NUMERICAL AND EXPERIMENTAL STUDY OF A MINE FACE VENTILATION SYSTEM FOR CFD CODE VALIDATION

A.M. Wala
Department of Mining Engineering
University of Kentucky
Lexington, Kentucky – USA

J.R. Stoltz
Department of Mining Engineering
University of Kentucky
Lexington, Kentucky – USA

J.D. Jacob
Department of Mechanical Engineering
University of Kentucky
Lexington, Kentucky – USA

ABSTRACT

The use of extended cut mining with remotely-controlled continuous miners is increasing rapidly across the U.S. coal industry. The accompanying higher advance rates present problems such as higher levels of methane liberation and more dust generation at the face area during coal extraction. These circumstances give rise to many questions concerning miner health and safety.

The design of an adequate mine face ventilation system, for the scenarios described above, requires considering the system in a three-dimensional, rather than a two-dimensional, manner. Computational Fluid Dynamics (CFD) is a promising numerical simulation method that has the potential to solve three-dimensional fluid flow problems and to aid in the design of proper face ventilation arrangements. Validation of these results with experimental data is required before any CFD code can be used for design purposes.

In this paper the authors will present a series of CFD simulations representing various face ventilation scenarios. Experimental data using a scaled (1:15) model in conjunction with the Particle Image Velocimetry (PIV) technique is being used for preliminary validation of these CFD results.

KEYWORDS

CFD, PIV, face ventilation, scale modeling

INTRODUCTION

In recent years, the refinement of continuous miner technology in underground coal mines in the United States (including the use of remote control, deep cuts, satellite or integrated roof bolting, and continuous haulage) has led to highly productive and efficient operations. However, this high production presents environmental problems, in that more dust is generated and higher levels of methane are liberated per unit time at the face during coal extraction. Many times this means a large quantity of air is needed in the face of the excavation heading for dilution. But this must be done with care. More air means higher air velocities, which typically lead to the entrainment of more dust.

Several ventilation techniques have been used in conjunction with these modern mining methods (Goodman, *et. al.*, 1990) including; (1) machine-mounted scrubbers, (2) extendable line curtains/flexible tubing, (3) jet fans, and (4) a combination of these methods. At many operations employing these technologies, increased dust entrainment has been countered by the use of large capacity scrubbers mounted on the continuous miners (Taylor, *et. al.*, 1996). These scrubbers have a significant influence on the airflow distribution (flow patterns) in the heading and the design of their air performance. Additionally, the location of the scrubber inlet and outlet ports must be done with great care, for the quality of the face environment to be satisfactory with regards to the health and safety (methane dilution) of mine personnel.

Because of the increasing use of scrubbers in modern continuous miner systems and also the increasing number of coal mining units that operate under more stringent dust standards due to excessive quartz in the mine dust, the design of a balanced ventilation system requires considering the system in three rather than two dimensions. Both theoretical and experimental methods are available for obtaining useful information regarding the design of such systems. However, theoretical methods are limited in utility due to inherent simplifications, whereas experimental methods are faced with high costs and the low availability and limitations of equipment for conducting detailed experiments.

In contrast, CFD is a rapidly developing numerical technique by which complex fluid flow problems can be solved on a digital computer (Brunner, 1995, Oberholzer and Meyer, 1995, Sullivan and Van Heerden, 1993, Wala, *et. al.*, 1996, Wala, *et. al.*, 1997, and Wala, et al., 2000). Many design alternatives can be explored in a short period of time, and the detailed three-dimensional results can be viewed graphically. This approach gives a better understanding of the flow behavior. During the interactive design process, this feedback often suggests design modifications that lead to a much better performance. The detailed information on airflow patterns established by CFD is usually readily rationalized, which leads to greater confidence in the final solution. However to develop this confidence, validation studies are required to ensure realistic numerical results.

COMPUTATIONAL FLUID DYNAMICS TECHNIQUE

Computational Fluid Dynamics (CFD) is a technique for modeling fluid flow problems both economically and within a relatively short time. CFD can be described (Anderson, 1995) as "the art of replacing the partial derivatives in the fluid motion equations with discretized algebraic form, which, in turn, are solved to obtain the flow field values at discrete points in time and/or space." CFD embraces a variety of technologies, including mathematics, computer science, engineering, and physics. These disciplines are brought together to provide the means of modeling fluid flows. CFD shows great promise as a design tool and has found wide applications in many different fields of engineering.

The application of CFD in mine ventilation could help engineers analyze complex air flow fields, contaminants transport, and design more economical and safer systems. In the last few years, some attempts have been made to solve mine ventilation problems with the CFD technique, and numerous papers have been published concerning these studies. These attempts were made 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 been developed to the extent that could really benefit the health and safety of miners. During the last three years in the Department of Mining Engineering, University of Kentucky, CFD has been used extensively for the detailed study of air flows across main airways of mine ventilation systems (Wala, *et. al.*, 1996 and Wala, *et. al.*, 1997). This ongoing research requires validation of the numerical results with experimental data to ensure that the model assumptions and numerical results are accurate (Wala, *et. al.*, 2000).

There is a need for extensive validation research to prove that CFD is a proper tool for the study of face ventilation systems. To begin this effort the authors performed the following tasks:

- developed three-dimensional numerical models and performed computer simulations using a CFD technique,
- designed and built a physical scaled (1:15) model of selected face ventilation systems corresponding to the CFD simulations,
- performed a series of preliminary measurements to determine the velocity field (flow patterns) applying the Particle Image Velocimetry (PIV) technique, and
- compared the simulated and measured data for the purpose of initial CFD code validation.

PRELIMINARY DATA FROM CFD SIMULATIONS

For this initial study, the authors decided to investigate the flow pattern in the empty face area (no mining equipment present). Future investigations will take into account the presence of underground equipment in the study area. The four CFD simulations presented in this paper are of various coal mine face scenarios, solved with the FLUENT (finite volume based) software package. These three-dimensional numerical models are 1:15 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. The purpose of 1:15 scale simulations is to allow for direct comparison to the aforementioned scaled physical model. The construction of this physical model will be discussed later in this paper. Scaling was performed using dimensional analysis based on the Reynolds Number (125,000 based on the inlet parameters).

The flow in the study area has complicated features, including separation and recirculation, necessitating the need for a proper boundary 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. The CFD models contain between 500,000 and 800,000 control volumes (grid cells) and use the two equation κ-ε model for the Reynold's Averaged Navier-Stokes (RANS) equations.

These four CFD simulation scenarios are as follows:

- A 9.1m (30 ft) box-cut with the slab,

- A 9.1 m (30 ft) box-cut without the slab,
- An 18.3 m (60 ft) box-cut with a 9.1m (30 ft) slab, and
- An 18.3m (60 ft) box-cut without the slab.

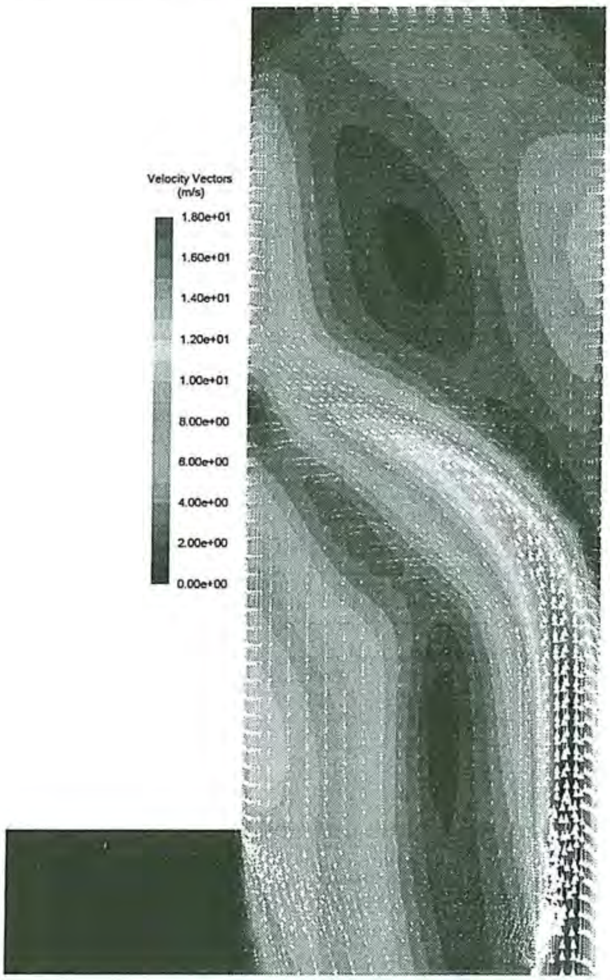

*Figure 1. **2-D** velocity vectors for the middle, horizontal plane for the 1:15 scaled, 9.1m (30 ft) box-cut with the slab*

*Figure 1a. **3-D** velocity vectors matching Figure 1*

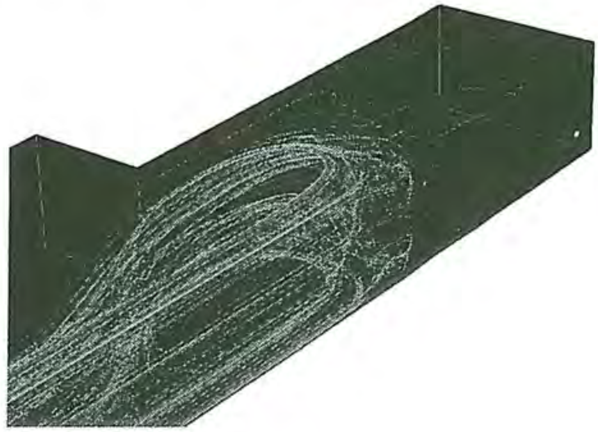

*Figure 1b. **3-D** pathlines corresponding to Figure 1*

Figures 1, 2, 3, and 4 show the air flow patterns found in the horizontal, middle plane of the face area for the first, second, third, and fourth simulation scenarios respectively.

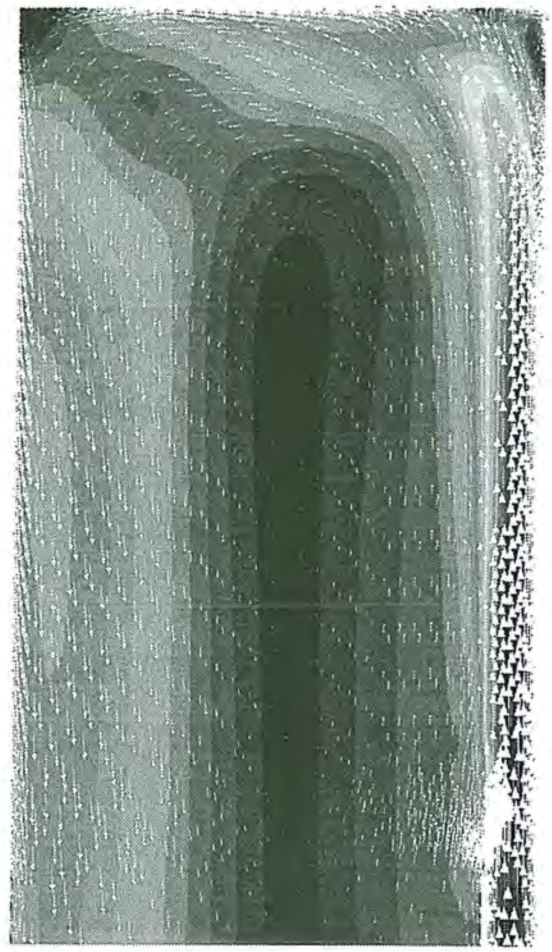

*Figure 2. **2-D** velocity vectors for the middle, horizontal plane for the 1:15 scaled, 9.1m (30 ft) box-cut without the slab*

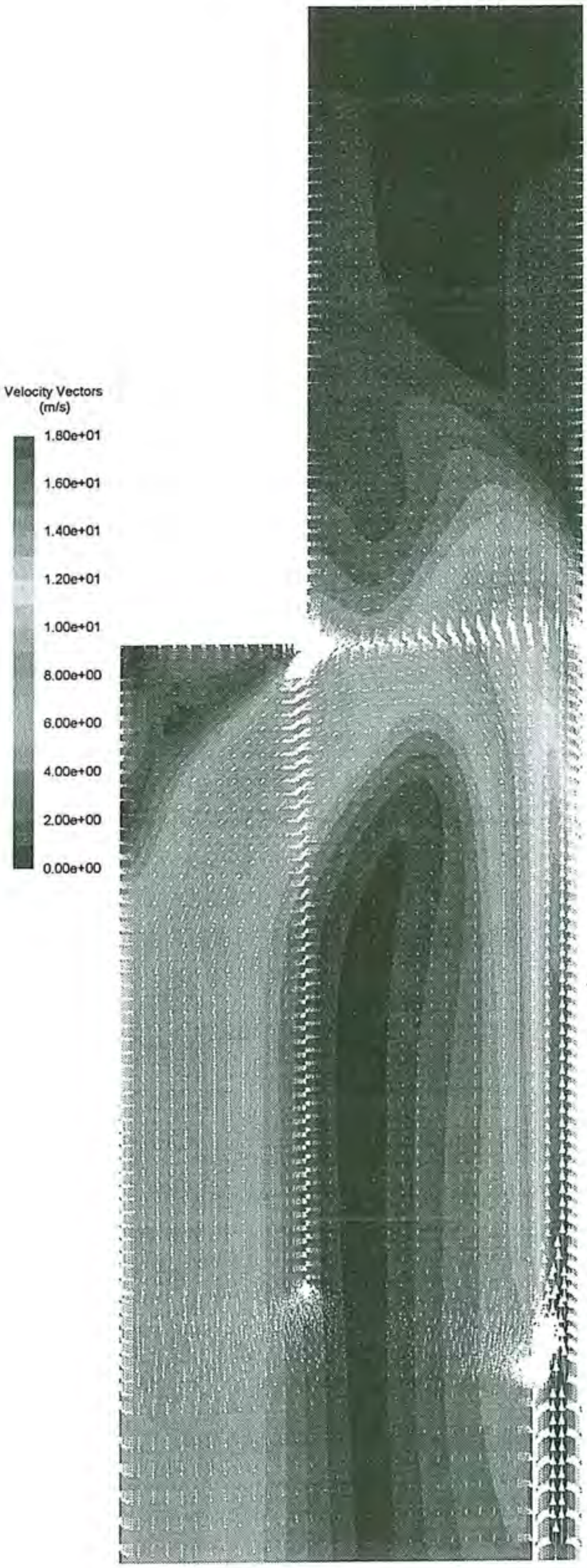

*Figure 3. **2-D** velocity vectors for the middle, horizontal plane for the 1:15 scaled, 18.3 m (60 ft) box-cut with a 9.1m (30 ft) slab*

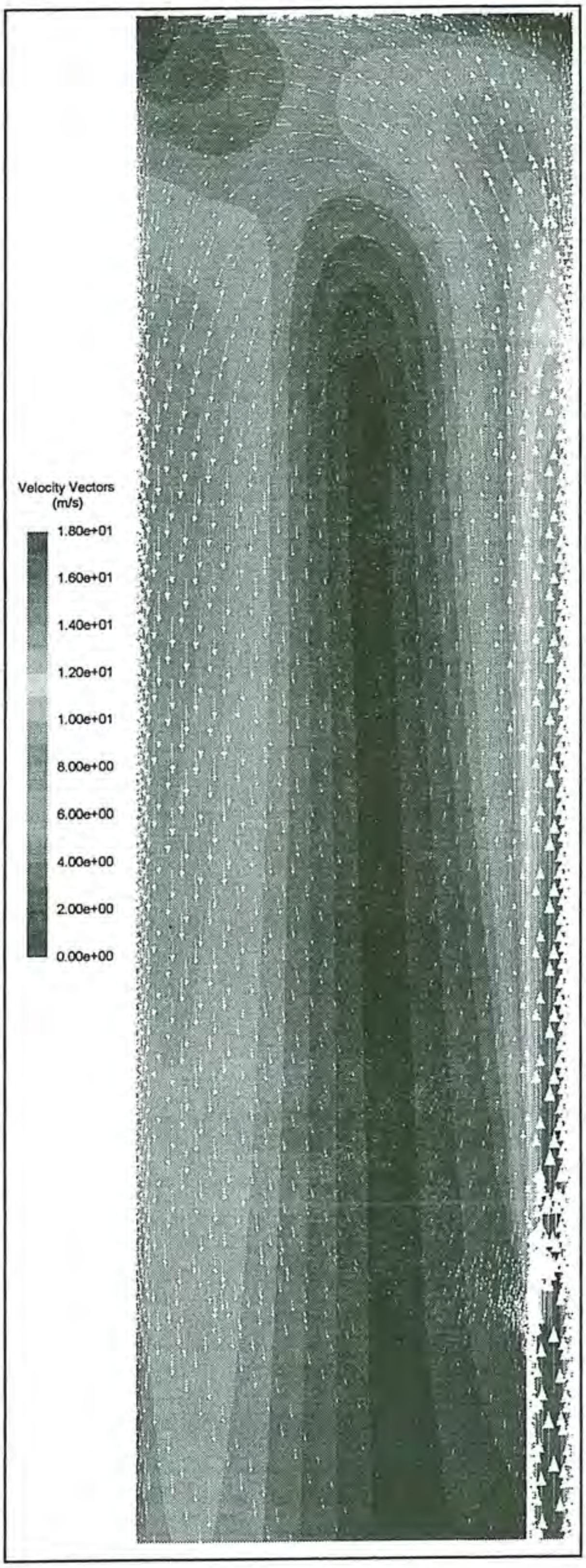

*Figure 4. **2-D** velocity vectors for the middle, horizontal plane for the 1:15 scaled, 18.3 m (60 ft) box-cut without the slab*

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, representative of the CFD simulations, was designed and built out of transparent Plexiglas, see Figure 5. Scaling of the geometry was performed using dimensional analysis based on the Reynolds Number. The model consists of three significant parts, see Figure 6: (1) middle section which represents an ordinary mining entry with a partition (line brattice) which divides the entry into a narrow and a wide airway, (2) working section which represents a mine face area, where movable Plexiglas walls can be used to replicate many different ventilation arrangements (from zero to 18.3 m (60 ft) deep box-cuts, with or without slab, etc.), and (3) intake/return section, where the ventilation system can be arranged as either blowing or exhausting by changing the fan location.

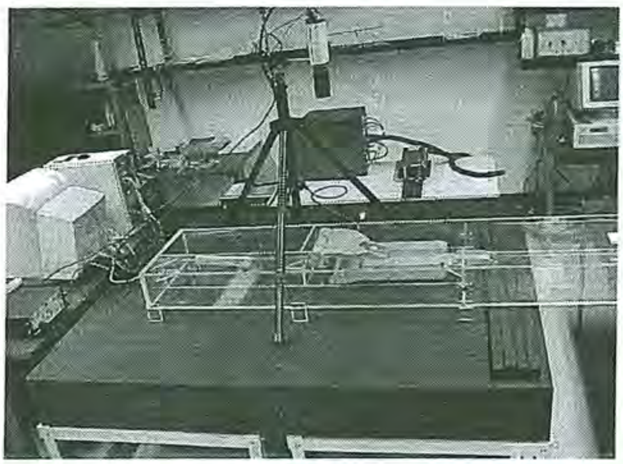

Figure 5. The physical model of the mine face area and the PIV system

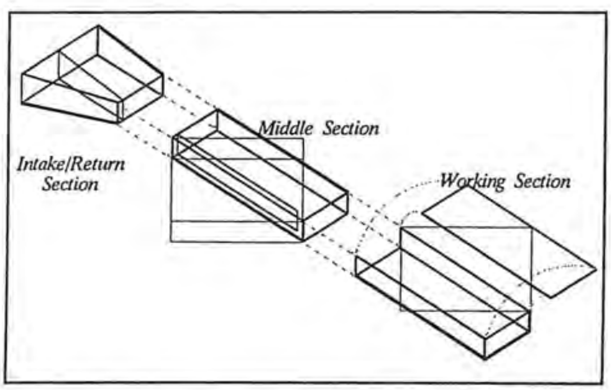

Figure 6. 3-D drawing of the physical model

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. Due to the relatively high air velocity in the model, a pulse laser system was required. PIV is a laser sheet technique for measuring two velocity components over an extended two- dimensional area. To perform these measurements, tracer particles must be added to the flow field. The laser-light sheet illuminates these particles twice within a short, controlled time interval. The reflected (scattered) light is recorded either on a single frame or on a sequence of frames by a CCD camera. In order to get the particle images during the light pulses, the computer software captures and stores these images in the form of numerical files. These digital images are divided into small sub-areas (interrogation areas) for analysis. The displacement vector for each interrogation area is determined by means of statistical methods (auto-correlation or cross-correlation). It is assumed that all particles within one interrogation area have moved homogeneously between the two images. The projection of the flow vectors into the plane of the light sheet (two component velocity vector) is calculated taking into account the time delay between the two laser pulses and the image magnification factor. For more details see references (Adrian, 1996, Grant, 1994, and Raffel, *et. al.*, 1998).

The integrated PIV system being used in this study, seen in Figures 5, 7, and 8, is comprised of the following components: (1) laser and light sheet optics, (2) image capture, (3) synchronizer, (4) smoke particle generator, and (5) image analysis and display.

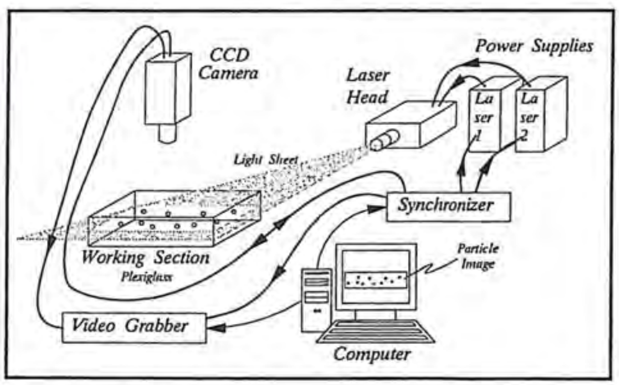

Figure 7. Schematic of the integrated PIV system

1. Laser and light sheet unit

A New Wave, dual head, Nd:YAG Laser (532-nm wavelength with a 15 Hz repetition rate) is used to illuminate the particles. Using a double pulsed laser allows for easy control of the duration time between the laser pulses. The light sheet optics include both a cylindrical and a spherical lens. The cylindrical lens diverges the laser beam into a sheet, and the spherical lens controls the light sheet thickness (about 1 mm at the focal length).

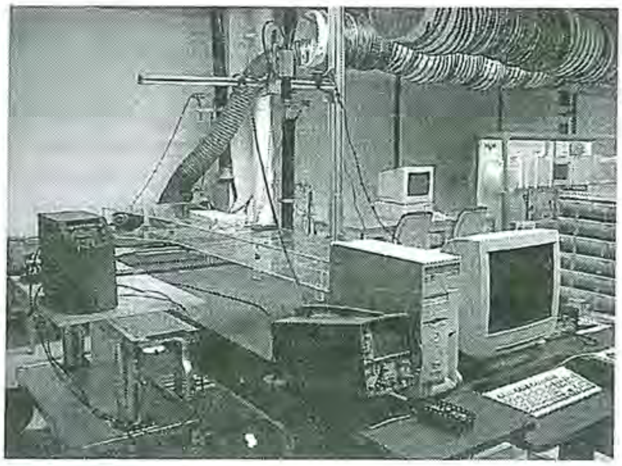

Figure 8. Laboratory arrangement of the PIV system and physical model

2. Image capturing

The particle images are recorded using a Roper Scientific-Kodak ES1.0, 10 bit, dual channel digital camera, which contains a square, mega-pixel array. The ES1.0 is capable of taking a pair of images with a controllable delay time between the pair ranging from 1 microsecond to 1 millisecond.

3. Synchronizer

The synchronizer controls the timing of all the PIV system components (laser triggering, duration time between laser pulses, and camera triggering) and links the PIV system with the computer/frame grabber.

4. Smoke Particle Generator

A SAFEX, oil based, smoke particle generator was used to make 1.0 μm size particles. This generator was adjustable, whereby the density of the smoke could be controlled.

5. Image Analysis and Display

This consists of an EPIX frame grabber and array processing software for performing the Fast Fourier Transforms and correlation. EPIX software can display and analyze the recorded particle images on the computer monitor, while the array processing software creates flow field vectors for a specific selected interrogation zone using either the auto-correlation or the cross-correlation method. A two-frame cross-correlation method with frame straddling is used for this study.

EXPERIMENTAL DATA FROM PIV STUDY

Future PIV work will utilize the system described above, which was designed specifically for this project. The preliminary PIV findings presented in this paper were gathered using a similar system made by TSI. The physical model of the face ventilation system was arranged for a 9.1 m (30 ft) box-cut with the slab. Due to the CCD camera's resolution, the average size of the smoke particles (approximately 1.0 μm), and the large velocity gradients, interpreting the airflow pattern for the entire face area (245 mm x 610 mm) as one frame is not currently possible. Therefore, the area was subdivided into 21 smaller regions and the flow patterns are developed for each, then combined together. To achieve a good transition between each region, the images overlap each other, see Figure 9. This process must be repeated for each two-dimensional plane of interest, using the TSI/PIV system. An example flow pattern for a horizontal plane is shown in Figure 10. For comparison, Figure 11 shows a CFD flow field.

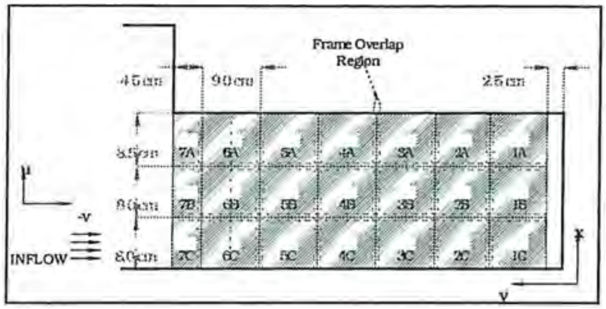

Figure 9. Plan view of the face area showing the 21 smaller regions for a 9.1m (30 ft) box-cut with the slab

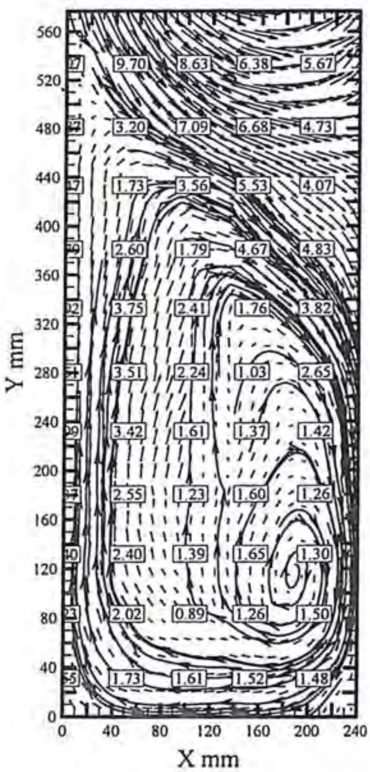

*Figure 10. PIV **2-D** velocity vectors for a horizontal plane of the 1:15 scaled, 9.1m (30 ft) box-cut with the slab still in place (Wala, et.al., 2000)*

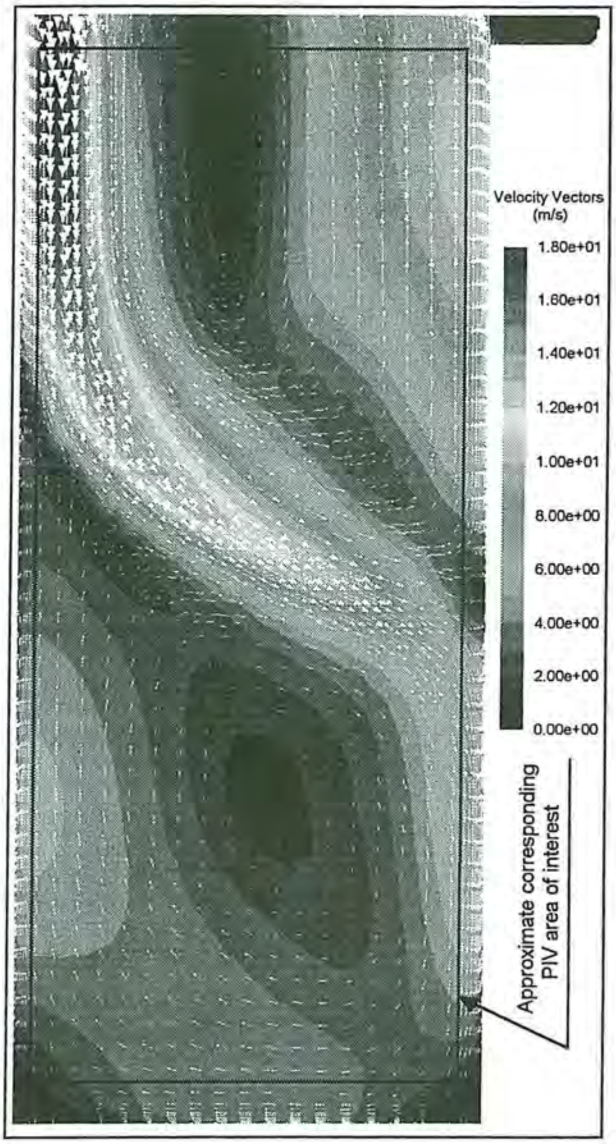

*Figure 11. CFD **2-D** velocity vectors for the middle, horizontal plane for the 1:15 scaled, 9.1m (30 ft) deep box-cut with the slab still in place*

CONCLUSIONS

- The PIV and CFD derived flow patterns are similar as can be seen in Figures 10 and 11.
- The air distribution of the face area is complex. The two CFD simulations with the slab, see Figures 1 and 3, show extensive air recirculation zones in the face area, and preliminary PIV data supports this phenomenon, see Figures 10 and 11.
- Extensive CFD and PIV investigations, including statistical validation studies, will follow this preliminary work.

ACKNOWLEDGEMENT

This study was supported by the National Institute of Occupational Safety and Health (NIOSH), under Grant No. R01/CCR415822-02. A special thanks is also extended to Dr. M.I. Hasan, from the Mechanical Engineering Department of the University of Kentucky, for his help in collecting preliminary PIV data.

REFERENCES

Adrian, R.J., 1996, "Stratagies for Imaging Flow Fields with Particle Image Velocimetry", Proceedings of the 27th AIAA Fluid Dynamics Conference, New Orleans, Louisiana

Brunner, D.J., 1995, "Examples of the Application of Computational Fluid Dynamics Simulations to Mine and Tunnel Ventilation," Proceedings of the 7th U.S. Mining Ventilation Symposium, Lexington, Kentucky

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

Grant, I., 1994, "Selected Papers on Particle Image Velocimetry," SPIE Milestone Series MS 99, SPIE Optical Engineering Press

Oberholzer, J.W. and Meyer, C.F., 1995, "The Evaluation of Heading Ventilation Systems Through the Use of Computer Simulations," Proceedings of the 7th U.S. Mine Ventilation Symposium, Lexington, Kentucky

Raffel, M., Willert, C., and Kompenhans, J., 1998, "Particle Image Velocimetry," Springer

Sullivan, P. and Van Heerden, J., 1993, "The Simulation of Environmental Conditions in Continuous Miner Developments Using Computational Fluid Dynamics," Journal of the Mine Ventilation Society of South Africa, January

Taylor, C.D., Rider, J., and Thimons, E., 1996, "Changes in Methane Concentrations Using High Capacity Scrubber with Exhausting and Blowing Ventilation," Preprint No. 96-167, SME Annual Meeting, Phoenix, Arizona, March

Wala, A.M., Yingling, J.C., and Zhang, J., 1996, "Application of Computational Fluid Dynamics in Mine Ventilation," Proceedings of the '96 International Symposium on Mining Science and Technology, Xuzhou, Jiangsu, October, A.A. Balkema, Publisher

Wala, A.W., Yingling, J.C., Zhang, J., and Ray, R., 1997, "Validation Study of Computational Fluid Dynamics as a Tool for Mine Ventilation Design," Proceedings of the 6th International Mine Ventilation Congress, Pittsburgh, Pennsylvania, May

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

Proceedings

of the

7th International Mine Ventilation Congress

Editor
Stanisław Wasilewski
Research and Development Center EMAG

June 17-22, 2001, Cracow, Poland