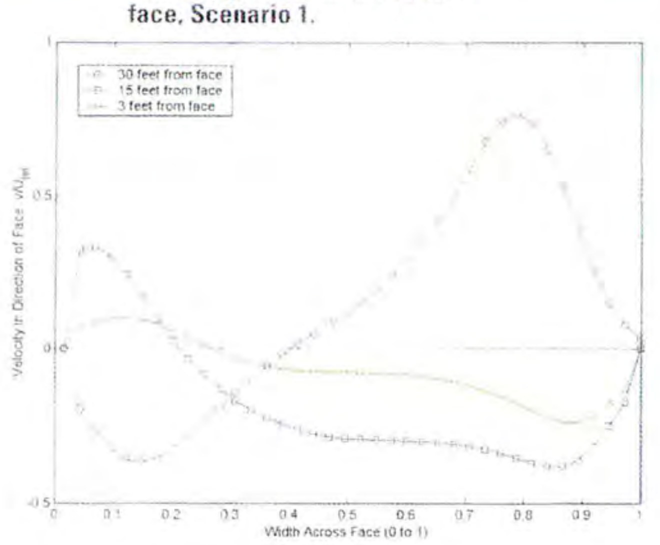
- Scenario 3: Third mining sequence — completion of the second 9.15-m- (30-ft-) deep box cut.
- Scenario 4: Fourth mining sequence — completion of the second 9.15-m- (30-ft-) deep slab cut.

There is also Scenario 5. It was identical to Scenario 2 except that the rectangular box simulating equipment (continuous miner) was placed in the face area.

For the initial study presented here, the authors decided to investigate the flow pattern in the empty face area (no mining equipment present). This was done to provide flow field models without disturbances caused by equipment.

FIGURE 9

Velocity profiles in direction of face, Scenario 1.



Future investigations will take into account the presence of underground equipment in the study area. However, as mentioned above, in the fifth scenario tested during this study, a simple rectangular box was placed in the face area. This rectangular box was constructed of transparent Plexiglas with dimensions of 7.8 x 3.24 x 0.95 m (25.6 x 10.6 x 3.1 ft), corresponding to the dimensions of a Joy 14CM continuous miner.

Quantification of the performance of the face ventilation for all five extended cut mining scenarios was the goal for this validation study. The geometry of the entry being tested was 2.1-m- (7-ft-) high and 6.1-m- (20-ft-) wide. The intake air in volume of 2.5 m^3/sec (5,300 cfm) was provided to the face area by a blowing brattice line (curtain).

Figures 4, 5, 6 and 7 show the velocity vectors from PIV measurements at the horizontal middle plane of the model for the first four scenarios, respectively. Figures 8a and 8b show the velocity vectors measurements at the horizontal middle plane of the miner (box) and above the miner, respectively. This ventilation arrangement shown in Figs. 8a and 8b was described as Scenario 5.

Figure 9 shows the velocity profiles for three locations along the box cut at the middle plane (Scenario 1). These three locations were 9.1 m (30 ft), 4.5 m (15 ft) and 1 m (3 ft) from the face.

Simulation data from CFD. The CFD simulations presented here are for the five scenarios described above. To perform this analysis, the commercial code Fluent 6.0, a finite volume-based software package, was run using the multiple processor interfacing (MPI) of the HP Superdome Complex at the University of Kentucky. This system consists of 224 HP PA-RISC processors at 750 MHz each. They allow the user to use up to four simultaneous computations each utilizing eight processors. By allowing Fluent to automatically partition a grid consisting of 900,000 elements, the time required per iteration was decreased to five seconds or 15% of the time required while using only one processor. However, the completion (convergence) of the calculations required, on average, from 10,000 to 20,000 iterations. This meant that the total run times for each scenario was approximately 25 hours.

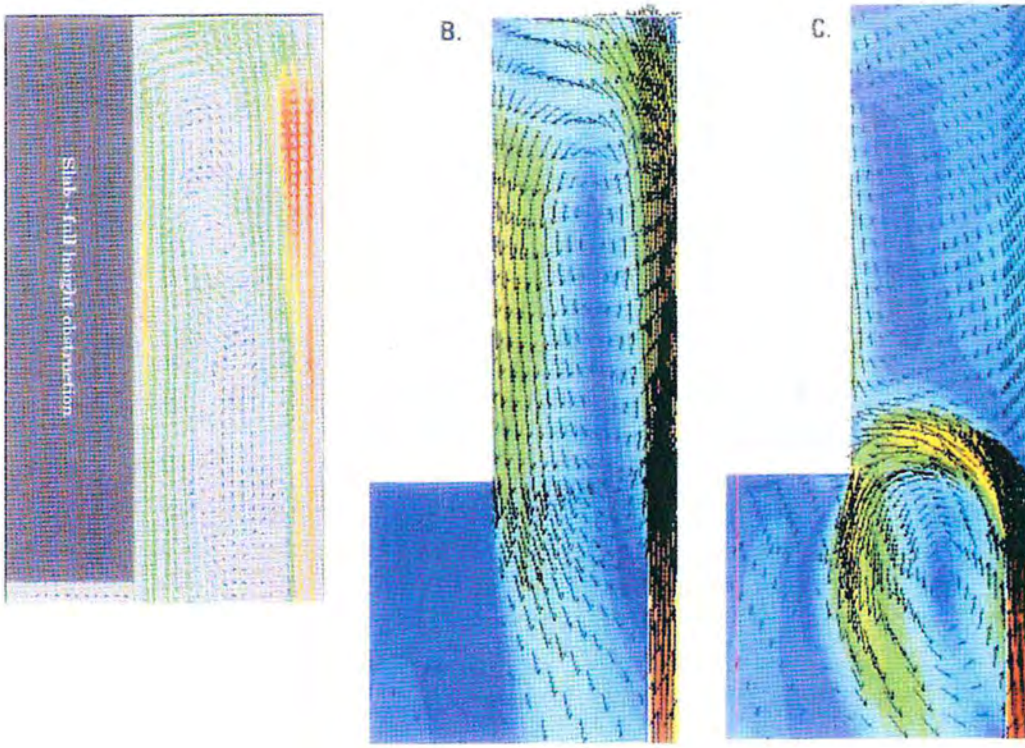
The three-dimensional numerical models analyzed are full-scale representations of a 2.1-m- (7-ft-) high, 6.1-m- (20-ft-) wide mine face area at various stages during its mining cycle. Scaling was performed using dimensional analysis based on the Reynolds Number (125,000 based on the inlet flow parameters).

First, the authors performed a grid sensitivity study on Scenario 1 using Fluent along with the k-ε turbulence model. For practicality, a "standard wall function" was used to resolve the wall boundary layer for all simulations. Wall functions approximate the boundary layer profile based on the Log-Law of the Wall.

Scenario 1 was chosen to demonstrate the sensitivity of the solutions to grids, codes' models in the numerical simulations because of the onset of separation in the flow located just past the end of the curtain (brattice line) and along the right-side wall. Such an abrupt turning of the flow makes it difficult to resolve unless adequate grid density and turbulence models are used. Interestingly enough, the authors also found that the variation of different commercial codes caused differ-

FIGURES 10A, 10B AND 10C

A. B. C.



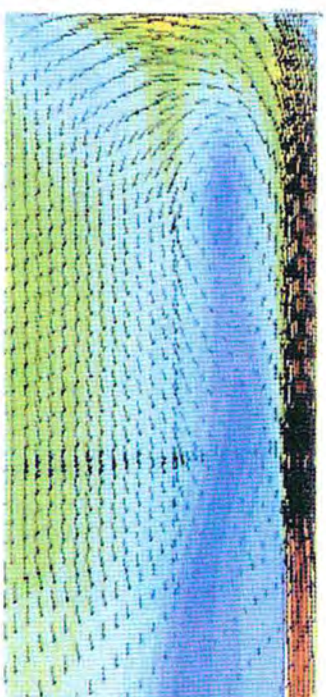
10A, left: CFD results using CFD 2000 and k-ε model, Scenario 1. 10B, center: CFD results using Fluent and k-ε model, Scenario 1. 10C, right: CFD results using Fluent and S-A model, Scenario 1.

ences in solution behavior. This may be caused by different numerical practices used by the code originators.

Although without showing it here, it was found that the solutions were sensitive to the grid mesh used. However, the authors were able to establish a grid-independent solution using the following practice. Three structured grid densities, including 600,000, 900,000 and 1.2 million cells, were solved to find any dependency of the solution on the grid size. All of the grid densities yielded the same solution. This meant that the solution was grid-independent. Several other cases were solved using the k-ε (two-equation) model and the Spalart-Allmaras (S-A) one-equation model (Spalart and Allmaras, 1992) to compare these two models. No changes were made other than the turbulence model. The results are shown below.

FIGURE 11

CFD results using Fluent and S-A model, Scenario 2.



Figures 10a and 10b show the simulation results obtained using CFD 2000 code with the k-ε model and Fluent with the k-ε model, corresponding to the Scenario 1 experiment shown in Fig. 4, respectively. CFD results for the same scenario using Fluent and the S-A model are also shown in Fig. 10c. As can be seen from this figure, only Fluent with the S-A model was in close agreement with the experimental data. It should be noted that the S-A model has been demonstrated to provide excellent behavior for flow with separation. Although the authors found similar conclusions in other experimental scenarios, only the comparisons using results obtain from Fluent with the S-A model are presented.

Figures 10, 11, 12, 13 and 14 show the airflow patterns found in the horizontal, middle plane of the face area for the Scenarios 1, 2, 5, 3 and 4, respectively. Figure 12 shows the result of a simulation that included the addition of a continuous miner near the face. Although not perfect, it was concluded that the current approach provides the best agreements with the experimental data.

It should be emphasized that the authors have not fully exhausted other combinations, such as using more advanced model and testing of another popular CFD code, CFX5. The current study only serves to demonstrate the need for a comprehensive validation work of the current CFD tools against mining-related benchmark experiments. This will become an important issue in the

FIGURE 12

CFD results using Fluent and S-A model, Scenario 5, mid-plane of miner.

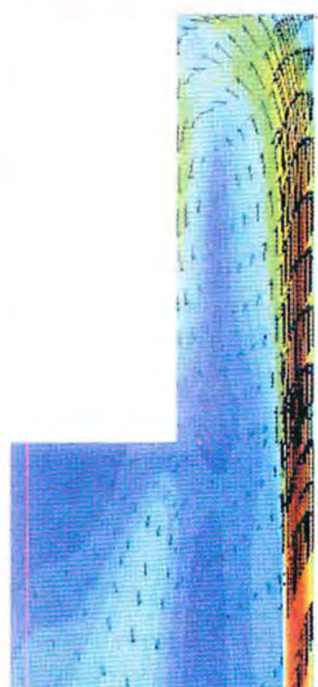
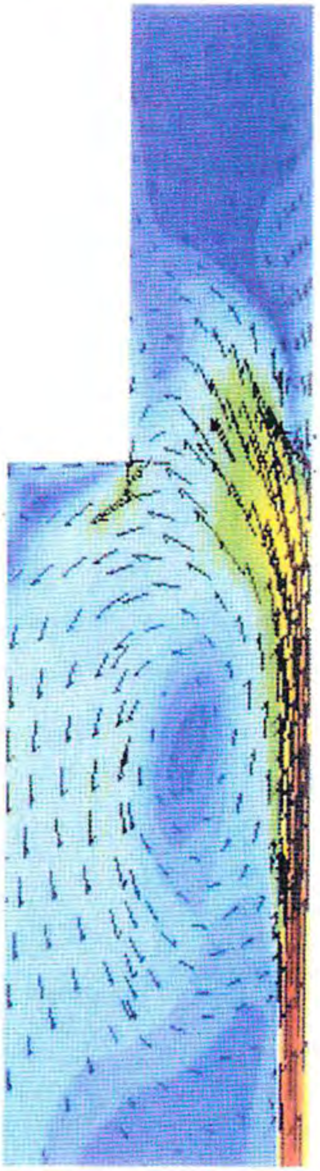


FIGURE 13

CFD results using Fluent and S-A model, Scenario 3.



transfer of the CFD technique in designing mine face-ventilation systems.

Conclusions

These studies indicate that this new way of using CFD to evaluate the mine face ventilation is very promising. However, many questions still need to be answered to develop full confidence in using CFD as a tool for design and optimization of the mine face ventilation for extended cuts.

Results from these studies demonstrated an increased understanding of the flow behavior in cases of mining the extended cut. The authors presumed that, until now, no one has fully understood the flow patterns in this mining scenario. This knowledge will change the way the face ventilation is approached. The study also shows that flow separation is the function of the geometry of the entry (viz., width of the box cut). The volume of the ventilated dead-end area and amount of air used for ventilation probably effect the flow separation as well. Results from these studies also demonstrated that when CFD is considered as a tool for mine face-ventilation evaluation and design, expertise in proper use of the codes and turbulence models is very important.

PIV and CFD flow patterns are similar. However, more research is needed for validation of the CFD results. This study will continue, with NIOSH support, for the next year. ■

Acknowledgment

This study was supported by the National Institute of Occupational Safety and Health under Grant No. R01/CCR415822. The authors would like to express their gratitude to Joy Mining Machinery for providing the scale model of the continuous miner used during this study.

References

Brunner, D.J., 1995, "Examples of the application of computational fluid dynamics simulations to mine and tunnel ventilation," *Proceedings*, Seventh U.S. Mining Ventilation Symposium, Lexington, KY.

Gillies, A.D.S., 1982, "Improved coal face ventilation through use of dust scrubber systems leads to greater production efficiency," *Proceedings*, First Mine Ventilation Symposium, Tuscaloosa, AL.

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

Hargreaves, D.M., and Lowndes, I.S., 2001, "Measurement and modeling air flow around a continuous miner in a development heading," *Proceedings*, Seventh International Mine Ventilation Congress, Krakow, Poland, June.

Moloney, K.W., Hargreaves, M.D., Lowndes, I.S., and Pearce, W., 1999, "Validation of computational models of auxiliary ventilation systems with experimental data," *Proceedings*, Eighth U.S. Mine Ventilation Symposium, Rolla, MO, June.

Spalart, P.R., and Allmaras, S.R., 1992, "A one-equation turbulence model for aerodynamic flows," AIAA Paper 92-0439.

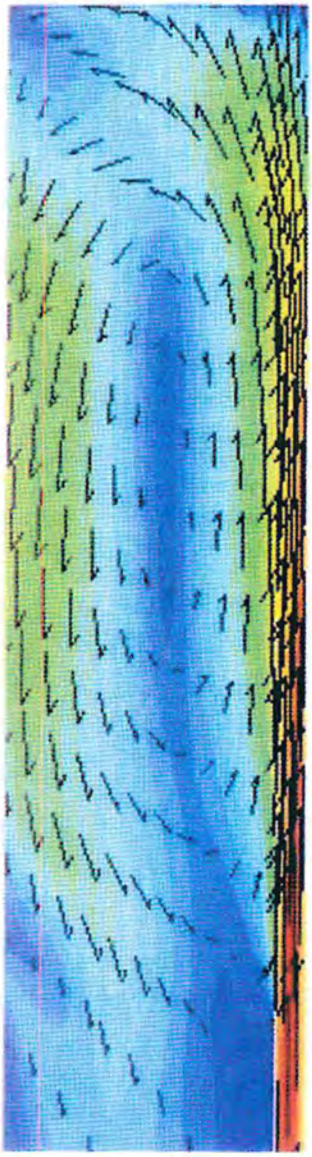
Sullivan, P., and Van Heerden, J., 1993, "The simulation of environmental conditions in continuous miner developments using computational fluid dynamics," *Journal Mine Ventilation Society of South Africa*, January.

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

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

FIGURE 14

CFD results using Fluent and S-A model, Scenario 4.



EVALUATION OF AN EXHAUST FACE VENTILATION SYSTEM FOR 20-FOOT EXTENDED CUT USING PHYSICAL SCALED MODEL

A.M. Wala, J.D. Jacob, T. Watkins, and L. Rangubhotla

ABSTRACT

The face ventilation plans for extended cut must be submit by the Coal Mining Company to the Mine Safety and Health Administration (MSHA) for evaluation and approval. This

evaluation process could be enhanced by having the opportunity to perform an experimental and a numerical study of the proposed ventilation arrangements. These studies could provide a clear determination and adequateness of the ventilation in the face area.

In this paper, the authors will present and discuss the data collected during the experimental study of face ventilation scenarios using a scaled physical model (1:15), equipped with in-house designed and developed Particle Image Velocimetry (PIV) system. The face ventilation arrangements evaluated in this paper are typical of the plans submitted to MSHA for approval by Eastern Kentucky coal companies that utilize continuous mining machines not equipped with scrubbers or mechanical face ventilation enhancing devices.

INTRODUCTION

Coal mine face ventilation plans, submitted to the Mine Safety and Health Administration by the Coal Mining Company for approval, must be evaluated prior to implementation. Critical factors such as the sweep of the face, workers exposure to dust, and compliance with health and safety regulations pertaining to acceptable methane concentration levels must be determined. To be able to evaluate these factors properly, new sophisticated engineering tools (methods) are needed. First is an experimental method using full or scaled physical modeling. However, a full scale modeling is very expensive. Second is a Computational Fluid Dynamics (CFD) technique, which can be potentially applied. This method still required more validations to be able to be utilized with confidence [5], [8], and [9].

The CFD is a new and rapidly developing numerical technique by which complex fluid flow problems can be solved on a digital computer. This technique can be an attractive approach to predict, evaluate, and design proper mine face ventilation systems. However, to develop confidence in the accuracy of CFD the verification between numerical and experimental results are required for validation purposes. During the last two and half years such a validation study, funded by NIOSH Grant #R01 CCR415822, are being carried out in the Department of Mining Engineering, University of Kentucky [8], [9].

As the part of this project the authors designed and developed a scaled physical model of the face ventilation systems (see Figure 1). This model has been shown to be valid/acceptable tool for a scaled study of the face ventilation systems, by validation test performed in the full-scale testing facilities at the NIOSH Pittsburgh Research Center, in June 2002.

Figure 1. The physical model of the mine face area.

Figure 2 shows measured two-dimensional flow distribution (velocity vectors) in scaled and full size testing gallery, for comparison purposes. After this encouraging result the authors are confident that scaled physical models could be used for validation of the face ventilation plans and help in the approval process.

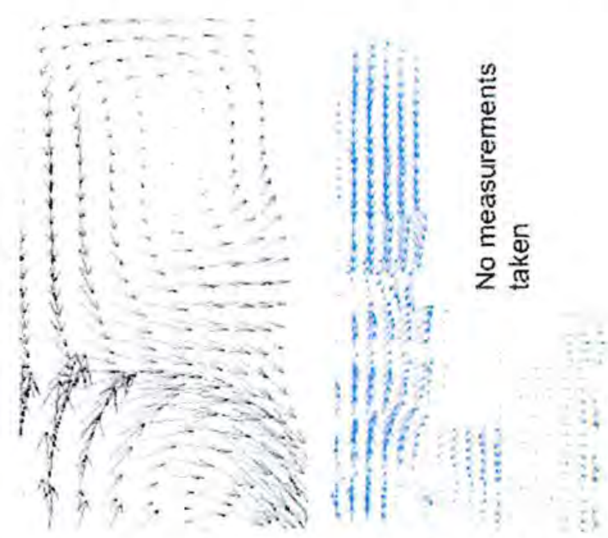


Figure 2. Comparison between scaled (model on left) and full-scale (NIOSH on right) flows.

NEED FOR FACE VENTILATION PLANS VALIDATION TOOLS

Blowing face ventilation systems with scrubber-equipped continuous mining machine are commonly used in the U.S. underground coal-mines for extended cuts. This type of the face ventilation system has shown be the most efficient if all components of the system are working properly. However, in cases of scrubber failure, continuous mining machine not being equipped with scrubber or with other mechanical face ventilation enhancing devices, the recommended blowing ventilation system must be replaced by the exhaust system.

Usually, when circumstances like those described above occur, the typical plans for exhaust face ventilation are submitted to MSHA for approval, see Figure 3. The characteristics parameters of these plans are as follows: (1) exhausting face ventilation systems; (2) cut depths of 20 feet; (3) the number of box (sump) cuts and slab cuts (cutting sequences) range from 2 to 4, (4) a line curtain air quantity of 2.1 m3/s (4,300 cfm), and (5) the cut one will always be on the curtain side.

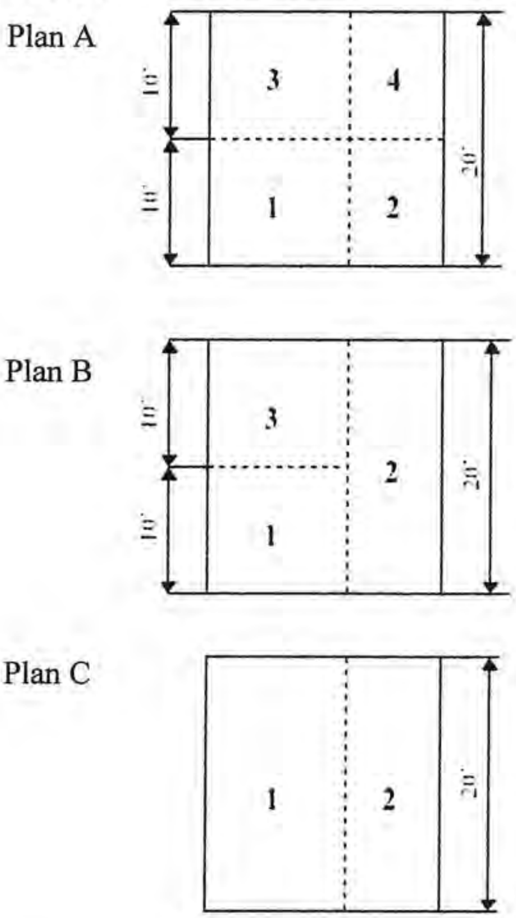


Figure 3. Three typical exhaust face ventilation arrangements for 6.1 m (20 ft) deep cuts.

Applying Plan "A" the authors have attempted to demonstrate how the experimental study, using scaled model equipped with the Particle Image Velocimetry system, can be useful to determine the flow distribution and evaluate control devices used for ventilation enhancements in the face area.

SCALED PHYSICAL MODEL FOR FACE VENTILATION PLANS VALIDATION

Description of the Scaled Physical Model

A 1:15 scaled model of a 2.1 m (7 ft) high, 6.1 m (20 ft) wide mine face area was designed and built out of transparent Plexiglas (see Figure 1). Scaling of the geometry was performed using dimensional analysis based on the Reynolds Number, Re=UL/v, where U and L are the velocity at the brattice line mouth and width of the inflow (distance between the brattice line and entry rib), respectively. The model consists of three significant parts. First, the middle section, which represents an ordinary mining entry with a partition (brattice line), divides the entry into a narrow 2 feet wide intake and an 18 feet wide return airways. Second is the working section, which represents a face area, where movable Plexiglas walls can be used to replicate many different ventilation configurations such as box cut or slab cut arrangements with brattice setback distances varying from zero to 18.3 m (60 ft). Third is the intake/return section, where the ventilation system can be arranged as either blowing or exhausting by changing the fan location. During this particular study, the model was setup as an exhaust ventilation system with a setback distance of 6.1 m (20 ft), see Figure 3.

Description of the Particle Image Velocimetry System

The PIV technique was proposed for measurement of instantaneous velocity fields of the time dependent flows in the scaled model. PIV is an optical technique for measuring two velocity components over an extended two-dimensional area. Due to relatively high air velocity in the model, a pulse laser system was required. To perform these measurements, tracer particles must be added to the flow field; highly reflective ~5 μm talc particles were used in the experiments discussed herein. The laser-light sheet illuminates these particles twice within a short time-interval. The reflected (scattered) light from each laser pulse is sequentially imaged by a CCD camera, and recorded digitally to a computer using a frame grabber. To create one averaged velocity field for a particular scenario, approximately fifty pairs of instantaneous images of velocity fields are recorded for digital analysis.

The schematic of the integrated PIV system used in this study, seen in Figure 4, is comprised of the following components: (1) laser and light sheet optics, (2) image capture system, (3) synchronizer, (4) smoke particle generator, and (5) image analysis and display. The laser is a double pulsed 50 mJ Nd:YAG laser and talc serves the tracer particles whose motions are recorded using a Kodak Megaplus 10-bit CCD camera. Images are stored to a PC using an Epix PIXCI-D digital frame grabber. The frame grabber/camera combination has a double trigger feature that allows two images to be acquired back-to-back within an extremely small time (~10 μs). Timing is controlled via a Taitech DG-100 timing control unit, which is monitored using an oscilloscope. This system is discussed in more in Turner, Wala, and Jacob paper [7].

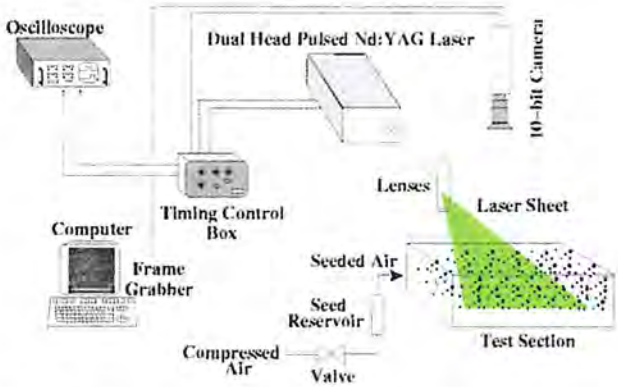


Figure 4. Schematic of the integrated PIV system.

Experimental Data from Scale Modeling

By utilizing the physical model of the face ventilation system (described above) numbers of different mining scenarios, representing coal mining process, using continuous miners with exhaust face ventilation system for 6.1 m (20 ft) depth extended cuts, were tested. For this particular study, as mentioned before, the 4 cut sequence mining operation was adopted.

▪ SCENARIO #1 - "NO EQUIPMENT IN THE FACE AREA"

To be able to show the general flow behavior (distribution) in the face area for the exhaust ventilation system, flow measurements were taken at the middle plane of the entry with no equipment. Figure 5a and 5b shows the velocity vectors in the face area just after the first cut and third cut were completed, respectively. As can be seen from these figures, most of the air is turning immediately back behind the curtain and only a limited amount is flowing toward the face. Nonetheless, whatever is getting into the face area creates large vorticity.

▪ SCENARIO #2 – "EQUIPMENT IN THE FACE AREA"

A scaled model of a continuous miner, simulating Joy Model 14 CM/10, was placed in the face area. Four different locations for the continuous miner, depending on cutting sequence, were investigated. These locations were as follows:

- halfway of the number 3 cut,
- end of the number 3 cut,
- beginning of the number 4 cut, and
- an end of the number 4 cut.

For all four arrangements the line curtain setback was 6.1 m (20 ft) and carries quantity of air equal 2.1m^3/s (4,300 cfm). Figures 6, 7, 8, and 9 shows the velocity vectors measured at the plane above the continuous miner, respectively, to the list above. Again, as can be see from all these figures, only a limited amount of air reach the location were is needed for gas dilution and dust removal. This means that at the face where the cutting head of the continuous miner is located there is a lack of air. Therefore, in such a case there is a question what can be done or what is being done to improve these situations. In the last fifteen years the National Institute of Occupational Safety and Health (NIOSH), and previously by the US Bureau of Mines (USBM) [1], [3], and [6], has conducted extensive research to determine and develop the most effective ventilation schemes for extended cut mining methods. Several ventilation enhancing devices including; (1) continuous miners with machine-mounted scrubbers, (2) extendable line curtains/flexible tubings, (3) jet fans, (4) sprayfans, and (5) a combination of these devices have been designed, tested and implemented [2], [3].

▪ SCENARIO #3 - "EQUIPMENT WITH VENTILATION ENCHANCING CONTROL DEVICE"

To be able to show changes/enhancement in the flow distribution in the face area for above tested scenarios, the authors applied the well known practice of extendable curtain, see Figure 10. The mining arrangements for this scenario are almost identical as for Scenario #2, with exception that simple extendable curtain forces more air to cross the cutting head of the continuous miner. The following four figures 11, 12, 13, and 14 shows the velocity vectors (flow distribution) for arrangements as described in Scenario #2 with the extendable curtain in place.

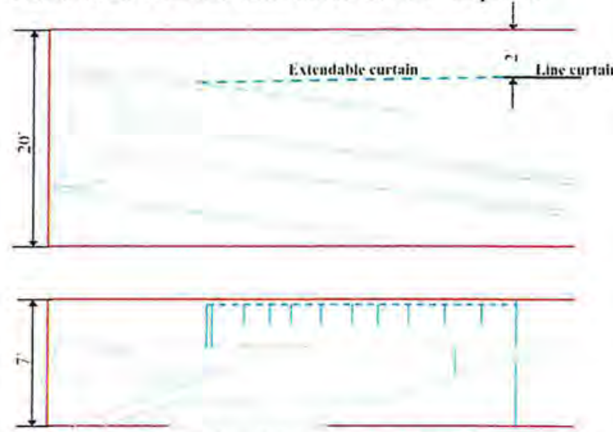


Figure 10. Schematic of the miner model with extendable curtain.

CONCLUSIONS

1. According to the objective of this paper, the authors using the existing scaled model of the face ventilation systems were able to demonstrate how useful this model can be for validation of the face ventilation plans and help in approval process.

2. Based on this study the performance of the exhaust face

ventilation system under Plan "A", (see Figure 3), for 4 cut sequence scenario with 20-foot exhaust curtain setback is not very promising. Some mines have obtained MSHA approval for such scenario. None of these mines have significant methane gas to be controlled.

3. The last part of the study shows how using relatively simple and inexpensive type of ventilation enhancement, by applying 10 feet extendable curtain, the ventilation in the face area could be improved.

ACKNOWLEDGEMENT

This study was supported by the National Institute of Occupational Safety and Health (NIOSH), under Grant No. R01/CCR415822. The authors would like to express their gratitude to Joy Mining Machinery, Inc. for providing the scale model of the continuous miner used during this study.

REFERENCES

1. Colinet, J.F., Jankowski, R.A., 1994, "Dust Control Considerations for Deep-Cut Mining When Utilizing Exhaust Ventilation and a Scrubber," USBM Report of Investigations, RI9615.

2. Divers, F., Volkwein, J.C., 1987, "Guidelines for Choosing Face Ventilation System," Coal Mining, October.

3. Goodman, G.V.R., Taylor, C.D., and Divers, E.D., 1990, "Ventilation Schemes Permit Deep Advance," Coal, October.

4. Hargreaves, D.M., Lowndes, I.S., 2001," Measurement and Modeling Air Flow Around a Continuous Miner in a Development Heading," Proceedings of the 7th International Mine Ventilation Congress, Krakow, Poland, June.

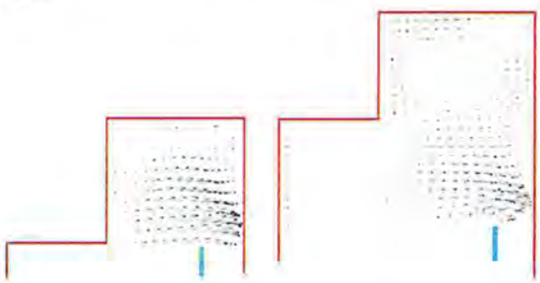
5. Moloney, K.W., Lowndes, I.S., 1999 , "Comparison of Measured Underground Air Velocities and Air Flows Simulated by Computational Fluid Dynamics," Transactions of the Institution of Mining and Metallurgy, 108 May-August.

6. Page, S.J., Mal, T., and Volkwein, J.C., 1994, " Unique Water Spray System Improves Exhaust Face Ventilation and Reduces Exposure to Respirable Dust on Continuous Miners in Laboratory Tests," Journal of the Ventilation Society of South Africa, October.

7. Turner, D., Wala, A.M., and Jacob, J.D., 2002, "Particle Image Velocimetry (PIV) Used for Mine Face Ventilation Study," Proceedings of the North American/Ninth US Mine Ventilation Symposium," Kingston, Canada, June.

8. Wala, A.M., Stoltz, J.R., and Jacob, J.D., 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.

9. Wala, A.M., Turner, D., and Jacob, J.D., 2002, "Experimental Study of Mine Face Ventilation System for Validation of Numerical Models," Proceedings of the North American/Ninth US Mine Ventilation Symposium," Kingston, Canada, June.



(a) (b)

Figure 5. Velocity field with no miner. (a) 1st cut (b) 3rd cut.

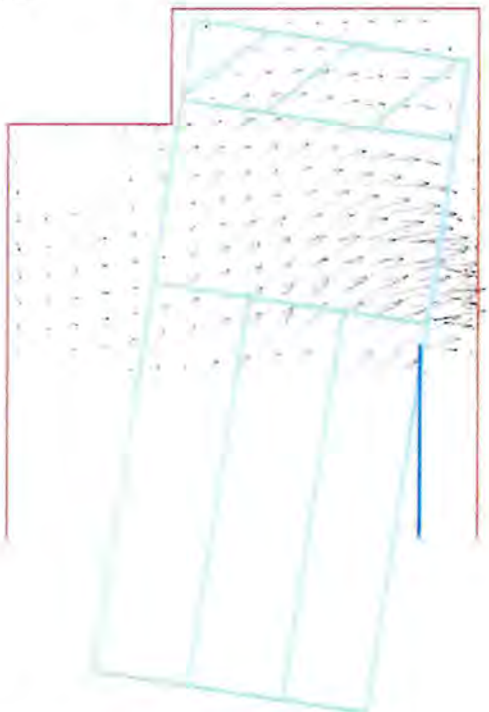


Figure 6: Halfway of the number 3 cut.

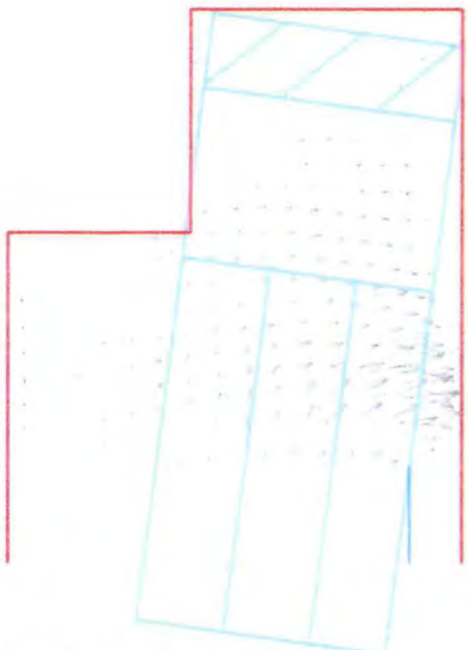


Figure 7: End of the number 3 cut.

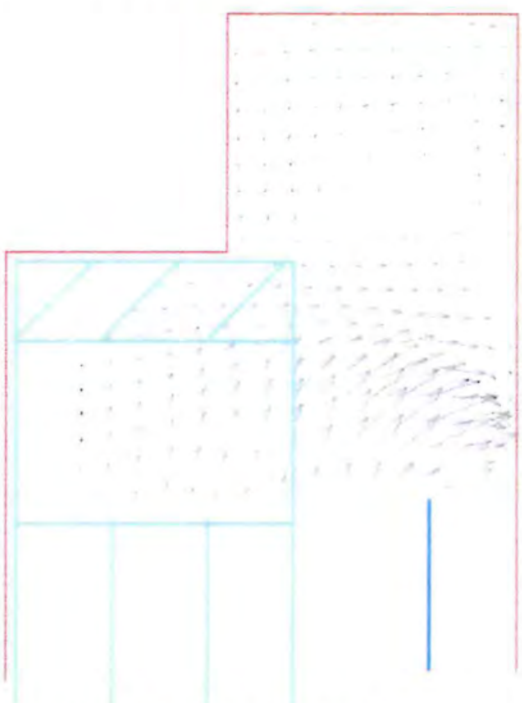


Figure 8: Start of the number 4 cut.

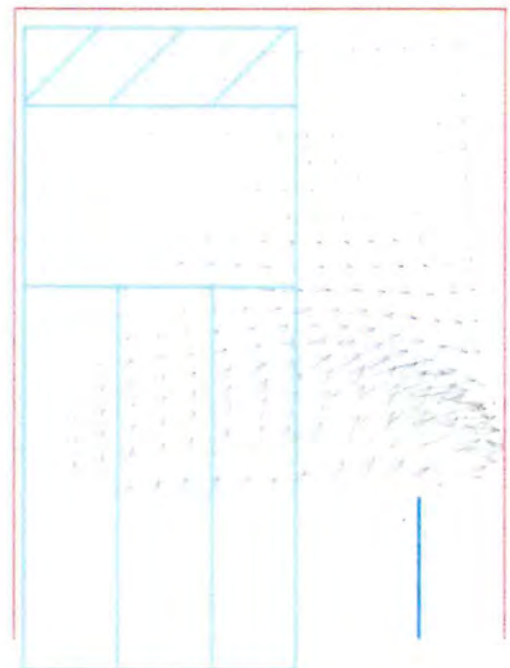


Figure 9: End of the number 4 cut.

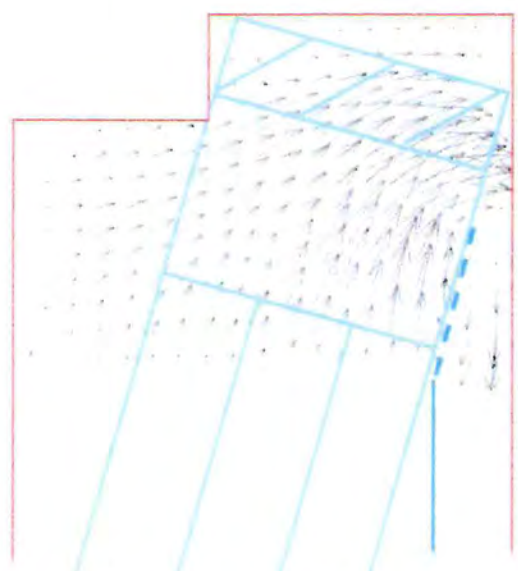


Figure 11: Start of the number 3 cut with extendable curtatin.

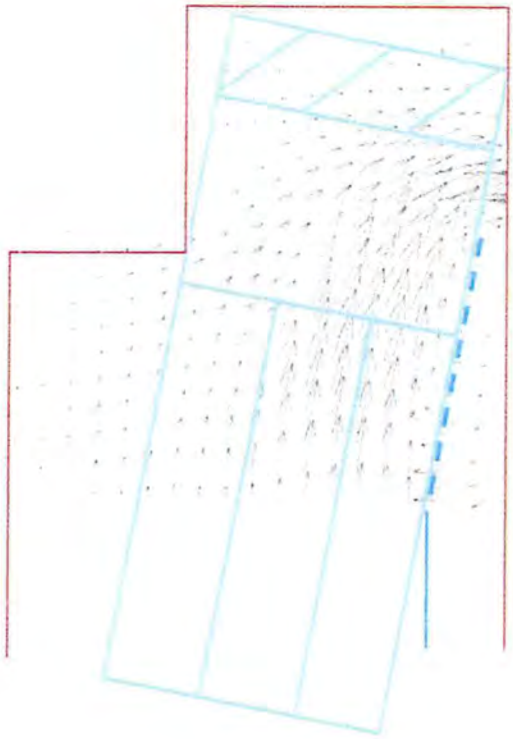


Figure 12: End of the number 3 cut with extendable curtain.

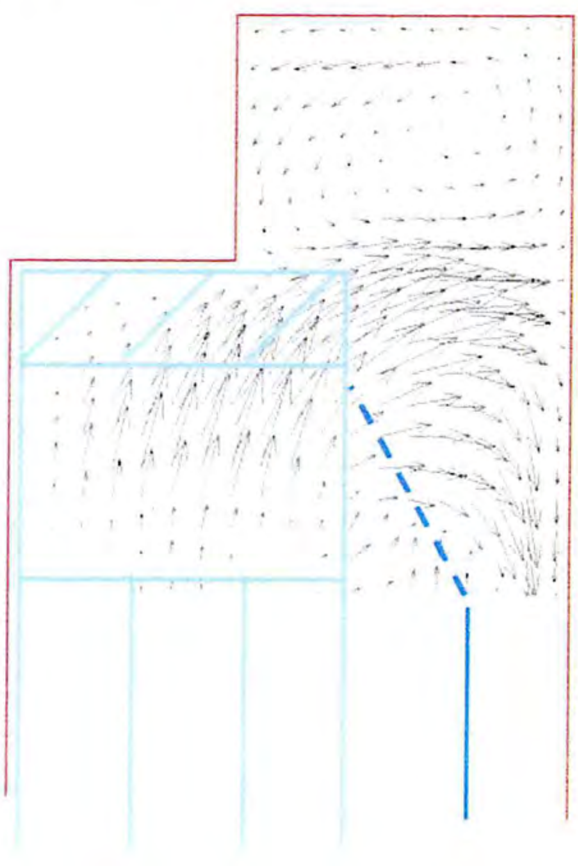


Figure 13: Start of number 4 cut with extendable curtain.

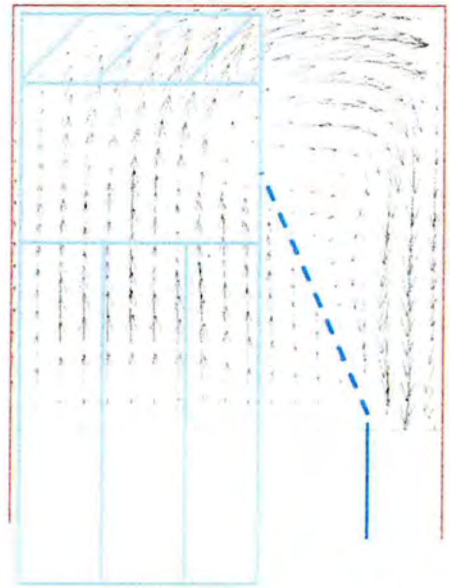


Figure 14: End of the number 4 cut with extendable curtain.

How Scrubbers Help Ventilate the Face During Deep Cut Mining with a Blowing Curtain

A.M. Wala
Department of Mining Engineering, University of Kentucky, Lexington, KY, U.S.A.

J.D. Jacob, G. Huang, J. Brown and L. Rangubhotla
Department of Mechanical Engineering, University of Kentucky, Lexington, KY, U.S.A.

ABSTRACT: In the Department of Mining Engineering at the University of Kentucky a comprehensive validation study of current Computational Fluid Dynamics (CFD) tools against mining-related benchmark experiments has been conducted. A wide array of ventilation schemes and mining configurations has been considered. In this paper, the authors will present, compare, and discuss the experimental and numerical data collected during the part of the study dedicated to the effect of machine-mounted dust scrubber on the performance of a face ventilation system for deep cut mining. These scrubbers, whose primary purpose is 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. Based on studies carried out by the former U.S. Bureau of Mines, NIOSH, MSHA and industry, the recommended ratio between the scrubber airflow $\mathbf{Q_s}$ and the ventilation airflow $\mathbf{Q_{in}}$, should be equal to one. The authors will demonstrate and discuss the flow behaviors in the face area for the different values of the $\mathbf{Q_s/Q_{in}}$ ratios.

1 BACKGROUND

During the past twenty years, since the advent of remote controlled continuous miners and deep cut (extended advance) for room and pillar mining operations (Wellman, 1981), the safety and productivity of mining operations has increased significantly. However, this high production presents environmental problems in that, per unit time, more dust is generated and higher levels of methane are liberated at the face during coal extraction. These conditions require the supply of more air into the face of the excavated entry for safe dilution. One of the ventilation techniques being used in conjunction with this modern mining method (Goodman et al., 1990) is the blowing curtain in conjunction with a machine-mounted dust scrubber. Implementation of this technique has a significant influence on the airflow patterns in the entry working area and the design of their air handling capacities.

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

The design of a balanced ventilation system for the above-described scenarios requires considering the system in three rather than two dimensions. With Computational Fluid Dynamics (CFD) many design alternatives can be explored in a relatively short period, and detailed three-dimensional results can be viewed graphically. This graphical type of information concerning airflow, gas, and dust distribution established by CFD are readily rationalized, and could lead to confidence in the final design of a mine face ventilation system. However, to develop this confidence, extensive validation studies are required involving comparisons between physical and numerical (CFD) modeling (Hargreaves and Lownsdes, 2001).

2 INTRODUCTION

During the last three years a validation study, funded by NIOSH Grant # R01 CCR415822, has been 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 performed the following tasks:

- Designed and built a scaled physical model of selected face ventilation systems (see Wala et al., 2003),
- Performed a series of measurements using Particle Image Velocimetry (PIV) (Turner et al., 2002), to determine the velocity fields in the model,
- Developed three-dimensional numerical models of the tested systems and performed computer simulations using numerous CFD codes, and
- Compared experimental and numerical data for the purpose of CFD code validation.

One of the scenarios being tested for validation purposes was to ventilate equipment-free entry for cutting depth of approximately 18.3 m (60 ft). According to the mining plane this 18.3 m (60 ft) deep cut should be completed in a four 9.2 m (30 ft) cuts (passes), using blowing curtain ventilation system, see Figure 1. The results from the simulation study, using a scaled physical model, are shown in Figures 2, 3, 4 and 5 (Wala et al., 2000, 2002 and 2003). These figures increased an understanding of the flow behavior in cases of mining a deep cut using a blowing curtain for ventilation.

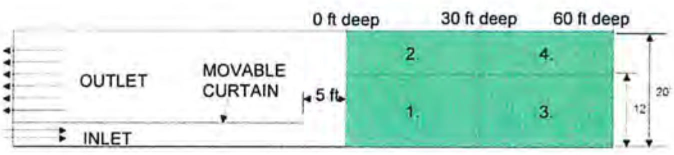


Figure 1. Test section geometry and the scenarios examined.

The major characteristic of the flow pattern, specifically for the box cutting sequence, is that 70 to 80% of the air leaving the end of the blowing curtain almost immediately separates from the rib and as a primary flow turns back toward the return. This separation is driven by an adverse pressure gradient which builds up as the width of the entry decreases. The rest of the curtain's airflow, approximately 20 to 30%, called the secondary flow, continues to flow toward the immediate face area in the opposite direction. As can be seen in general, the ventilation air delivered by the blowing curtain does not flow into the immediate face area.

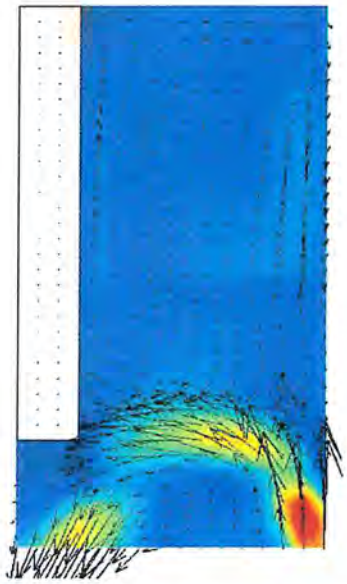


Figure 2. PIV velocity magnitude and vectors for a 30 ft deep cut with a slab.

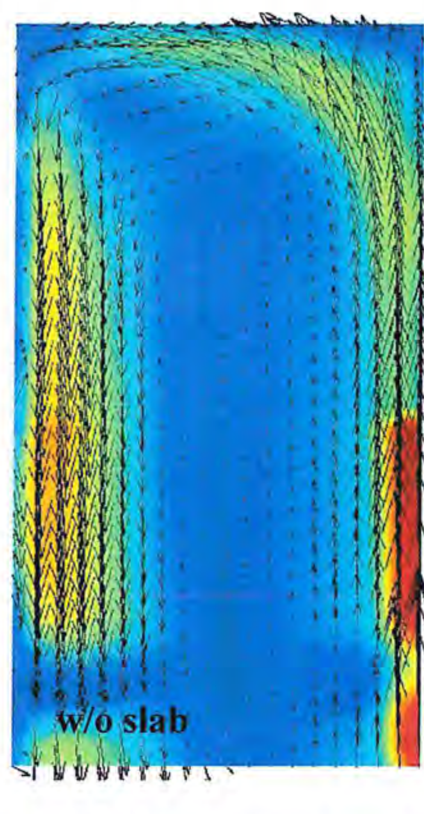


Figure 3. PIV velocity magnitude and vectors for a 30 ft deep cut without a slab.

During the validation studies the authors tested over a hundred scenarios for different combinations of such parameters as curtain setback, intake air quantity delivered by the curtain, tight-rib distance (curtain distance from the rib), and mining sequence. The data collected during these studies allows direct conclusions concerning the flow behavior.

The authors presumed that, until that time, the flow patterns for these mining scenarios had not been fully understood because of limited opportunity to monitor the flow patterns during full-scale experimental studies performed underground or on the surface. The only sources of information found by authors concerning the similar flow behavior as shown in Figures 2-5 were recognized and described by Luxner (1969) and Taylor et al., (1997).

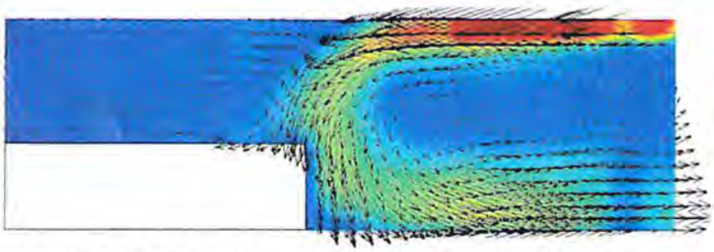


Figure 4. PIV velocity magnitude and vectors for a 60 ft deep cut with a slab.

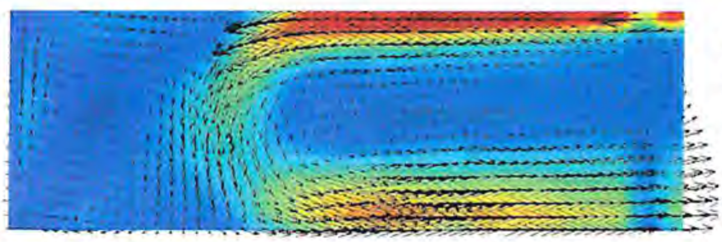


Figure 5. PIV velocity magnitude and vectors for a 60 ft deep cut without a slab.

To ensure that scaled model test results interpret conditions reliably in the underground face area the cooperative validation testing has been performed by the Mining Engineering Department, University of Kentucky and NIOSH, using full-scale testing gallery at the NIOSH Pittsburgh Research Center. These studies proved that a 1:15 scaled laboratory model used for the project is a valid tool and realistically simulated underground conditions for the face ventilation systems. Observation showed the flow patterns for one of the scenario tested using laboratory scaled model and at the full-scale model as very similar, including recirculation.

As mentioned before, since remote controlled continuous miners were introduced, deep cut mining was an opportunity for mine operators to increase productivity. However, the presence of airborne coal dust in the face area was a major obstruction. This coal dust can cause poor visibility around the cutting head and can expose the miners to high levels of respirable dust. A practical means of removing this dust is with a dust-collecting scrubber mounted directly on the mining machine (U.S. Bureau of Mines, 1983). When the scrubbers were added to the face ventilation system of coal mines, the existing arrangements of the face ventilation systems were altered to accommodate these scrubbers.

Many studies concerning face ventilation with scrubbers were carried out in the United States in the last twenty years. The authors would like to mention some of them, which have contributed greatly to this topic; Gillies (1982), Clouse (1982), Volkwein and Thimons (1985), Jayaraman et al. (1992), Taylor et al. (1997), Thimons et a1. (1999), Kissell (2003). Most of the researchers, followed the Luxner comments concerning higher effectiveness of the blowing system versus the exhaust system, recommended the blowing curtain as the best suited ventilation system for scrubber applications, particularly because the free jet of high initial momentum leaving the end of the line curtain should penetrate into the immediate face area sweep the face and flow into the return.

This is true, according to Figures 2-5, as long as the entry width is more than 16 foot, as was discovered during this study (Wala et al., 2003). This means that only during the slab cutting scenario does the advantage of the blowing curtain prevail. For the rest of the mining scenarios the ventilation air does not like to flow to the immediate face area where it is mostly needed in the gassy mines for methane dilution.

The effects of the machine-mounted scrubber system on levels of methane and respirable dust were tested first by Ingersoll Rand Research Inc., (Clouse, 1982). These studies have shown that properly designed machine–mounted scrubber systems can allow curtain setbacks of 12.2 m (40 ft), while still controlling respirable dust and methane levels. At the same time, the machine-mounted scrubber, whose primary role was to collect respirable dust at the face (dust source location), was found to be also an air-moving device (ventilation device).

Based on the Ingersoll Rand studies, MSHA recommended a ratio between the scrubber flow $\mathbf{Q_s}$ and the intake flow $\mathbf{Q_{in}}$ of close to one. A study performed by Jayaraman, 1992, supports these finding, however, latest studies performed by Taylor at el., 1997 and by Thimons at el., 1999 show that the relations between face ventilation performance and scrubber/ventilation flow ratios are more complicated than was previously thought.

The authors, based on the results from the experimental studies using physical and numerical modeling, will demonstrate and discuss the flow behavior in the face area for the varied $\mathbf{Q_s/Q_{in}}$ ratios.

3 EXPERIMENTAL STUDIES

In order to study the effect of the scrubber on the face ventilation system, a scaled physical model of such a system was constructed in the Ventilation Laboratory at the Mining Engineering Department, University of Kentucky.

Figure 6. Simplified scrubber model in test section.

Figure 7. Machine mounted scrubber model in test section.

3.1 Description of the Physical Model

3.1.1 Entry Model with Fans

The analysis of face airflow patterns was undertaken using 1:15 physically scaled model. This scaled model of a typical underground room and pillar coal mine with an entry of 2.1 m (7 ft) high and 6.1 m (20 ft) wide was designed and built out of transparent Plexiglas. Scaling of the entry flow rate was performed using dimensional analysis based on the Reynolds Number, Re=UL/ν, where U and L are the velocity at the brattice line mouth and width of the inflow (distance between the brattice line and entry rib), respectively. The model consists of three significant parts:

- The working section, which represents a face area, where movable Plexiglas walls can be used to replicate many different ventilation configurations such as box-cut or slab-cut arrangements with brattice setback distances varying from zero to 18.3 m (60 ft).
- The middle section, which represents an ordinary mining entry with a partition (brattice line/curtain), divides the entry into a narrow 0.6 m (2 ft) wide intake and an 5.5 m (18 ft) wide return airways.
- The intake/return section, where the ventilation system can be arranged as either blowing or exhausting by changing the fan location.

The primary airflow through the entry was drawn by a main fan in series with a smaller axial speed controllable auxiliary fan. The maximum available airflow induced through the model corresponds to the 3.0 m^3/s (6,500 CFM) for the underground conditions.

3.1.2 Scrubber Model

The scrubber system for this modeling study did not exactly simulate the underground scrubber arrangements. Since the scrubber system could not be modeled exactly as it appears in a mine due to limited space, a simplified model of the scrubber was simulated using open ended 2 inch diameter PVC pipe. The airflow though this pipe was drawn by a suction system. The maximum airflow induced by the scrubber system corresponds to 1.4 m^3/s (3,000 CFM) for the underground conditions.

For validation purposes two different arrangements with the scrubber were tested. First, as shown in Figure 6, a cylindrical "scrubber" was simulated by the straight PVC pipe placed in the middle on the entry floor, 1.5 m (5 ft) from the face. Second, as shown in Figure 7, the same PVC pipe was placed on the top of a continuous miner model, corresponding to the Joy Model 14CM Continuous Miner, located in the face area.

The model was arranged to simulate two critical ventilation scenarios as follows:

- SCENARIO #1
  The end of the box cut during the first mining pass (sequence), with a setback distance being 10.6 m (35 ft) (see Figure 1).
- SCENARIO #2
  The end of the box cut during the third mining pass (sequence), with a setback distance being 18.3 m (60 ft) (see Figure 1).

3.2 Particle Image Velocimetry (PIV)

PIV was used to measure instantaneous velocity fields of the time dependent flows in this scaled model. PIV is a non-intrusive, optical technique for the measurement of planar flow velocities. The technique measures the velocity of a fluid indirectly by means of the measurement of the velocity of tracer particles within the flow. A plane of the flow is illuminated twice within a short time interval, and the images of the illuminated particles are digitally recorded on two separate image frames of a CCD camera (Turner et al., 2002). Fourier analysis is used to extract the planar particle velocity information from the image pairs. The experimental set-up of the PIV measurement system consists of four sub-systems: (1) Seeding, (2) Illumination, (3) Recording, and (4) Evaluation. The integrated PIV system used in this study is comprised of the following components: (1) laser and light sheet optics, (2) image recording system, (3) synchronizer, (4) particle generator, and (5) image analysis and display, (Turner et at., 2002).

4 RESULTS AND DISCUSSION

4.1 Experimental Data from Scale Modeling

Based on the results collected during the flow distribution study in the equipment-free entry, shown in Figures 2-5, it was found that the worst ventilation conditions occur during the box cutting sequences (see Figures 2 and 4). Using the physical model of the face ventilation system, these two critical ventilation scenarios, described above, represent possible mining arrangements for a 18.3 m (60 ft) deep cut in four passes.

The performance of the face ventilation for theses two extended cut mining scenarios was a goal for this validation study. The geometry of the entry examined was 2.1m (7 ft) high and 6.1 m (20 ft) wide. The maximum intake air in volume of 3.0 m^3/s (6,500 CFM) was provided to the face area by means of a blowing curtain.

Figures 10 and 11 show the flow streamlines from PIV measurements at the horizontal middle plane of the model for SCENARIO #1 and SCENARIO #2, with and without the scrubber, respectively. As previously mentioned, the scrubber for this particular case was simulated as a simple pipe located on the floor of the empty entry. The general observation from these two tests are that scrubbers help to bring the air into the immediate face area during the box cut mining sequences, whereas it would not penetrate to the face otherwise. This makes the 18.8 m (60 ft) deep cut mining possible.

We were also interested in determining the relationship between the performance of the face ventilation system and the flow ratio between the intake and scrubber flows. Figures 8a, 8b, and 8c show the flow streamlines at the horizontal middle plane of the entry for three different flows ratios Q_s/Q_{in} of 0.14, 0.34, and 0.53, respectively. From a methane removal point of view for this equipment empty entry scenario, the eddies indicate poor ventilation of the face due to excessive recirculation.

Figures 9a, and 9b shows similar flow streamlines as shown in Figure 8b and 8c, except during these tests the continuous miner model was present in the face area. The measurements were taken at the horizontal plane above the miner. By comparing these figures it can be seen that equipment present in the face creates additional obstructions to the flow and decreases the effectiveness of the ventilation system.

4.2 Numerical Modeling

The numerical simulation for these two scenarios described above was performed using CFX commercial CFD (Computational Fluid Dynamics) software package. Based on the intensive validation studies performed as a part of the entire project, the CFX software package along with the SST (Shear Stress Transport) turbulence model was selected for its ability to accurately predict separation due to adverse pressure gradients, as opposed to the k-ε model.

PIV results from the experimental studies of SCENARIO #1 with the cylindrical scrubber (Figures 6) and continuous miner with on-board mounted scrubber (Figure 7), served as a baseline for the numerical simulation validation. In these studies, the scrubber mass flow ratio was normalized by the mass flow at the intake flow (behind the curtain). Thus, the results presented in Figures 14 illustrate predicted flow fields for the cylindrical scrubber at the scrubber flow ratios of 0.2, 0.3, 0.4, 0.7 and 1.0, respectively. The boundary condition for the scrubber is set as a constant velocity outlet which is released outside the original geometry. These boundary conditions were set on the scaled model during the experimentations. The grid density was based on the coarse grids tested for the previous simulation exercises.

4.3 Discussion

In comparing the numerical (CFD) results with the experimental (PIV) results, one will see good similarity in the flow behavior in the face area. However, some flow pattern discrepancies can be noticed. The experimental results show the location of the separation point to be more sensitive to smaller values of the scrubber mass flow ratio. This flow behavior could be explained by existence of the unsteady flow along the wall (rib). However, once the scrubber mass flow ratio reaches a level near 1.0 the separation is removed and the flow carries to the immediate face area. This is notable since most scrubbers in actual mining applications are required to have a mass flow ratio equivalent to 1.0. Also, it can be seen that as the flow ratio approaches one the recirculation (eddies) in the immediate face area increased.

Figure 11 show simulated 3-D flow streamlines in the face area when a simplified virtual continuous miner with scrubber was inserted into the entry for the 9.1 m (30 ft) box cut scenario. The miner was equipped with a virtual scrubber which is incorporated into the geometry of the mine such that the scrubber's inlets are placed on each side of the miner and the exhaust is placed at its rear. Unlike the previous scrubber, this scrubber was modeled with the "fan" UDF (User Defined Function). This method of modeling allows the user to put a pressure gradient condition on a plane within the scrubber's duct essentially modeling a blower mounted to the miner. The main difference between this method and the previously used method is that the scrubber's exhaust is released within the mine as opposed to outside. This better represents the scenario of a miner with on-board scrubber and would be very difficult for an experimentalist to implement into a scaled model due to flow restrictions on fans small enough to fit these scaled geometries. Figure XX illustrates the resulting flow over the miner without the use of a scrubber. The effect of the scrubber can clearly be seen by comparing Figures 2-5 with Figures 8-11.

5 CONCLUSIONS

- These scrubbers were shown to be very useful tools in moving air into the immediate face area where ventilation is poor, especially during the box cutting sequences.
- Scrubbers introduced as dust collection devices help to bring the air to the immediate face area and should be termed face ventilation devices, as well. Without a dust scrubber, the blowing curtain is ineffective for controlling respirable dust and methane as can be seen from Figures 2, 3, and 4.
- With scrubbers, more efficient ventilation can be obtained in the face areas without the need for miners to extend ventilation controls inby permanent roof supports.
- The flow ratio between intake and scrubber is an extremely important factor, which could be controlled for optimization of the face ventilation system.

6 REFERENCES

Clouse, B.E., 1982, "Assess Feasibility of Machine Mounted Scrubber System for Ventilation Coal Mine Working Face. Phase 1, Interim Report," Ingersoll Rand Reaserch Co., New Jersey.

Gillies, A.D.S., 1982, "Improvement Coal Face Ventilation Trhrough Use of Dust Scrubber Systems Leads to Greater Production Efficiency." Proceedings, First Mine Ventilation Symposium, Tuscaloosa, AL.

Goodman, G.V.R., Taylor, C.D., and Divers, E.D., 1990, "Ventilation Schemes Permit Deep Advance," Coal, October.

Goodman, G.V.R., Taylor, C.D., Colinet, J.F., and Thimons, E.D., 2000, "Research by NIOSH for Controlling Respirable Dust and Methane Gas on Continuous Mine Faces," Proceedings: Seventh International Mine Ventilation Congress, Krakow, Poland.

Thimons, E.D., Taylor, C.D., and Zimmer, A., 1999, "Ventilating the Box Cut of a Two-Pass 40-ft Extended Cut," Proceedings: Eight US Mine Ventilation Symposium, Kingston, Canada.

"Ventilation Schemes Permit Deep Advance," Coal, October, 2XXX.

Hargreaves, D.M., Lowndes, I.S., 2001, " Measurement and Modeling Air Flow Around a Continuous Miner in a Development Heading," Proceedings, Seventh International Mine Ventilation Congress, Krakow, Poland.

Luxner, J.V., 1969, "Face Ventilation in Underground Bituminous Coal Mines."Bureau of Mines RI 7223.

Taylor, C.D., 2000, "Effect of Water Sprays and Scrubbers Exhaust on Face Methane Concentrations." Proceedings: Seventh International Mining Ventilation Congress, Crakow, Poland.

Taylor, C.D., 1997, "Impact of Unbalanced Intake and Scrubber Flows on Face Methane Concentrations," Proceedings: Seventh International Mine Ventilation Congress, Pittsburgh, PA.

Wala, A.M., Stoltz, J.R., and Hassan, M.I., 2000, " Scale Modeling of a Mine Face Ventilation System for CFD Code Validation" Proceedings: Third International Symposium on Scale Modeling, Nagoya, Japan.

Wala, A.M., Stoltz, J.R., and Jacob, J.D., 2001, " Numerical and Experimental Study of a Mine Face Ventilation System for CFD code Validation," Proceedings Seventh International Mine Ventilation Congress, Krakow, Poland.

Wala, A.M., Jacob, J., Brown, J., and Huang, G., 2003. "New Approaches to Mine-Face Ventilation," Mining Engineering, No. 3, March.

Wellman, T.S., 1981, "Deep Cut Remote Control Mining." Mining Congress Journal." October.

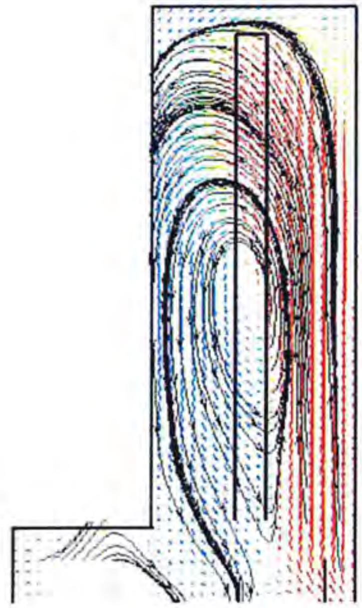


Figure 8b. Streamlines from PIV measurements with the simplified scrubber model for a 35 ft setback with a slab; Q_s/Q_{in}=0.34.

Figure 8a. Streamlines from PIV measurements with the simplified scrubber model for a 35 ft setback with a slab. (a) Q_s/Q_{in} of 0.14.

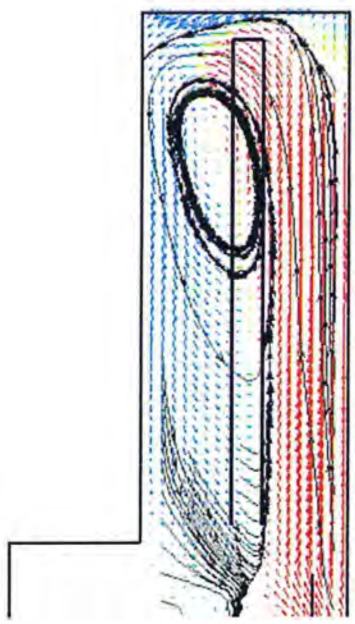


Figure 8c. Streamlines from PIV measurements with the simplified scrubber model for a 35 ft setback with a slab.; Q_s/Q_{in}= 0.53.

Figure 9a. Streamlines from PIV measurements with the machine mounted scrubber model for a 35 ft setback with a slab; Q_s/Q_{in}= 0.30.

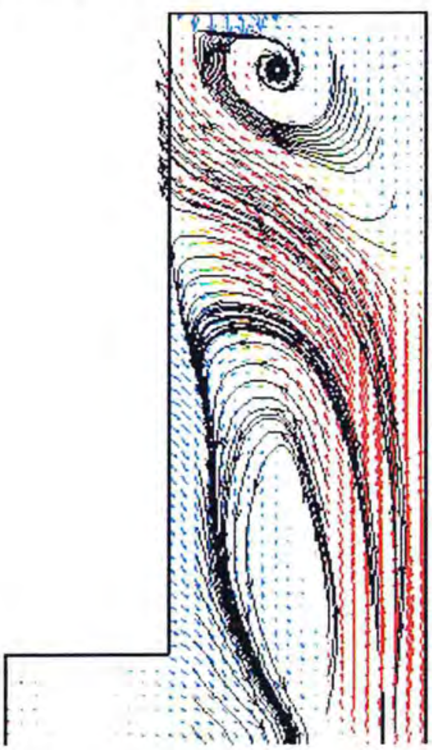


Figure 9b. Streamlines from PIV measurements with the machine mounted scrubber model for a 35 ft setback with a slab; Q_s/Q_{in}= 0.53.

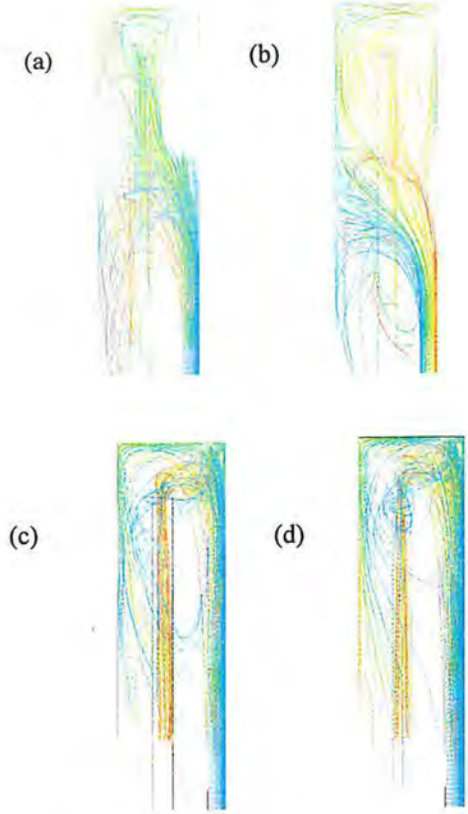


Figure 10a, 10b, 10c, 10d. CFD results for a 30ft deep cut with a simplified scrubber model for a 35 ft setback; (a) Q_s/Q_{in}= 0.20; (b) Q_s/Q_{in}= 0.40; (c) Q_s/Q_{in}= 0.70; (d) Q_s/Q_{in}= 1.0.

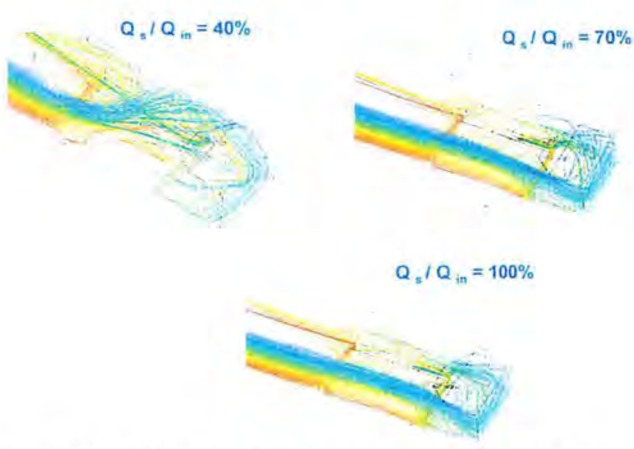


Figure 11a, 11b, 11c. CFD results for a 30ft deep cut with a machine mounted scrubber model; (a) Q_s/Q_{in}= 0.40; (b) Q_s/Q_{in}= 0.70; (c) Q_s/Q_{in}= 1.0.

Prediction of Mine Face Ventilation using Computational Fluid Dynamics

University of Kentucky

June 27, 2002

Introduction

This report includes results from analyses performed using CFD (Computational Fluid Dynamics) for the purpose of predicting the flow within an open face mine. Several scenarios are explored to compare the CFD results to experimental data taken with PIV (Particle Image Velocimetry). Up to this point, only one commercial CFD program, Fluent, has been utilized but other commercial and non-commercial codes should be tested. If a CFD program is found to effectively predict the flow in the mine face, then the program can be used to make advancements in mine ventilation technology. New products could be developed faster and cheaper by minimizing the amount of required experimentation.

There were four major scenarios examined with PIV by Wala and Jacob at the University of Kentucky. These scenarios were then analyzed using the commercial CFD software package, Fluent. The objective was to fully utilize the software's many features to obtain an accurate prediction of the onset separation that occurs during three of the four scenarios. This separation of the flow prevents proper ventilation near the face and allows for methane build-up at cutting face. The build-up of methane at the cutting face increases the probability that an explosion will occur and should be prevented by proper ventilation. If CFD can predict the onset separation that occurs, new ways of preventing the separation can be developed on the computer instead of making expensive prototypes and using expensive test equipment.

1 Background

1.1 Four Mining Scenarios

Four scenarios were examined to simulated the process of coal extraction from a mine face. In all scenarios the geometry of the mining vehicle was excluded for the purpose of simplifying the modeling procedure. The path by which the remote control miners take during normal operation was broken into four main geometries including (a) 30 foot box cut, (b) 30 foot slab cut, (c) 60 foot box cut and (d) 60 foot slab cut. Illustrated in figure1 are the geometries along with the dimensions for the full scale mine face.

1.2 Experimental Data (PIV)

Experiments were performed using PIV to generate two-dimensional plots of velocity-vectors, velocity contours, vorticity contours, etc. These two-dimensional plots were taken in the horizontal plane that is offset from the bottom at a distance equivalent to half of the total height. This plane will be referred to as the mid-plane throughout this report. For more details on the experimental data see the report by Wala[1]. Figure 2 are the velocity-vector plots yielded by PIV for the previously mentioned scenarios.

2 Setup

2.1 Grid Generation

Grid generation for CFD simulations is a very important component of the analysis setup. Two main points were considered while generating grids for each case including the shape of cells and the size of cell near the walls. First, a non-uniform structured grid, composed only of quadrilateral cells, was used for all simulations presented in this report. Second, the size of the grid cell near the wall is calculated based on the value of y^+ recommended for the turbulence model being used. For details on this calculation, see the Fluent User's Guide[2].

2.2 Grid Sensitivity Study

Because the solution of an analysis can be dependent on how the model is discretized, it is proper procedure to seek a grid-independent solution before accepting any results with confidence. A grid-independent solution is achieved by performing a grid-sensitivity study which ensures that mesh, or grid, will no longer affect the solution. A grid-sensitivity study is performed by comparing the results of three differently sized grids. The original grid should be modified by increasing the total number of points while conserving the ratios of the points in each direction. For example, brick A contains 1000 grid cells, or points, with 10 points in the x-direction, 5 in the y-direction and 20 in the z-direction. To create brick B by doubling the total number of cells while conserving the ratios of the cells in each direction, the number of cells in each direction must be multiplied by ${}^3\sqrt{2}$ or $1.25992105 \approx 1.26$. If the results from the three grids compare well, then the solution is considered to be grid-independent.

> *Note that a grid-independent solution has been reached for some of the following results but not for all.*

2.3 Boundary Conditions

Because we were attempting to match our results with PIV results, the boundary conditions of the model were very important. All scaling is done by keep the Reynold's number constant and adjusting the velocity to compensate for the smaller geometry. An inlet velocity of 28 m/s is used to simulate the velocity generated by the seeded fan system on the PIV setup[1]. Next, the density and viscosity are those found for air at room temperature in BASIC FLUID MECHANICS[3], which are 1.20 kg/m^3 and $1.51 \cdot 10^{-5}$ m^2/s, respectively. Also, the roughness on the walls was assumed to be $1 \cdot 10^{-5}$ to represent the plexiglas that was used in the PIV model. Finally, other parameters are specific to the turbulence model used.

There are many different methods to model the turbulence and some ways are more effective than others depending on the application. Also, some turbulence models are more easily converged than others. The Reynolds Average Numerical Simulation (RANS) models are usually faster but are only accurate in certain applications and geometries. Three RANS turbulence models including the Standard $k-\varepsilon$, Spalart-Allmaras and Shear-Stress Transport $k-\omega$ models were used and the results were compared to find the most robust and most accurate model.

Because the $k-\varepsilon$ model allows the use of a standard wall function, it is possible to decrease the total number of grid points by assuming the boundary layer can be predicted with one equation. Fluent recommends a $y^+ \geq 30$ while using the standard wall function, thus the calculated grid size for the cells near the wall is a dimensionless value of $.01722 \approx .035ft$. Next, the boundary condition parameter k and ε are used to represent turbulence kinetic energy and turbulence dissipation rate, respectively. Another important parameter is the turbulent viscosity, μ_t, which we assume to be 10% of the reference viscosity. The relationship between these three parameters is given in the Fluent User's Guide[2] as $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$, where $C_\mu = .09$ is constant. k is given the value of 1.96 m^2/s^2 and ε is calculated to be 229,000 m^2/s^3.

The Spalart-Allmaras and $k-\omega$ SST models have similar grids, however, the boundary condition parameters for turbulence are different. There is only one parameter for the S-A model, which is $\mu_t = 0.1 * \mu_{ref}$. The SST model contains two parameters for modeling turbulence k and ω. k is the same term that was given for the $k-\varepsilon$ model and ω is related with the equation:

$$\omega = \varepsilon/(C_\mu k)$$

3 Results

3.1 Convergence

Because the analysis that was performed in this report implicitly solved a set of segregated partial differential equations, an iterative solution was yielded. When solving equations iteratively, there is usually some imbalance of continuity and momentum equations by which we quantify in this case with the residuals. As the CFD program completes more iterations, the residual plot for the solution should yield a negatively sloped curve. As the curve continues to drop, the slope should approach zero creating a horizontal line. Once the residual plot has flattened for more than 1000 iterations the value of the residual should be checked. If the value of the residual at the flat line is less than 10^{-3}, a value recommended by Fluent, then the solution can be considered converged.

3.2 The $k-\varepsilon$ Model

3.2.1 30 FOOT BOX CUT

A grid-independent solution was found for the 30 foot box cut scenario using the $k-\varepsilon$ model with a standard wall function. Three grid sizes were solved to convergence as shown in figure 3. Comparing these results to the benchmarked PIV results reveals that Fluent's implementation of the $k-\varepsilon$ model is inadequate for correctly predicting the onset separation in this particular case.

3.2.2 30 FOOT SLAB CUT

A grid-independent solution was found for the 30 foot slab cut scenario using the $k-\varepsilon$ model with a standard wall function. Three grid sizes were solved to convergence as shown in figure 4. These results compare very well with the experimental data. One theory to explain this correlation is that the flow does not separate, thus it is very easy to compute. This case does not hold as much importance as the other three.

3.2.3 60 FOOT BOX CUT

A grid-independent solution was found for the 60 foot box cut scenario using the $k-\varepsilon$ model with a standard wall function. Three grid sizes were solved to convergence as shown in figure 5. Fluent did predict a separation of the flow near the same region in which PIV found separation. However, the separation was noticeably further down stream and the velocity seems to dissipate more rapidly than in PIV. This comparison can be considered a mild success because it did predict the separation, but apparent differences between Fluent and PIV prevent us from having confidence in Fluents ability to predict this scenario with accuracy.

3.2.4 60 FOOT SLAB CUT

A grid-independent solution was found for the 60 foot slab cut scenario using the $k-\varepsilon$ model with a standard wall function. Three grid sizes were solved to convergence as shown in figure 6. As no separation is predicted by Fluent, the comparison with experimental data shows no correlation. There, the implementation of the $k-\varepsilon$ model in the Fluent 6.0 package in inadequate for accurately predicting the onset separation found in this scenario.

3.3 The Spalart-Allmaras Model

3.3.1 30 FOOT BOX CUT

Although a grid-independent solution was not found while using the Spalart-Allmaras (S-A) turbulence model with the Fluent 6.0 package to compute the 30 foot box cut scenario, some promising results were discovered and are illustrated in figure 7. In sub-figure (a), the first order upwind scheme was used for a grid containing 1.2 million elements and the solution converged yielding a result that compares very well with PIV results. However, using the same grid and implementing the QUICK scheme, as seen in (b), which should yield a more accurate solution, the residuals begin to diverge. This leads us to believe that the stability of Fluent's implementation of the S-A model is unstable. Once more, as seen in (c), a coarse mesh using 800 thousand grid cells does converge yielding a solution that once again correlates very well with the experimental data. Therefore, for this scenario, we fear that Fluent's implementation of the S-A model is capable of predicting the correct solution, but may be very difficult to converge.

3.3.2 30 FOOT SLAB CUT

When modeling the 30 foot slab cut scenario using the S-A model and two different mesh densities, there were problems with converging the results. As seen in figure 8, only the coarse grid case actually yielded a reduction in the residual of more than $1.0E-3$ and the residual began to slope back up near $20,000$ iterations. Although the results from all three runs look similar to the experimental data, they cannot be trusted to the difficulty involved with converging Fluent's implementation of the S-A model. It should also be noted that this case is believed to be the easiest scenario to solve. Unlike the results from the 30 foot box cut scenario, not even the first order upwind scheme was able to converge this scenario.

3.3.3 60 FOOT BOX CUT

For the 60 foot box cut scenario five differently sized grids were attempted. As seen in figure 9(a), once again convergence became an issue with the computation. In figure 9(b) and (c), the residuals decreased below $1E-3$ but never settled into a flat line. Although the results from these two runs correlate well with the PIV data, the solutions stability is questionable. Next, in figure 10(a) the residual plot seemed to flatten but the curved still maintained an upward slope and some spikes. The large increase in the residual is explained by an increase in the under-relaxation factor. While the result for this case does predict a separation, it still does not compare well with the experimental data because the separation point is predicted much too far upstream in the flow. A similar result is shown in figure 10(b). Here the residual began to increase and a reduction of the under-relaxation factor falsely produced a drop in the residual and a stabilization in the plot. This produces evidence by reducing the under-relaxation factor, the user can produce a seemingly converged solution that is not truly converged. Therefore, it is concluded that Fluent's implementation of the S-A model is not stable enough to easily predict this scenario.

3.3.4 60 FOOT SLAB CUT

Many solutions are presented for the 30 foot slab cut scenario because this scenario is potentially the most difficult flow to predict. As shown in figure 11, three different grid densities were solved to convergence. Although, all three cases resulted in a separation of the flow, agreement between the cases could not be found. Instead, a trend was discovered such that the distance from the separation point to the face increases with the grid density. Thus, proving that the grid-independent solution has not been reached. Also, if this trend continues, correlation with the experimental data will never be found. Next, figure 12 shows a variety of cases in which the solution never converged. This again raises question to Fluent's implementation of the S-A model.

3.4 The $k-\omega$ SST Model

Only two scenarios have been modeled with the $k-\omega$ SST model. The 60 foot box cut and 60 foot slab cut have been modeled with one coarse mesh and one fairly fine mesh. The residuals and velocity vector plots can be seen in figures 13 and 14. As seen in the figures, even an extremely coarse mesh generated problems with stability. All of these solutions are invalid due to the lack of convergence and proper measures should be taken by the user to try to stabilize the residuals.

4 Conclusions

In conclusion, we have seen some problems with using the commercial software package Fluent to predict the onset separation that occurs during certain mining scenarios. It is shown that the solution is highly dependent on the grid and the selection of a turbulence model. We can conclude that if converged, the S-A model does predict the onset separation better than the $k-\varepsilon$. However, Fluent's limited ability to converge the S-A model, inconsistency in results yielded from the S-A model, and the lack of correlation between these results and experimental data suggests that the S-A model does not work well for the scenarios presented in this report.

5 Future Work

There is still much work to do before the completion of this project including:

further testing of the $k-\omega$ SST model, reduced height scenarios for comparison with PIV data, and testing with different software packages such as CFX 5.5.

- Complete Grid Sensitivity study for all scenarios using $k-\omega$ turbulence model.
- Test these cases with other commercial software such as *CFX* and *Star-CD*.
- Testing scenarios with a reduced height of 5.5 ft for comparison with PIV measurements
- Model the miner in the mine face and compare with PIV measurements.

6 Figures

6.1 Geometry for the Four Scenarios

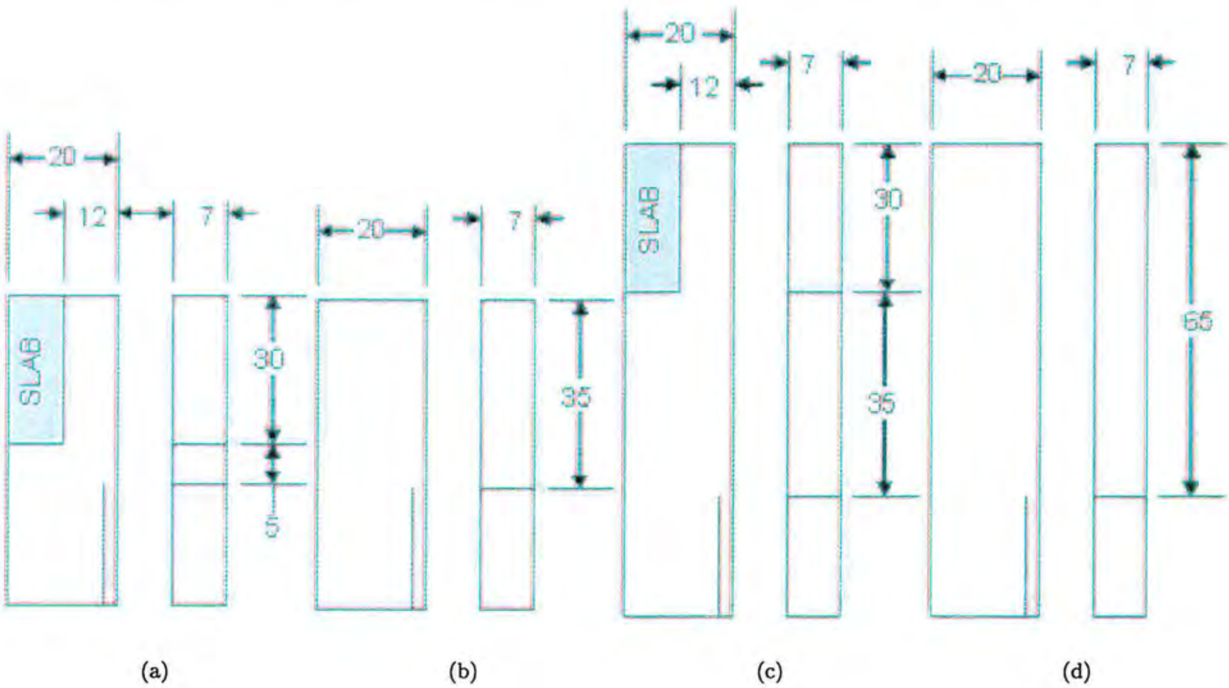
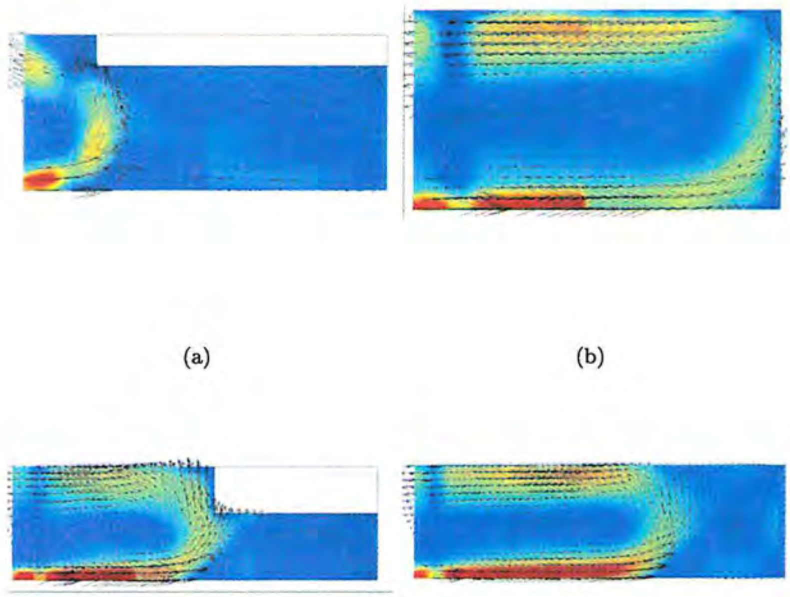


Figure 1: Geometry and dimensions for the four scenarios used to simulate the path of the miner. (a) 30 ft box cut, (b) 30 ft slab cut, (c) 60 ft box cut and (d) 60 ft slab cut.

6.2 Particle Image Velocimetry (PIV)

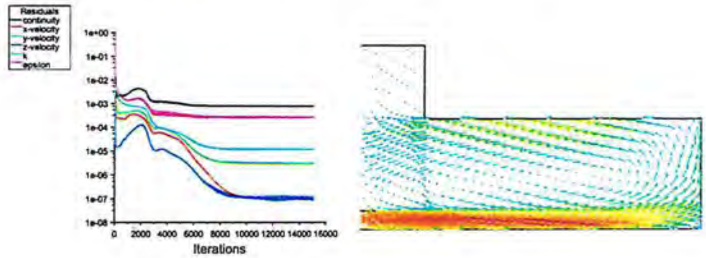


(a) (b)

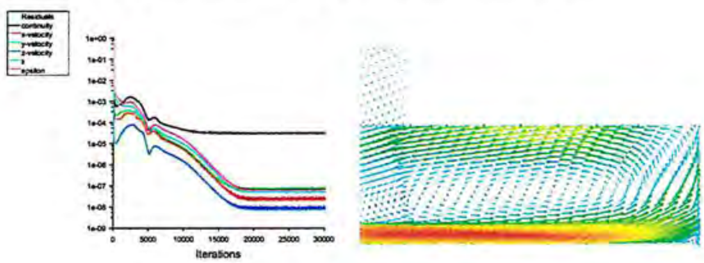
(c) (d)

Figure 2: Velocity-vector plots for a) 30 foot box cut, b) 30 foot slab cut, c) 60 foot box cut and d) 60 foot slab cut, using the PIV experiments taken by Dr. Wala and Dr. Jacob.

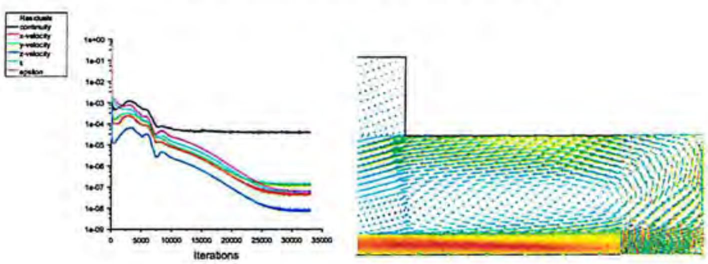
6.3 The $k - \varepsilon$ Model with Standard Wall Function



(a) 600 thousand grid cells, 1st order upwind



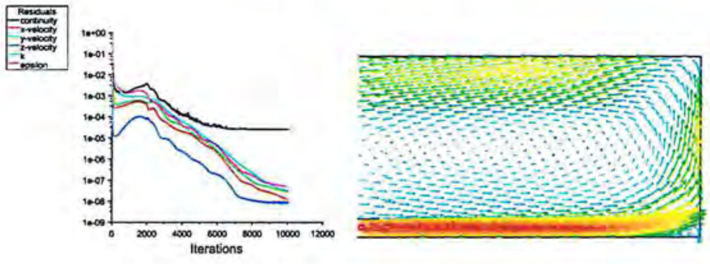
(b) 900 thousand, 1st order upwind



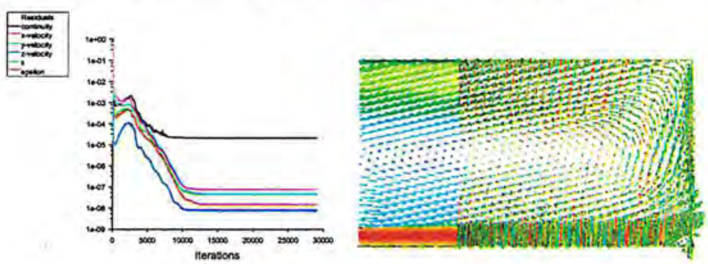
(c) 1.2 million grid cells, 1st order upwind

Figure 3: Residuals and velocity-vector plots for the 30 ft box cut scenario using the $k - \varepsilon$ turbulence model.

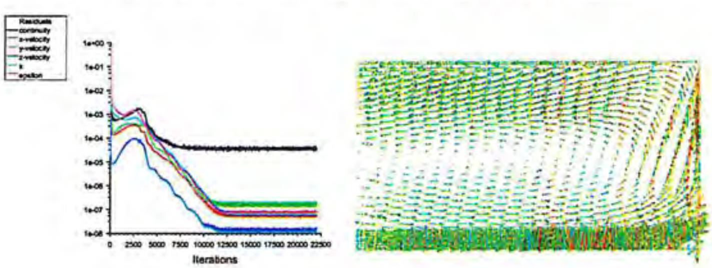
The $k - \varepsilon$ Model with Standard Wall Function continued



(a) 700 thousand grid cells, 1st order upwind scheme



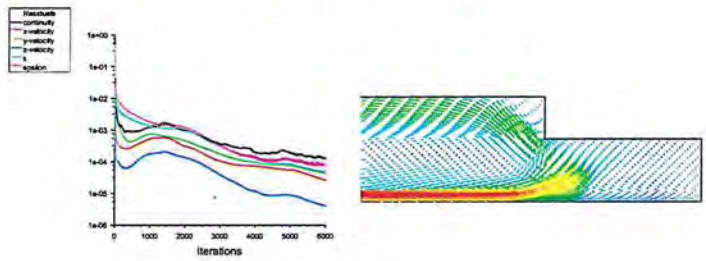
(b) 975 thousand grid cells, 1st order upwind scheme



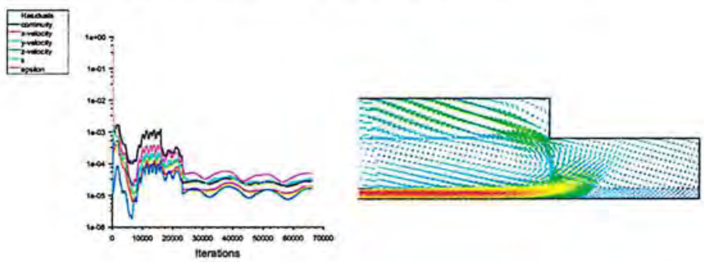
(c) 1.2 million grid cells, 1st order upwind scheme

Figure 4: Residuals and velocity-vector plots for the 30 ft slab cut scenario using the $k - \varepsilon$ turbulence model.

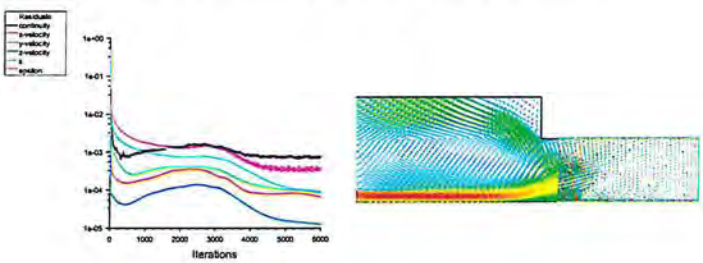
The $k - \varepsilon$ Model with Standard Wall Function continued



(a) 600 thousand grid cells, QUICK scheme



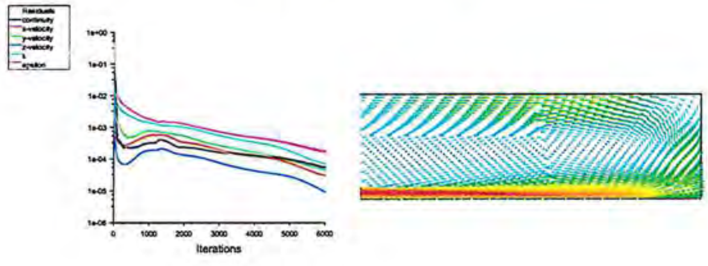
(b) 900 thousand grid cells, QUICK scheme



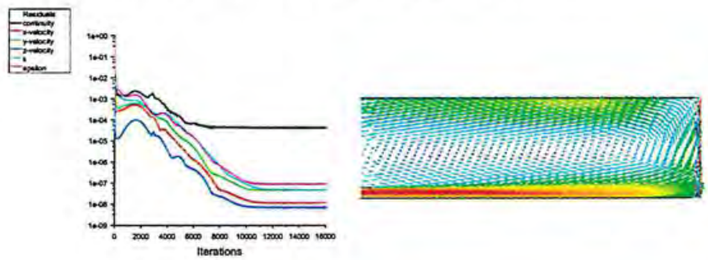
(c) 1.2 million grid cells, QUICK scheme

Figure 5: Residuals and velocity-vector plots for the 60 ft box cut scenario using the $k-\varepsilon$ turbulence model.

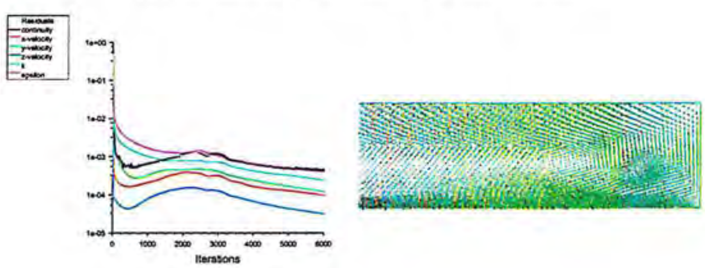
The $k-\varepsilon$ Model with Standard Wall Function continued



(a) 600 thousand grid cells, QUICK scheme



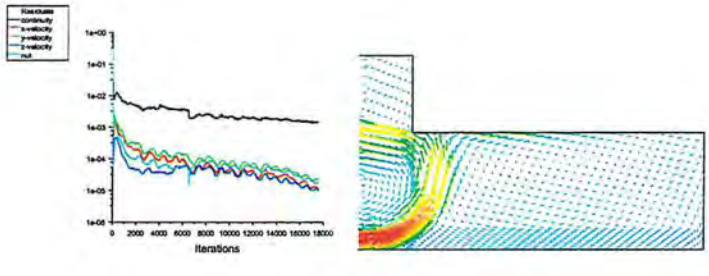
(b) 1.0 million grid cells, QUICK scheme



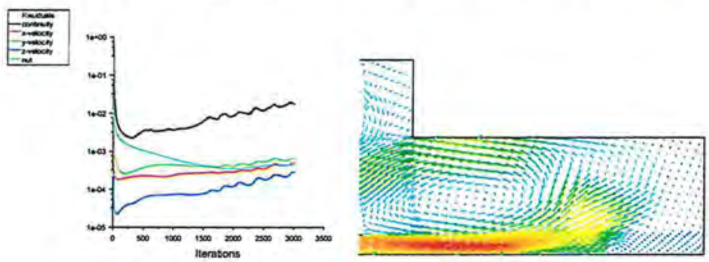
(c) 1.2 million grid cells, QUICK scheme

Figure 6: Residuals and velocity-vector plots for the 60 ft slab cut scenario using the $k-\varepsilon$ turbulence model.

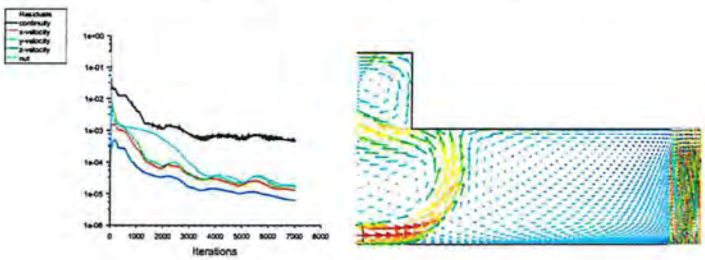
6.4 The Spalart-Allmaras Model



(a) 1.2 million grid points solved with first order upwind



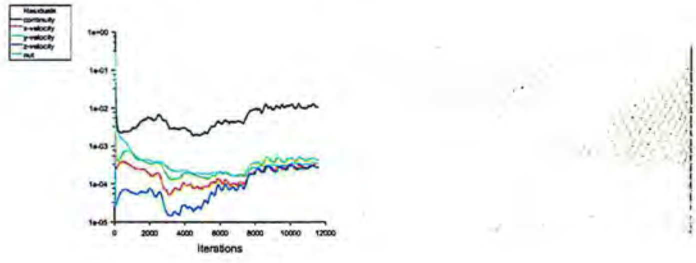
(b) 1.2 million grid points solved with the QUICK scheme.



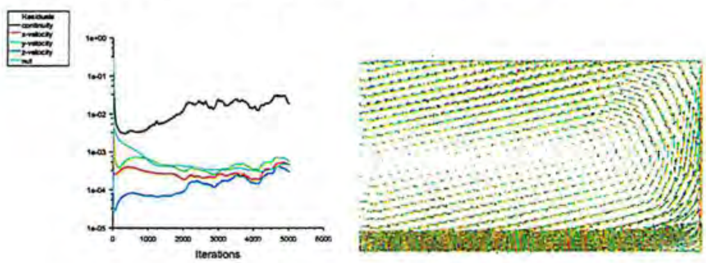
(c) 800K grid points solved using QUICK.

Figure 7: Residuals and velocity-vector plots for the 30 ft box cut scenario using the Spalart-Allmaras turbulence model.

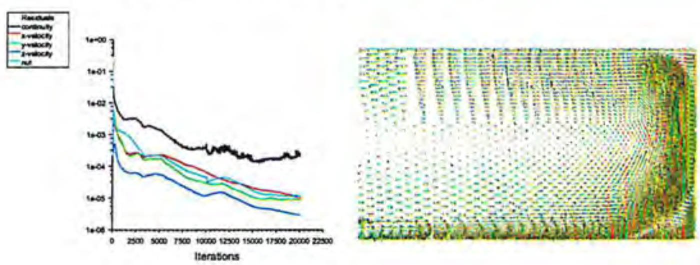
The Spalart-Allmaras Model continued



(a) 1.2 million grid points, first order upwind scheme



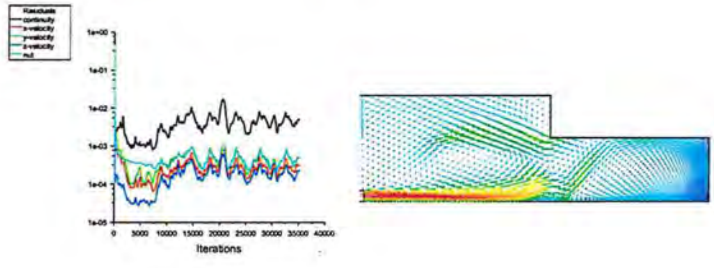
(b) 1.2 million grid points, QUICK scheme



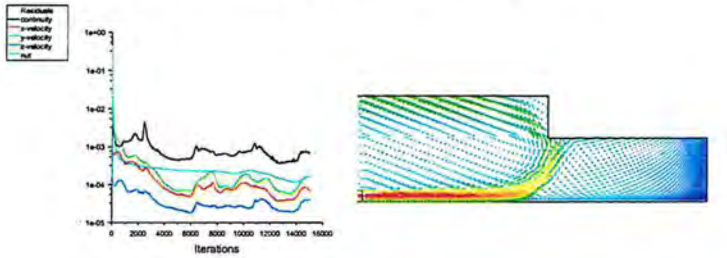
(c) 800K grid points, QUICK scheme

Figure 8: Residuals and velocity-vector plots for the 30 ft slab cut scenario using the Spalart-Allmaras turbulence model.

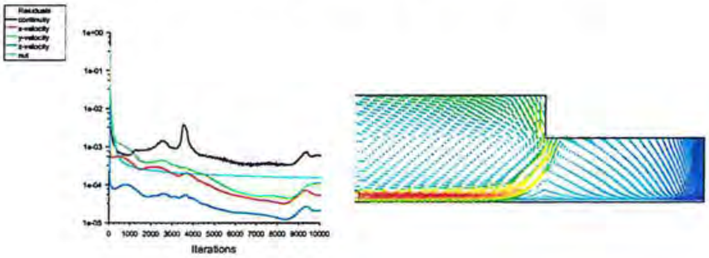
The Spalart-Allmaras Model continued



(a) 700 thousand grid cells, QUICK scheme



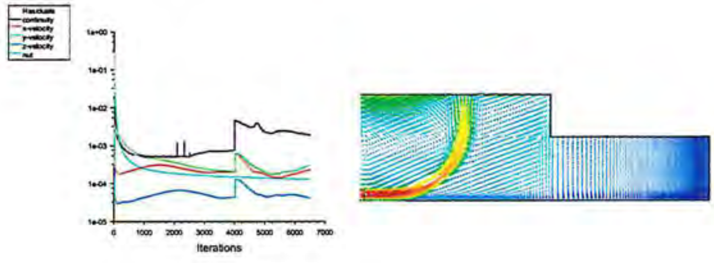
(b) 1.4 million grid cells, QUICK scheme



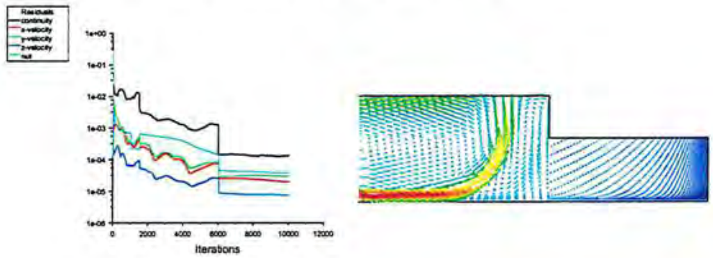
(c) 2.5 million grid cells, QUICK scheme

Figure 9: Residuals and velocity-vector plots for the 60 ft box cut scenario using the S-A turbulence model.

The Spalart-Allmaras Model continued



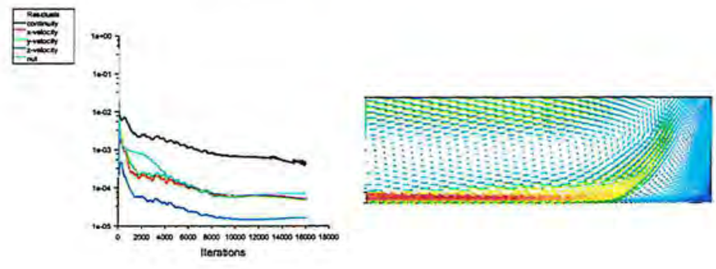
(a) 1.5 million grid cells, QUICK scheme



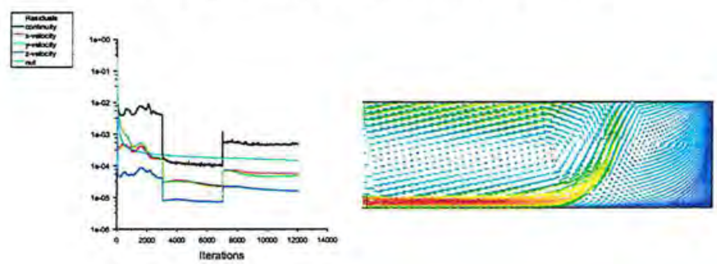
(b) 2.2 million grid cells, QUICK scheme

Figure 10: Residuals and velocity-vector plots for the 60 ft box cut scenario using the S-A turbulence model.

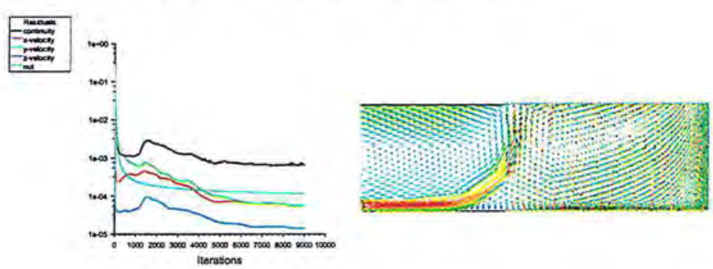
The Spalart-Allmaras Model continued



(a) 950 thousand grid cells, QUICK scheme



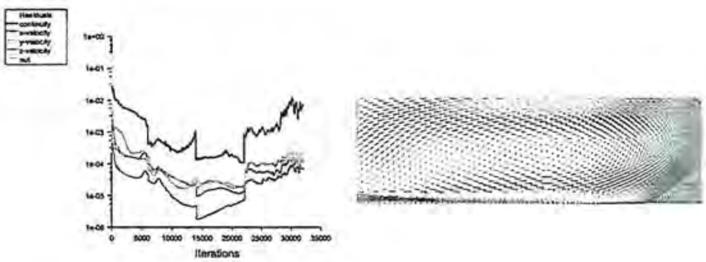
(b) 1.8 million grid cells, QUICK scheme



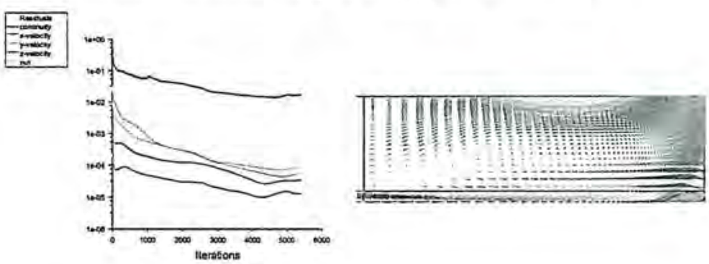
(c) 3.0 million grid cells, QUICK scheme

Figure 11: Residuals and velocity-vector plots for the 60 ft slab cut scenario using the S-A turbulence model.

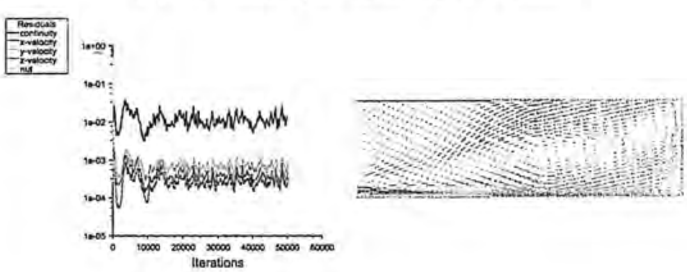
The Spalart-Allmaras Model continued



(a) 700 thousand grid cells, QUICK scheme



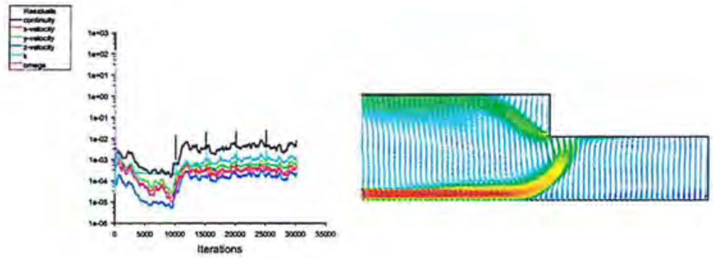
(b) 2.0 million grid cells, QUICK scheme



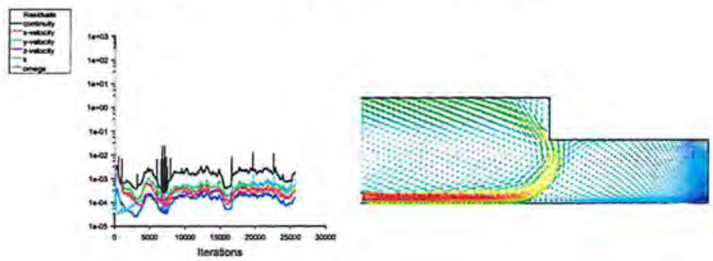
(c) 300 thousand grid cells, QUICK scheme

Figure 12: Residuals and velocity-vector plots for the 60 ft slab cut scenario using the S-A turbulence model.

6.5 The $k - \omega$ SST Model



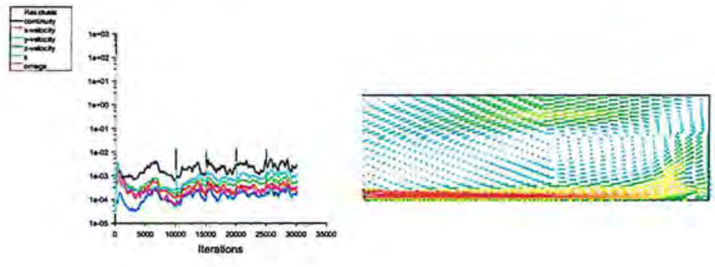
(a) 270 thousand grid cells, QUICK scheme



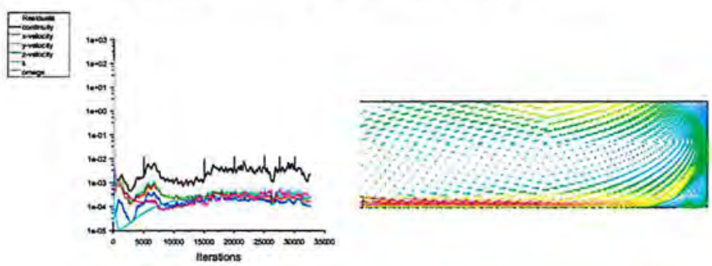
(b) 700 thousand grid cells, QUICK scheme

Figure 13: Residuals and velocity-vector plots for the 60 ft box cut scenario using the $k - \omega$ SST turbulence model.

The $k - \omega$ SST Model continued



(a) 300 thousand grid cells, QUICK scheme



(b) 950 thousand grid cells, QUICK scheme

Figure 14: Residuals and velocity-vector plots for the 60 ft slab cut scenario using the $k - \omega$ SST turbulence model.

References

[1] Wala, Andrezj, Jacob, J.D., Huang, G.P., and Brown J.T. Numerical and Experimental Study of a Mine Face Ventilation System for CFD Validation. A Pre-Print of the 2002 SME Annual Meeting in Phoenix, AZ.

[2] Fluent User's Guide

[3] Wilcox, David C. BASIC FLUID MECHANICS. DCW Industries, Inc. First Edition. 1997. Pages 1-13.

Prediction of Mine Face Ventilation using Computational Fluid Dynamics

University of Kentucky

October 6, 2003

Introduction

The objective of this document is to update NIOSH on the progress that has been made in the past year with respect to CFD predictions of mine face ventilation cases using commercial software. Several studies will be presented including a comparison of Fluent with CFX for all four scenarios discussed in the previous years report, modeling of a cylindrical scrubber within the mine and simulations of a the flow around a continuous miner, with scrubbers on-board, at the cutting face.

Comparing Fluent to CFX

A comparison of Fluent and CFX was performed by simulating the four scenarios presented in last years report. To eliminate variation in grid generation and refinement an external software, GridPro, was utilized to generate all the grids in this study. GridPro is an automatic grid generator which yields a structured grid that can be imported directly into both of the solvers mentioned above. Three grids for each scenario (coarse, moderate, fine) were computed using both Fluent and CFX in an effort to find a grid-independent solution. The results of said solutions are presented in this section.

Baseline PIV Results

Dr's. Andrzej M. Wala and Jamey D. Jacob of the departments of Mining and Mechanical Engineering, respectively, performed the PIV experiments of each scenario using a 1:15 scale model of a mine face. The results of these experiments, figures 1 through 4, will be used as the baseline for our simulations. In three of the four cases, an onset separation can be seen where flow reverses before reaching the face. This separation is driven by an adverse pressure gradient which builds up as the width of the mine decreases. The κ-ω SST (Shear Stress Transport) turbulence model is was selected for its ability to accurately predict separation due to adverse pressure gradients.

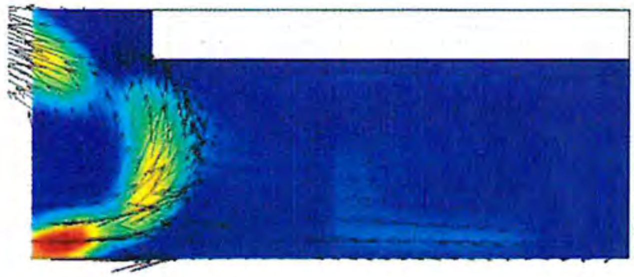


Figure 1: Velocity vectors and contours from PIV experiments for 30 Foot Box Cut

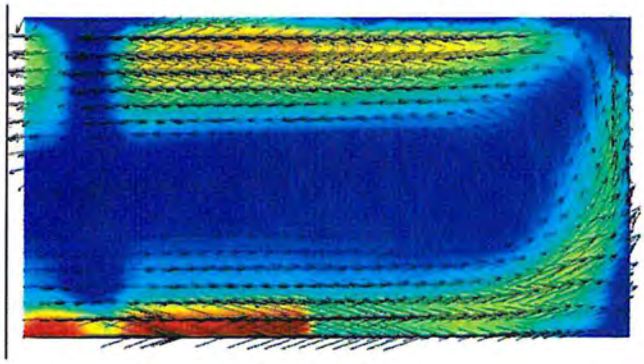


Figure 2: Velocity vectors and contours from PIV experiments for 30 Foot Slab Cut

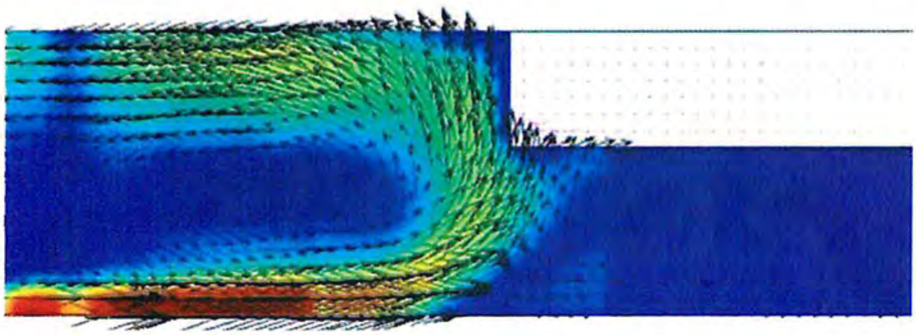


Figure 3: Velocity vectors and contours from PIV experiments for 60 Foot Box Cut

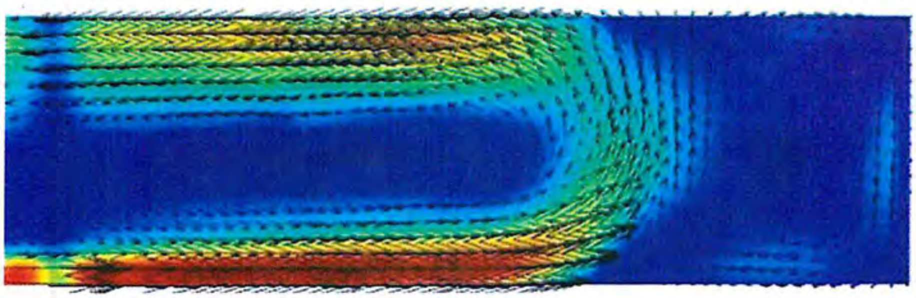


Figure 4: Velocity vectors and contours from PIV experiments for 60 Foot Slab Cut

CFD Results

30 Foot Box Cut

In the scenario for the 30 foot box cut an onset separation develops only a few feet downstream from the end of the curtain at the beginning of the slab. This flow separation produces a low velocity recirculation of air deep into the mine. In an actual mine, this flow pattern could allow dangerous levels of methane gas to build near the cutting face. Although the introduction of methane gas to the flow is not being simulated here, a correlation between flow patterns containing air only and air with methane is assumed. Figures 5, 6 and 7 illustrate the results yielded by Fluent and CFX for the three grid densities (coarse, moderate and fine). Comparing these results to the baseline results, figure 1, one can see that Fluent predicts the desired characteristics independent of grid density. Although, the moderate and fine grid results for CFX correctly predict the separation point, the coarse grid solution did not. Thus, some confidence in the CFX solutions is lost. However, since the finer grids yield a proper solution, one could make the argument that this is the grid-independent solution for CFX.

30 Foot Slab Cut

In the 30 foot slab cut scenario no onset separation occurs as that flow carries entirely to the face before redirecting. Simulating this scenario tests the codes ability to decipher a difference between flows with and without separation. Although it is known that predicting a flow field without separation is much easier, the code must be able to predict the flow field with the same method used for the flow fields with separation. This is especially important since the CFD tools in question may be used to develop and optimize mining techniques and setups. Comparing the solutions in figures 8, 9 and 10 to the PIV result illustrated in figure 2 similar flow patterns are found. Noticing that for all three grid levels the flow carries to the mine face before turning, one can confirm that both codes predict an accurate grid-independent solution.

60 Foot Box Cut

In the 60 foot box cut scenario onset separation occurs at the beginning of the slab as shown in figure 3. As in the 30 foot box cut, this separation generates a low velocity recirculation of air deep into the mine. If this flow field were generated during normal operation of a real mine, dangerous levels of methane could collect near the cutting face. In figures 11, 12 and 13, it is shown that both codes predict the separation in the proper location for the coarse, moderate and fine grid cases. Therefore, one can claim that the grid-independent solution generated by both codes is accurate for this scenario.

60 Foot Slab Cut

The 60 foot slab cut scenario occurs when the mine is cut to full width, 20 feet, and 60 feet deep. One may expect the flow field to resemble that of the 30 foot slab cut since such a resemblance was seen for the two box cut cases. However, an interesting result is found for the scenario in question when one views the PIV result illustrated in figure 4. A late separation of the flow occurs in the mine roughly 20 feet shy of the cutting face. As seen in figures 14, 15 and 16, both codes had difficulty predicting this flow field. Due to the highly unstable nature of the flow, an accurate steady-state simulation of flow structure was not feasible. The codes' inability to generate a converged solution yielded variation in the results. Although less variation was seen in Fluent's prediction of the flow structure, neither code was able to produce converged, accurate solutions for this scenario. One suggestion may be to approach the problem from a transient standpoint to capture any oscillations that may occur in the flow. However, this method will be costly with respect to computation time and is not guaranteed to yield a correct solution. In any case, the 60 foot box cut scenario deserves further study before results can be utilized as a reliable design tool.

Modeling Scrubbers using Fluent

In this section, a cylindrical scrubber was inserted into the 30 foot box cut geometry to represent experiments performed by Wala and Jacob. Results for these studies are presented in the annual report submitted by Dr. Wala to NIOSH in 2003. In these studies, the scrubber mass flow ratio is normalized by the mass flow at the inlet, or behind the curtain. Thus, the results presented in figures 17 through 21 illustrate predicted flow fields at scrubber mass flow ratios of 0.2, 0.3, 0.4, 0.7 and 1.0, respectively. The boundary condition for the scrubber is set as a constant velocity outlet which is released outside the original geometry. This boundary condition is set by the apparatus available to Wala for PIV experimentations. The grid density was based on the coarse grids simulated in the previous section. It should be noted that a grid-independent solution has not been found for these results due to time constraints.

In comparing the results shown in this report with the PIV results presented by Wala, one will notice a discrepancy. Wala's experiments show the location of the separation point to be more sensitive to smaller values of the scrubber mass flow ratio. However, once the scrubber mass flow ratio reaches a level near 1.0 the separation is removed and the flow carries to the cutting face. Since most scrubbers in actual mining applications are required to have a mass flow ratio greater than or equivalent to 1.0, one could argue that Fluent's predictions are accurate within the range of applicable problems. However, users of Fluent as a design tool should make note of the limitations that are found in this study when applying Fluent to scenarios beyond or on the borders of this range. A potential scenario that should be avoided is an attempt to reduce the requirement for the scrubber mass flow rate for the purpose of using smaller, less expensive blowers.

Modeling a Continuous Miner with On-board Scrubbers using Fluent

Once CFD is validated for benchmarked applications, the strength of CFD can be fully utilized. Prior to this section, the geometries of the models presented in this report can be regarded as "simple". Thus, experimentation seems to be a more effective and productive means of study. However, these simplified geometries are used as benchmarks to validate the CFD codes for use in more realistic applications. Because CFD does not require a model shop, geometries are no longer time consuming to build, relative to the time it takes to generate a precise scale model of intricate detail. Geometries can be imported directly from 3-D solid modeling software or manually generated in Gambit, Fluent's grid generation package. Also, the CFD user has access to many more boundary conditions than the experimentalist at a fraction of the cost. For example, study the flow over a continuous miner with scrubbers on-board is a much easier and less expensive task for a CFD user than for an experimentalist. The experimentalist would be forced to make a decision between a full-scale model and a scaled model. To put an actual continuous miner in a full scale mine and set up the measuring devices would be expensive and time consuming. Building a scaled miner and mine using a rapid prototyping technique and setting up the measurement apparatus would also be somewhat less expensive but still time consuming. However, dumping a 3-D rendering of the miner into a virtual mine, generating a grid and running the simulation with the appropriate boundary conditions would take an experienced CFD user a matter of weeks and cost of only the software and hardware. Furthermore, once the software and hardware are paid for, they can be re-used for other models. The power of CFD now becomes apparent, but more work must be done to ensure the quality of the solutions it generates.

The results in this section provide a look ahead to see some of the possibilities that CFD has to offer. In these simulations, a simplified virtual continuous miner was inserted into the mine for the 30 foot box cut scenario. Figures 22 and 23 show different views of the virtual geometry. The miner is equipped with a virtual scrubber which is incorporated into the geometry of the mine such that the scrubber's inlets are placed on each side of the miner and the exhaust is placed at its rear. Unlike the previous scrubber, this scrubber is modeled with the "fan" UDF (User Defined Function) available in Fluent. This method of modeling allows the user to put a pressure gradient condition on a plane within the scrubber's duct essentially modeling a blower mounted to the miner. The main difference between this method and the previously used method is that the scrubber's exhaust is released within the mine as opposed to outside. This better represents the scenario of a miner with on-board scrubber and would be very difficult for an experimentalist to implement into a scaled model due to flow restrictions on fans small enough to fit these models. One downside to using the "fan" UDF is that the input is a pressure gradient which cannot be directly related to a mass flow rate without first solving the flow field. However, within a matter of four simulations, a scrubber mass flow ratio, normalized by the inlet mass flow as before, of 0.95 is reached.

Figure 24 illustrates the resulting flow over the miner without the use of a scrubber. The effect of the scrubber can be seen by comparing figures 25 through 28 with figure 24. The corresponding scrubber mass flow ratios for figures 25, 26, 27 and 28 are 0.37, 0.67, 0.83 and 0.95, respectively.

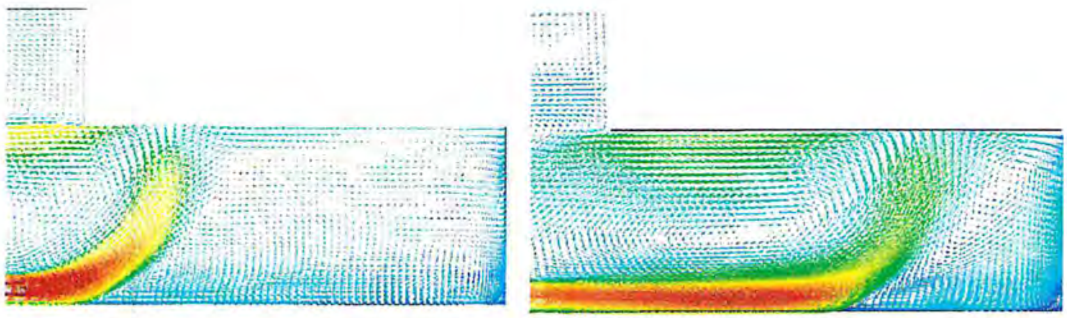


Figure 5: Velocity vectors for 30 foot box cut using Fluent (left) and CFX (right) with a coarse grid.

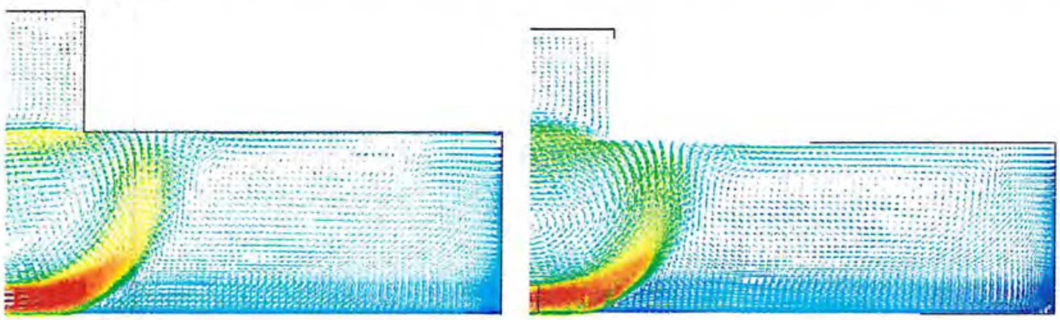


Figure 6: Velocity vectors for 30 foot box cut using Fluent (left) and CFX (right) with a moderate grid.

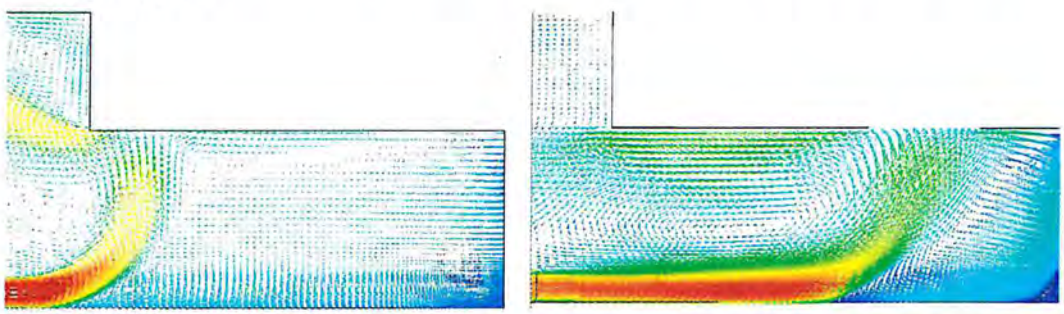


Figure 7: Velocity vectors for 30 foot box cut using Fluent (left) and CFX (right) with a fine grid.

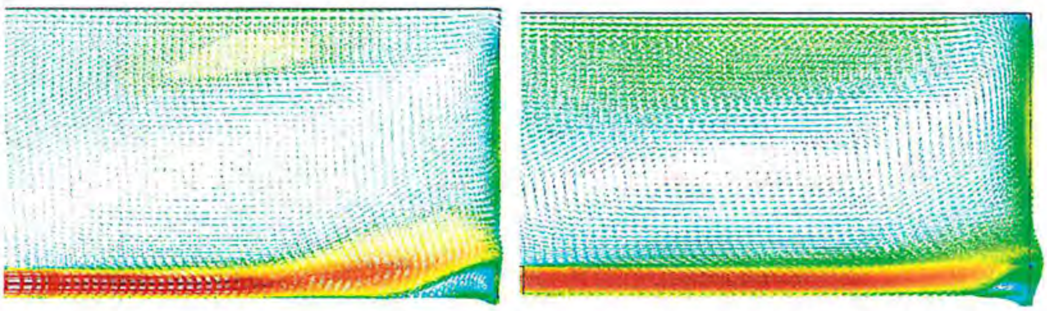


Figure 8: Velocity vectors for 30 foot slab cut using Fluent (left) and CFX (right) with a coarse grid.

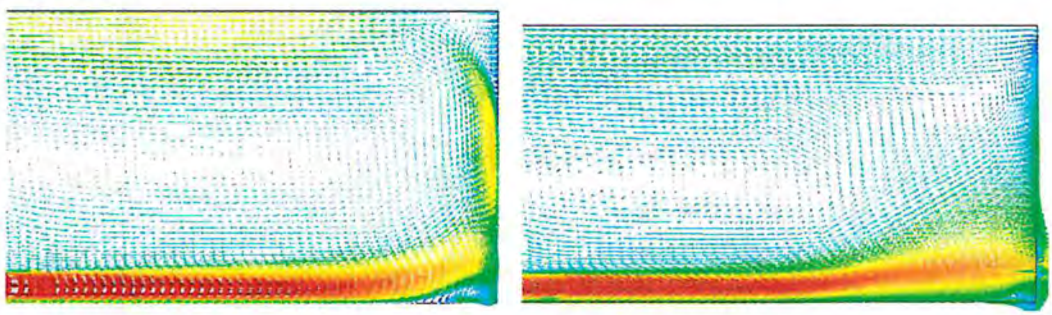


Figure 9: Velocity vectors for 30 foot slab cut using Fluent (left) and CFX (right) with a moderate grid.

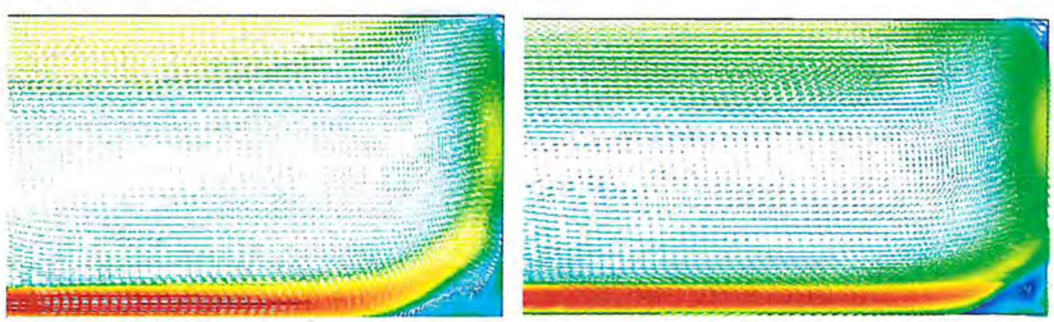


Figure 10: Velocity vectors for 30 foot slab cut using Fluent (left) and CFX (right) with a fine grid.

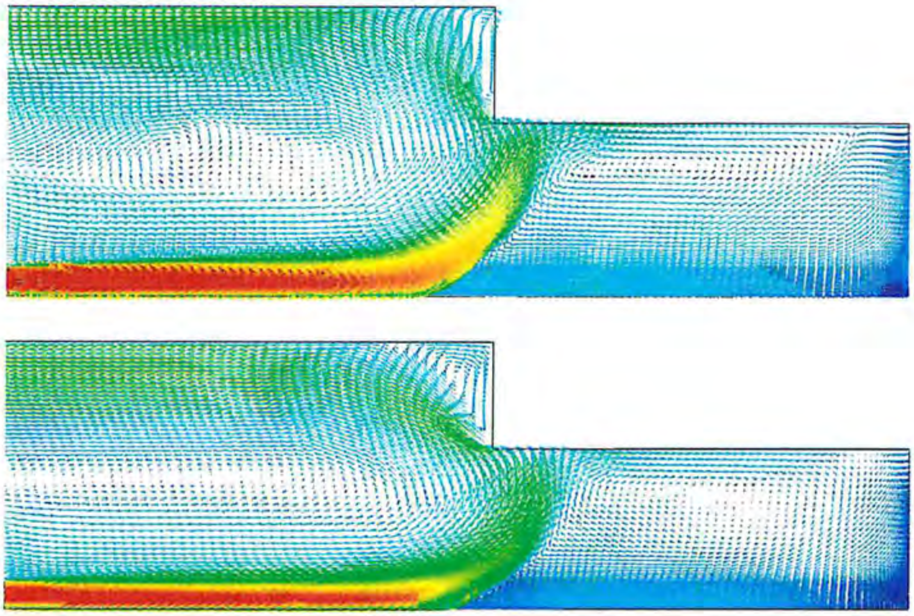


Figure 11: Velocity vectors for 60 foot box cut using Fluent (top) and CFX (bottom) with a coarse grid.

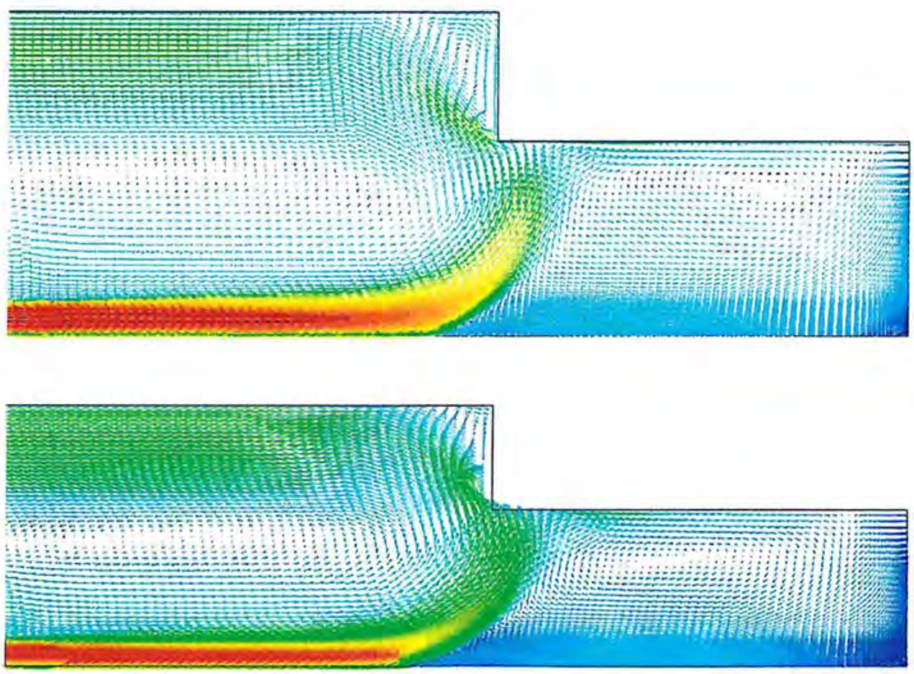


Figure 12: Velocity vectors for 60 foot box cut using Fluent (top) and CFX (bottom) with a moderate grid.

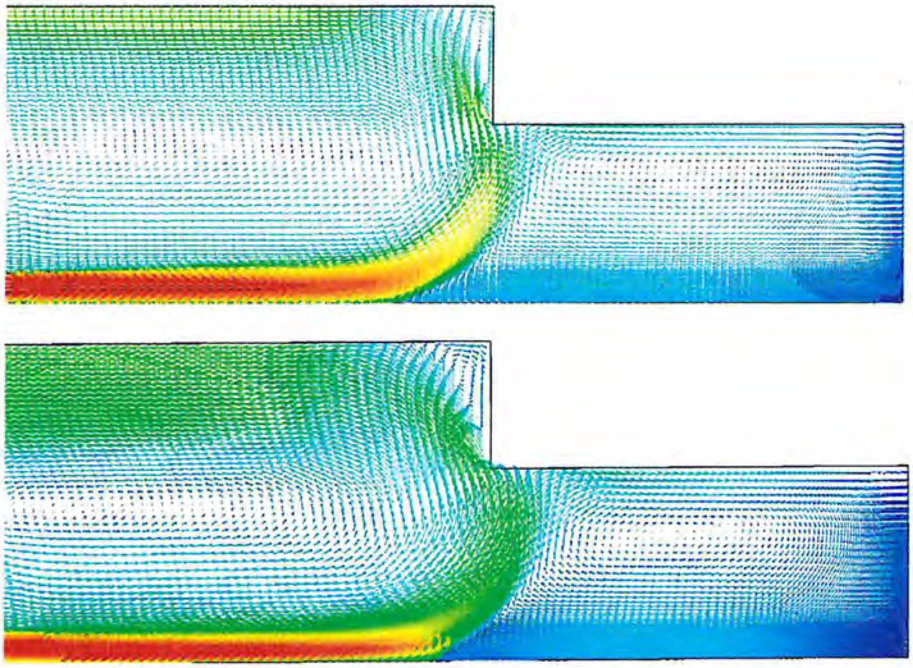


Figure 13: Velocity vectors for 60 foot box cut using Fluent (top) and CFX (bottom) with a fine grid.

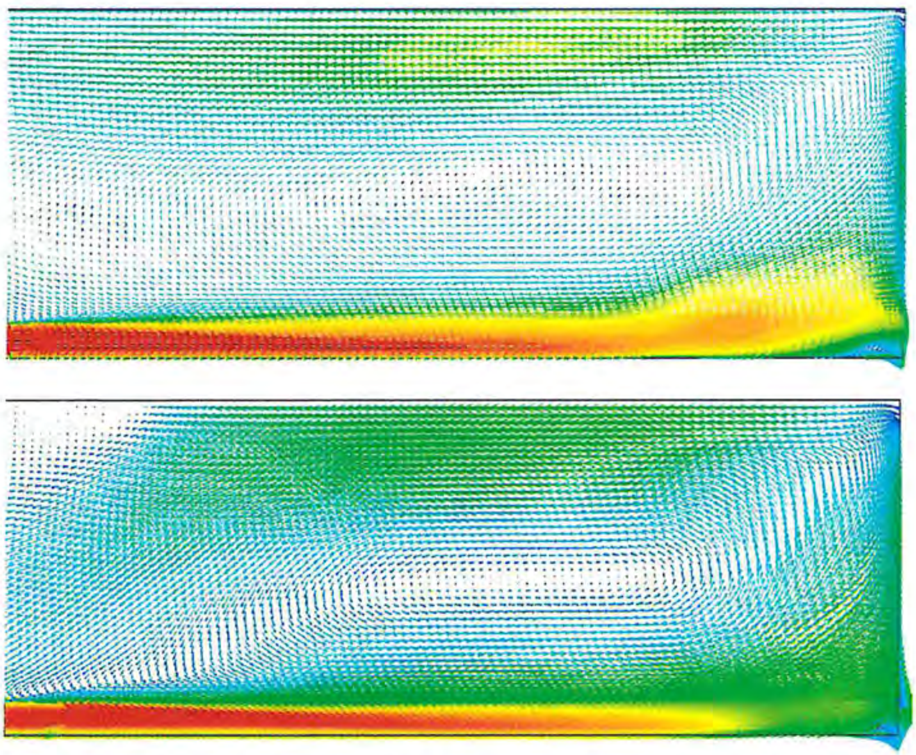


Figure 14: Velocity vectors for 60 foot slab cut using Fluent (top) and CFX (bottom) with a coarse grid.

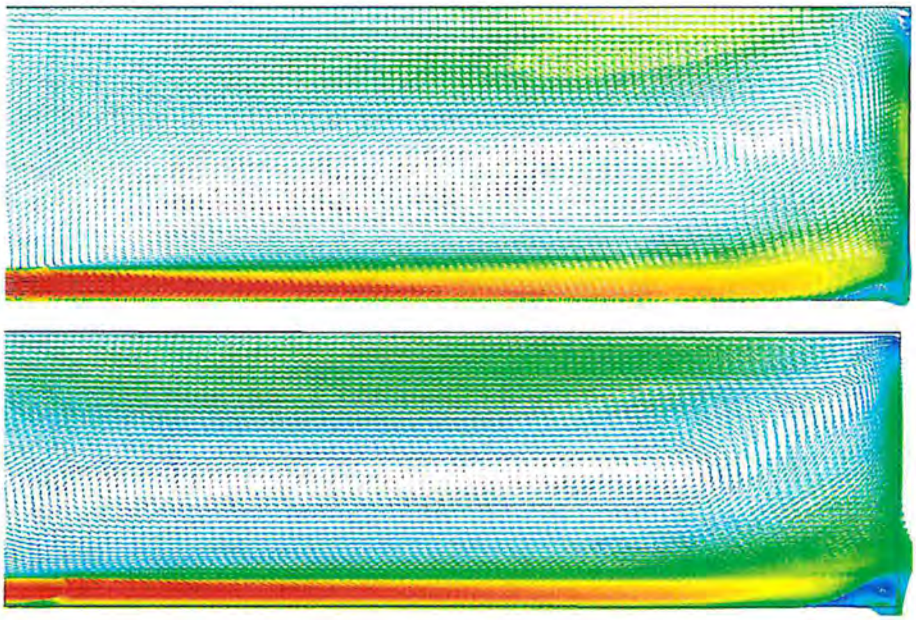


Figure 15: Velocity vectors for 60 foot slab cut using Fluent (top) and CFX (bottom) with a moderate grid.

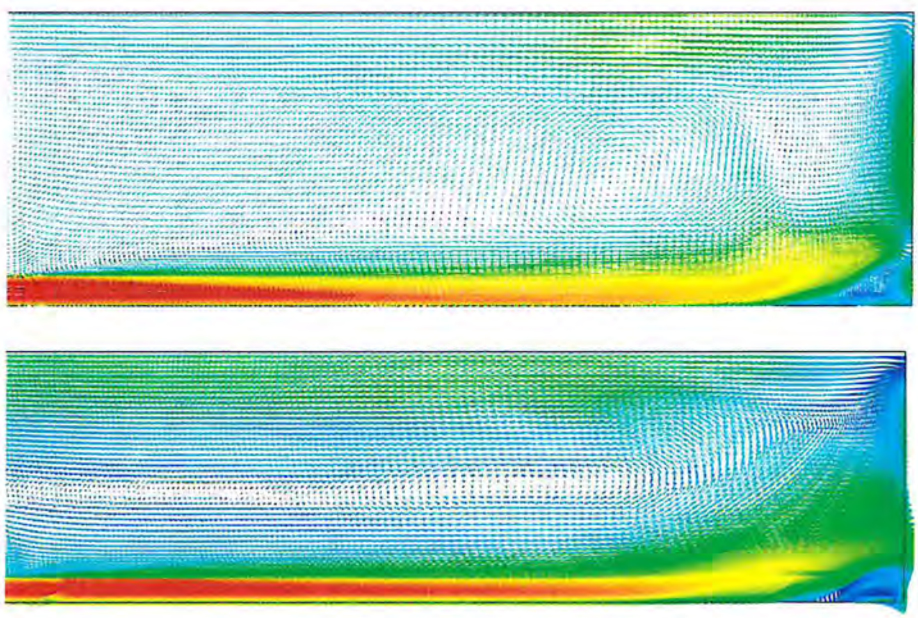


Figure 16: Velocity vectors for 60 foot slab cut using Fluent (top) and CFX (bottom) with a fine grid.

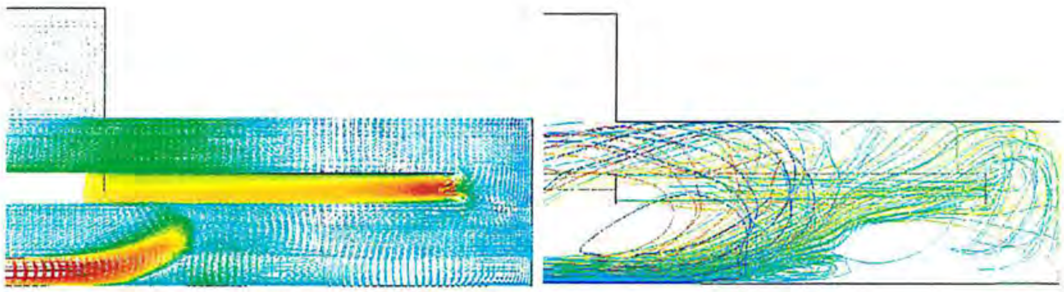


Figure 17: (Left) Velocity vectors in the Scrubber Plane and (right) path lines from the top view for the 30 foot box cut scenario with scrubber mass flow ratio of 20%.

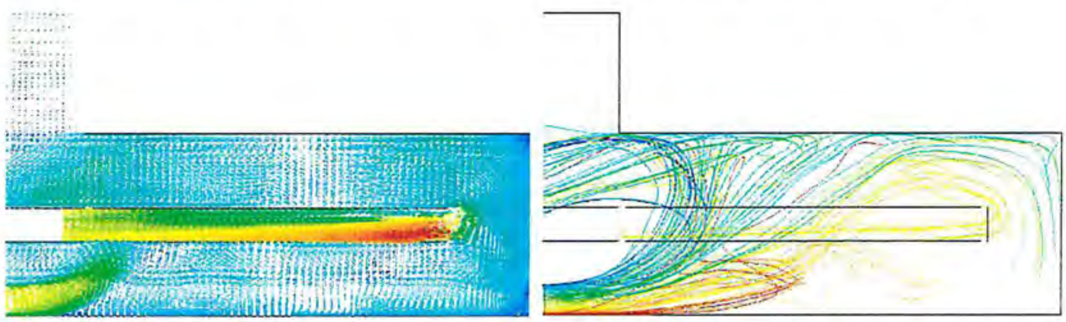


Figure 18: (Left) Velocity vectors in the Scrubber Plane and (right) path lines from the top view for the 30 foot box cut scenario with scrubber mass flow ratio of 30%.

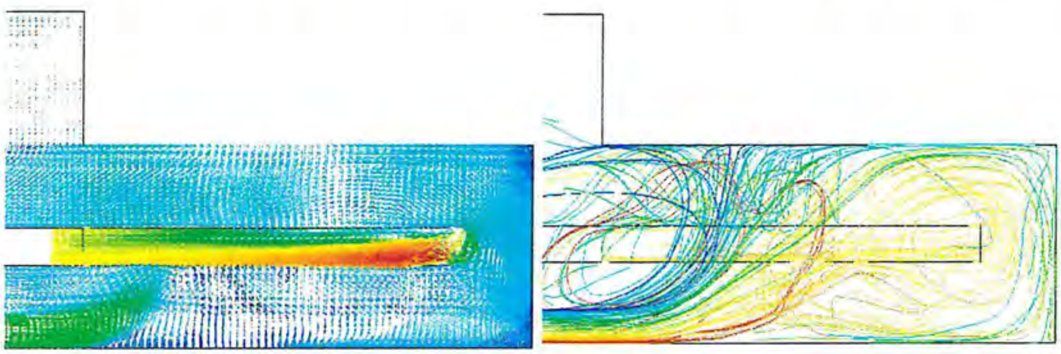


Figure 19: (Left) Velocity vectors in the Scrubber Plane and (right) path lines from the top view for the 30 foot box cut scenario with scrubber mass flow ratio of 40%.

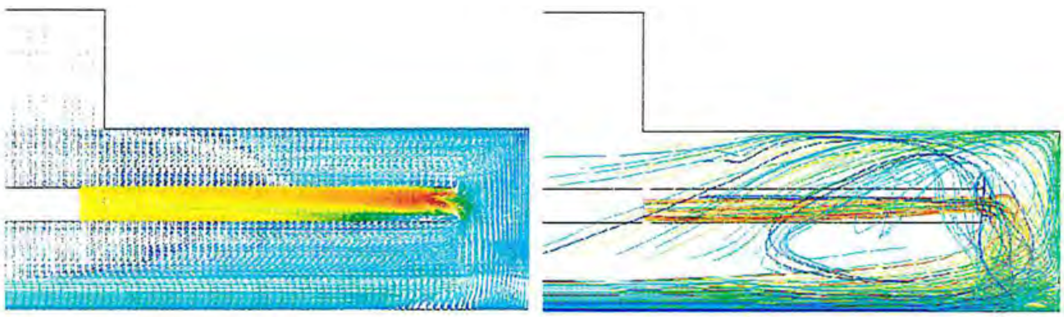


Figure 20: (Left) Velocity vectors in the Scrubber Plane and (right) path lines from the top view for the 30 foot box cut scenario with scrubber mass flow ratio of 70%.

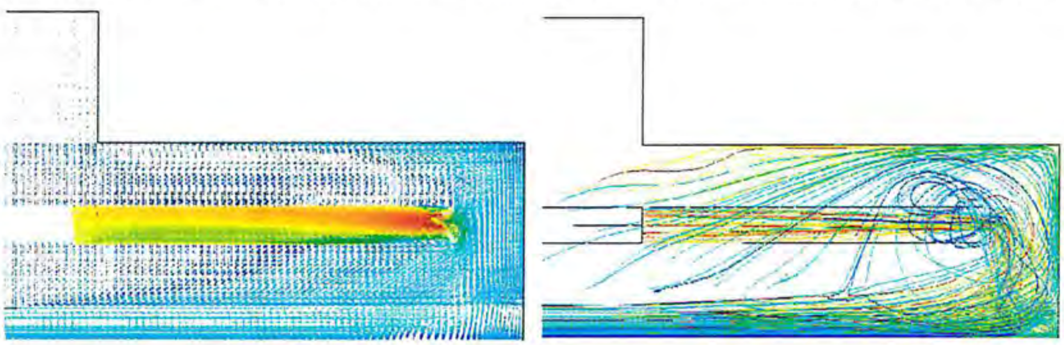


Figure 21: (Left) Velocity vectors in the Scrubber Plane and (right) path lines from the top view for the 30 foot box cut scenario with scrubber mass flow ratio of 100%.

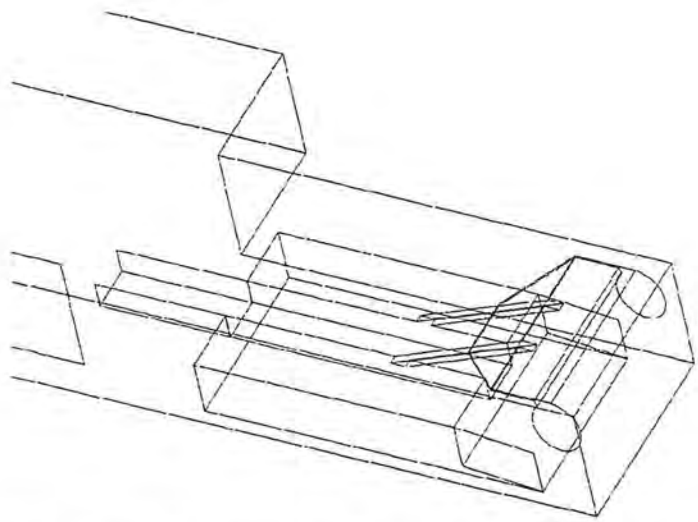


Figure 22: Isometric view of a simplified representation of a continuous miner inserted into the 30 foot box cut scenario.

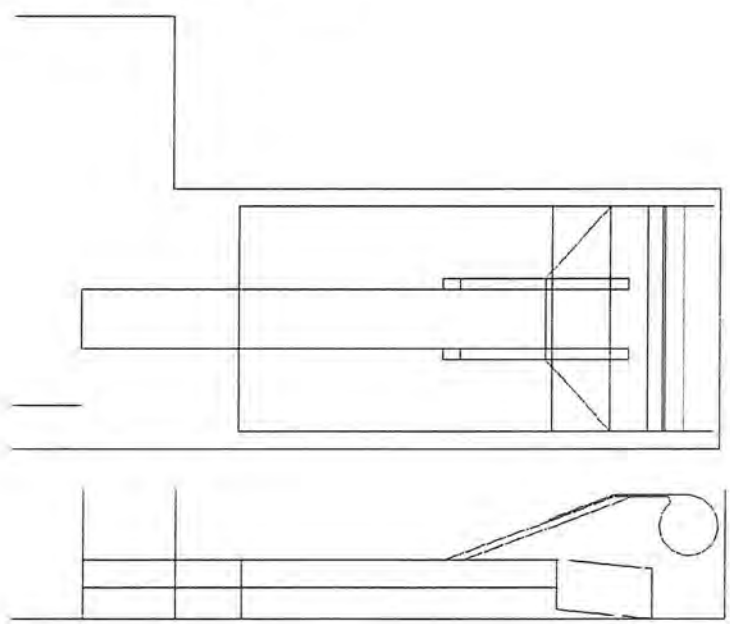


Figure 23: A top view (top) and a side view (bottom) of a simplified representation of a continuous miner inserted into the 30 foot box cut scenario.

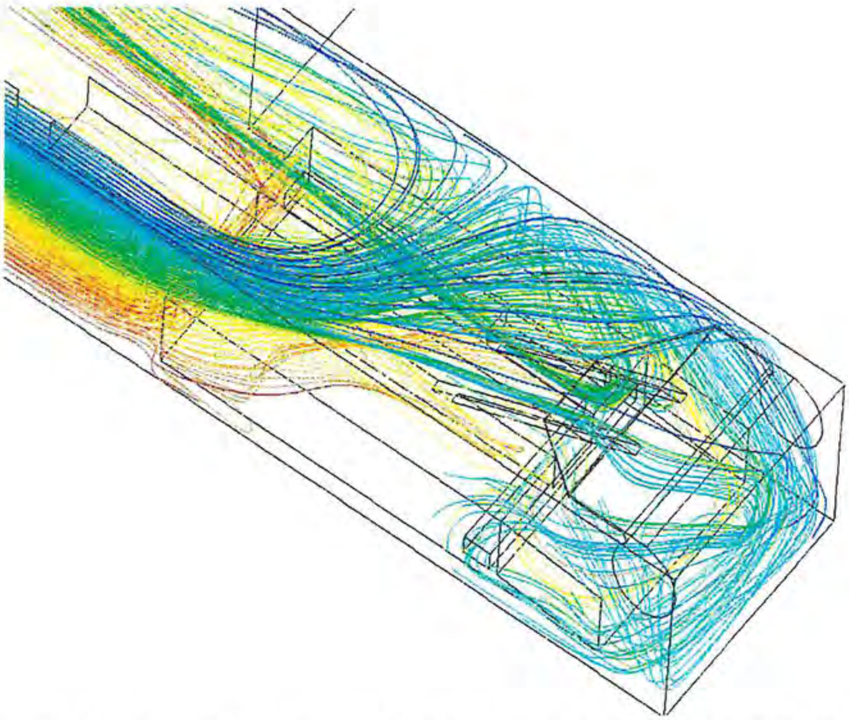


Figure 24: 3-D path lines illustrating the flow over a continuous miner in the 30 foot box cut scenario without a scrubber.

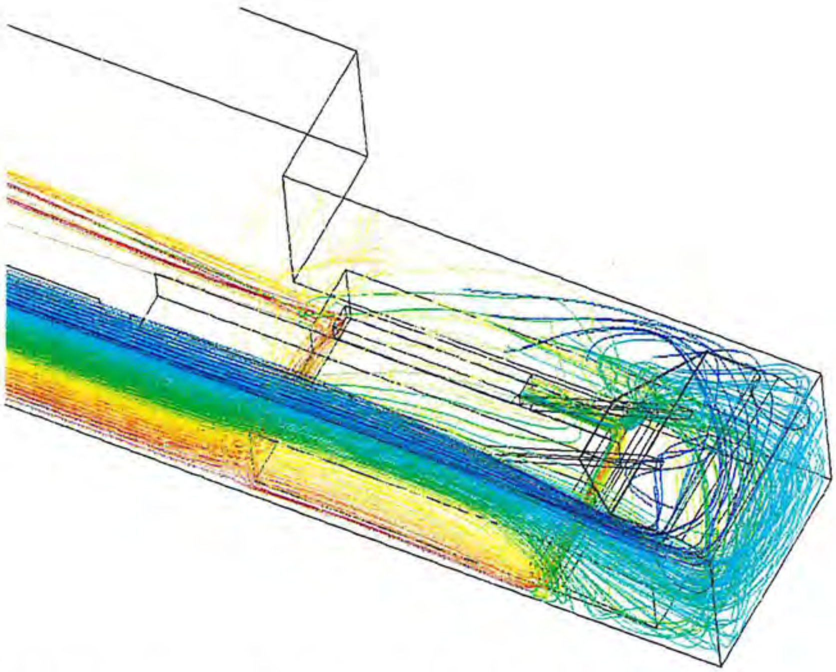


Figure 25: 3-D path lines illustrating the flow over a continuous miner in the 30 foot box cut scenario with a scrubber mass flow ratio of 0.37.

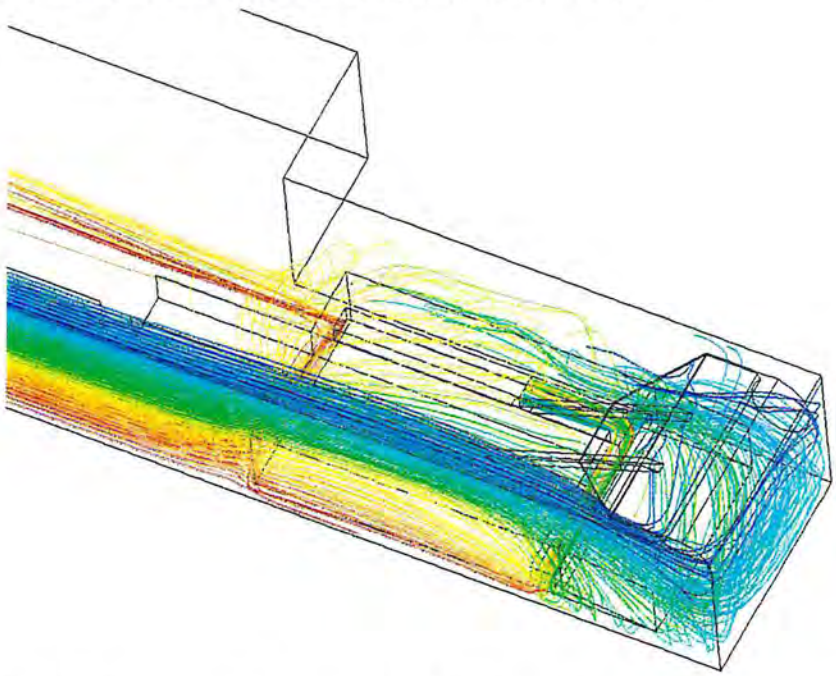


Figure 26: 3-D path lines illustrating the flow over a continuous miner in the 30 foot box cut scenario with a scrubber mass flow ratio of 0.67.

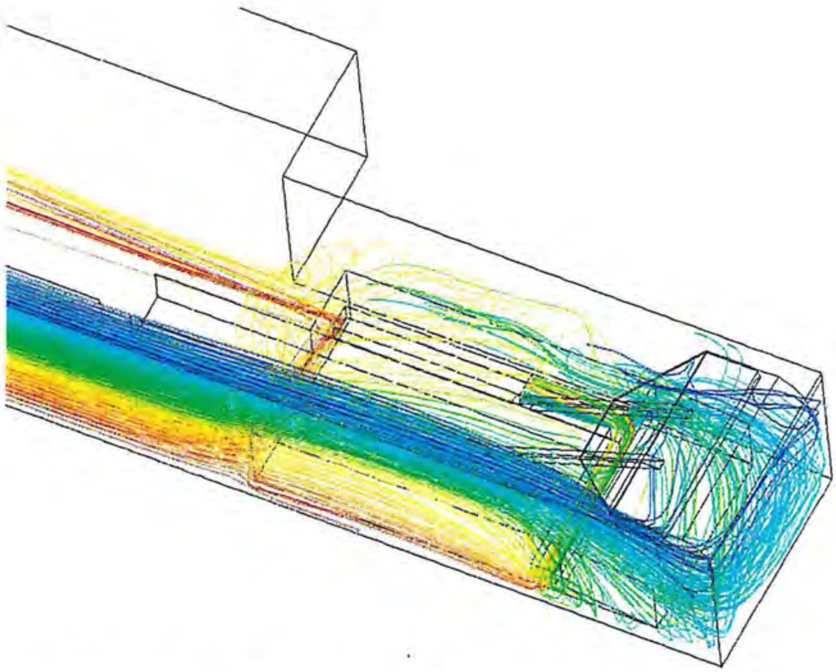


Figure 27: 3-D path lines illustrating the flow over a continuous miner in the 30 foot box cut scenario with a scrubber mass flow ratio of 0.83.

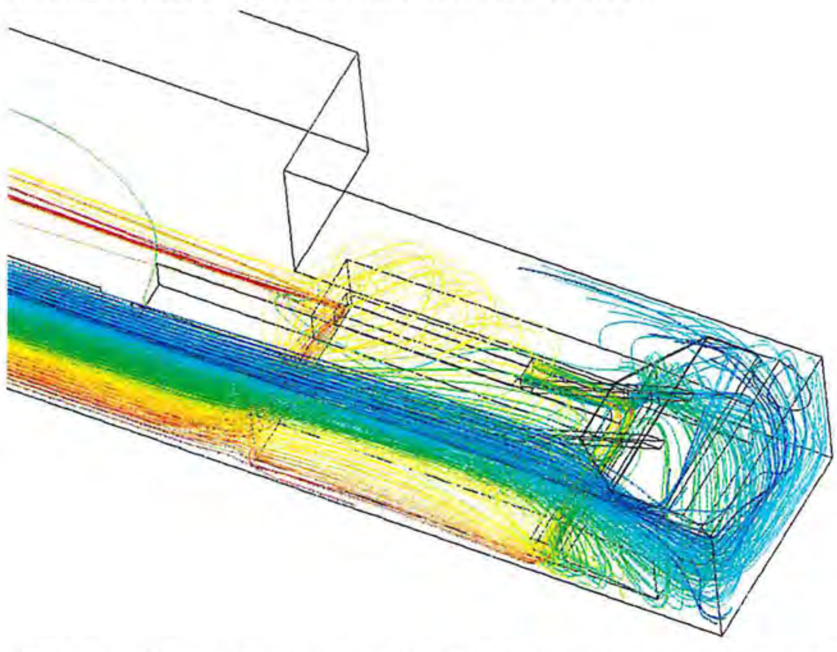


Figure 28: 3-D path lines illustrating the flow over a continuous miner in the 30 foot box cut scenario with a scrubber mass flow ratio of 0.95.

Title: CFD Analysis of Mine Face Ventilation Systems
Investigator: Dr. Andrzej Wala
Affiliation: University of Kentucky
City & State: Lexington,
Telephone: 6062572959
Award Number: 5R01CC415822-03
Start & End Dates: 7/1/1999-6/30/2003
Total Project Funding: $399,682
Program Area : Control Technology & Personal Protective Equipment

Final Report Abstract:
The use of extended (deep) cut mining with remotely controlled continuous miners is commonly utilized across 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 that higher levels of methane are liberated at the lace during coal extraction. This requires an increase in the air supply to the face of the heading for adequate methane dilution. Higher air velocities are a consequence, which, in turn, lead 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 recalculate and filter air in the immediate face area. These circumstances give rise to many concerns concerning miners' health and safety.
The former u.s. Bureau of Mines (currently NIOSH), Mine Safety and Health Administration, 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 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 potential to generate face ventilation designs without the previously mentioned disadvantages.
The major objective of this proposed project was to investigate the feasibility of developing a tool that facilitates the design of face ventilation systems to provide healthier and safer working environments for underground miners. The potential ability of this approach to reliably, quickly, and inexpensively test alternative face ventilation configurations will reduce the time between the generation of a design concept and its implementation by minimizing the tedious and costly experimental work required with existing design methodologies. If feasible, much improved designs for face ventilation systems should result

because it will be possible, for the first time, to screen a wide range of alternative configurations and quickly identify the most promising design concepts for a particular field application.

To achieve these objectives mentioned above, the authors conducted the following studies: (1) Development of a scaled physical model for examination of various ventilation scenerios and measurement of the airflow patterns using PIV (particle image velocimetry). These will be compared with simulations using commercial CFD software packages (CFD-2000, FLUENT, and CFX) with the objective of validating the CFD approach; (2) To refine modeling (numerical and scaled-physical) approaches, and to confirm that mathematical models and numerical method (CFD) can be used for design purposes of the face ventilation systems the full scale laboratory (gallery) tests will be performed at the Pittsburgh Research Center by the authors in conjunction with the MOSH, Dust and Ventilation Group; (3) based on the study performed the authors will down select a CFD software package, which could be, in cooperation with the software development company, converted to the specialized design tool utilized by the industry . It was determined that at the present time, CFD is not mature enough to be reliably used for ventilation design, though it can be used during the design process should sufficient validation be available.

Publications:

Wala AM, Stoltz JR, Hassan MI: Scale Modeling of A Mine Face Ventilation System for CFD Code Validation. Proceedings of the Third International Symposium on Scale Modeling, Nagoya, Japan, Septermber 10-13, 2000

Wala AM, Stoltz JR, Jacob JD: Numerical and Experimental Study of A Mine Face Ventilation System for CFD Code Validation. Proceedings of the Seventh International Mine Ventilatio Congress, Krakow, Poland, June 17-22, 2001

Wala MA, Jacob JD, Huang PG, Brown JT: 2002, A new approach to evaluate main face ventilation, Preprint 02-177:2002 SME Annual Meeting, Phoenix, AR February.Turner D, Wala MA, Jacob JD: 2002, Particle image velocimetry (PIV) used for mine face ventilation study, Proceedings: North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, June.

Wala MA, Jacob JD, Rangubhotla L, Watkins T: 2003, Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model, Perprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH February.

Wala MA, Jacod JD, Huang PG, Brown JT: 2003, New approaches to mine-face ventilation, Mining Engineering, Vol. 55, No. 3, March.

Wala MA, Jacob JD, Huang G, Rangubhotla L: 2004 How scrubbers help ventilate the face suring deep cut mining with a blowing curtain, Proceedings: 10th U.S. Mine Ventilation Symposium/North American, Anchorage, AL, May.

Reports

Brown, J. T., 2002, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, June.

Brown, J. T., 2003, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, October.

Theses

Rangubhotla, Lavanya, 2004, "Investigation of Characteristics of Bounded Wall Jets in Dead End Mine Headings", MS Thesis, December.

Publications

Journal and Conference Publications

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

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2002, "A new approach to evaluate mine face ventilation," Preprint 02-177: 2002 SME Annual Meeting, Phoenix, AR, February.

Turner, D., Wala, M.A., Jacob, J.D., 2002, "Particle image velocimetry (PIV) used for mine face ventilation study," Proceedings: North American/9th U.S. Mine Ventilation Symposium, Kingston, Ontario, June.

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

Wala, M.A., Jacob, J. D., Rangubhotla, L., and Watkins, T., 2003, "Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model," Preprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH, February. (Accepted for publication in Mining Engineering.)

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2003, "New approaches to mine-face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J.D., Huang, 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/ North American, Anchorage, AL, May.

Publications Included in the Report

Wala, M.A., Jacob, J.D., Huang, P.G., and Brown, J. T., 2003, "New approaches to mine-face ventilation," Mining Engineering, Vol. 55, No. 3, March.

Wala, M.A., Jacob, J. D., Rangubhotla, L., and Watkins, T., 2003, "Evaluation of an exhaust face ventilation system for 20-foot extended cut using scaled physical model," Preprint 03-148: 2003 SME Annual Meeting, Cincinnati, OH, February. (Accepted for publication in Mining Engineering.)

Wala, M.A., Jacob, J.D., Huang, 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/ North American, Anchorage, AL, May.

Brown, J. T., 2002, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, June.

Brown, J. T., 2003, "Prediction of mine face ventilation using Computational Fluid Dynamics," Progress Report, October.