Tunnelling and Underground Space Technology 33 (2013) 1–11

Contents lists available at SciVerse ScienceDirect

Tunnelling and Underground Space Technology

journal homepage: www.elsevier.com/locate/tust

Development of a remote analysis method for underground ventilation systems using tracer gas and CFD in a simplified laboratory apparatus

Guang Xu [a,*], Kray D. Luxbacher [a], Saad Ragab [b], Steve Schafrik [c]

[a] Virginia Tech, Department of Mining and Minerals Engineering, School of Engineering, Blacksburg, VA 24060, United States
[b] Virginia Tech, Department of Engineering Science and Mechanics, School of Engineering, Blacksburg, VA 24060, United States
[c] Virginia Tech, Virginia Center for Coal and Energy Research, Blacksburg, VA 24060, United States

ARTICLE INFO

Article history:
Received 15 March 2012
Received in revised form 10 August 2012
Accepted 7 September 2012
Available online 15 November 2012

Keywords:
Mine incident
Underground mine ventilation
CFD modeling
Tracer gas
Gas chromatography

ABSTRACT

Following a disaster in a mine, it is important to understand the state of the mine damage immediately with limited information to manage the emergency effectively. Tracer gas technology can be used to understand the ventilation state remotely where other techniques are not practical. Computational fluid dynamics is capable of simulating and ascertaining information about the state of ventilation controls inside a mine by simulating the airflow and tracer distribution. This paper describes a simulation of tracer gas distribution in a simplified laboratory experimental mine with the ventilation controls in various states. Tracer gas measurements were taken in the laboratory experimental apparatus, and used to validate the numerical model. The distribution of the tracer gas, together with the ventilation status, was analyzed to understand how the damage to the ventilation system related to the distribution of tracer gases. Detailed error analysis was performed and the discrepancies between experimental and simulated results were discussed. The results indicate that the methodology established in this study is feasible to determine general ventilation status after incidents and can be transferred to field experiment. Because it is complex to simulate the actual condition of an underground mine in a laboratory, the model mine used is simplified to simulate the general behavior of ventilation in a mine. This work will be used to inform planned on-site experiments in the future and the proposed methodology will be used to compare collected and simulated profiles and determine the general location of ventilation damage at the mine scale.

1. Introduction

After a severe underground mine incident, such as a roof fall, outburst, water inrush, or explosion that may cause tunnel collapse, underground information must be gathered immediately to estimate the extent of damage for rescue and recovery operations. In these situations, communications between underground miners and rescuers on the surface may be tenuous at best, because very few commercially available communications systems have been proven to meet the basic requirements for emergency communication (Gurtunca, 2008). Some alternate methods can be used to gather information, such as collection of air samples from boreholes, utilization of a video camera via borehole to visualize underground status, and utilization of rescue robots underground if possible. However, none of these methods are reliable and efficient enough to stand alone. In an emergency situation, accurate information regarding the mine status is invaluable not only to save miners' lives, but also to help decision makers manage the emergency effectively, and to increase safety for rescuers.

In some incidents, such as explosions, all the communication lines maybe damaged and collapse may occur with the location difficult to pinpoint from the surface. However, the airflow paths and ventilation patterns will change according to the location of damage. Therefore, the location of damage can be approximately determined by remote measurement of ventilation parameters. Due to the complexity of the ventilation system, employment of the tracer gas method is an effective means and has been used in many situations where conventional techniques are inadequate or cannot be effectively employed (Klinowski and Kennedy, 1991; Thimons and Kissell, 1974). Numerical simulations using Computational Fluid Dynamics (CFD) can be used to model the ventilation status and the data from tracer gas measurement allow for further analysis, prediction, and confirmation of the underground ventilation status together with the location of the damage.

Tracer gas was first used in the building ventilation systems in the 1950s (Dick, 1950) and has been widely used for ventilation analysis both in buildings and underground mines (Kennedy et al., 1987). Tracer gas based ventilation measurement is an effective method to detect air flow routes, estimate air flow quantity, and other complex ventilation problems (Arpa et al., 2008; Thimons and Kissell, 1974). Sulfur hexafluoride (SF_6) is widely

* Corresponding author. Tel.: +1 540 553 5288.
E-mail address: xuguang0530@gmail.com (G. Xu).

0886-7798/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.tust.2012.09.001

accepted as a standard mine ventilation tracer (Kennedy et al., 1987), because it can be detected in low concentrations, is nontoxic, odorless, colorless, chemically and thermally stable, and does not exist naturally in the environment (Thimons and Kissell, 1974). The applications of tracer gases in underground mines include measurement of turbulent diffusion (Arpa et al., 2008), methane control (Mucho et al., 2000), study of mine ventilation recirculation of return into intake air, transit flow times through stopped areas, effectiveness of auxiliary fans, and estimation of volumetric flow rates (Thimons et al., 1974; Thimons and Kissell, 1974), air leakage investigation, and evaluation of dust control measures (Timko and Thimons, 1982).

In recent years, Computational Fluid Dynamics (CFD) has become a powerful tool and has been commonly used to model the underground mine air flow behavior and solve relative problems (Hargreaves and Lowndes, 2007; Ren and Balusu, 2010). It has been used in a number of areas, including modeling ventilation airflow patterns (Hargreaves and Lowndes, 2007; Heerden and Sullivan, 1993), study and control of coal spontaneous heating and underground fire (Yuan and Smith, 2008; Ren and Balusu, 2005; Huang et al., 2001), optimization of gob inertisation (Balusu et al., 2002), dust control (Toraño et al., 2011; Heerden and Sullivan, 1993), and methane management (Karacan et al., 2006; Toraño et al., 2009). The combination of experimental measurement and CFD modeling of tracer gas has been used to study airflow and contaminant transport in indoor environments and other industrial applications (Cheong et al., 2003; Yang, 2004), but little research has been done to model underground tracer gas applications, especially the use of these techniques to model and analyze the ventilation and mine environment following an incident that alters the ventilation system.

CFD has also been used in many studies to investigate underground tunnel risks. Hua et al. (2011) used CFD to develop an optimal smoke control strategy for tunnel fires. The model was validated using the test results from a similar tunnel and an optimal smoke control strategy was found based on the model results. Se et al. (2012) used CFD to investigate the effect of an active fan group on the airflow structure and temperature distribution in a tunnel with varied fire sources. Gao et al. (2012) used Large Eddy Simulation to study the dispersion of fire-induced smoke in a subway station and the influence of natural and mechanical ventilation was investigated. Risks challenging underground mines are also common in underground tunneling and construction, particularly the methane explosion described in (Copur et al., 2011), the fires scenarios mentioned above, floods, and earthquakes. The proposed methodology can also be potentially applied to those situations to better understand the ventilation status remotely, and thus manage the emergency effectively with significant impacts on safety. Also, the tracer gas tests, sampling, and analysis techniques used in this study can be applied to underground tunnel ventilation survey to investigate ventilation efficiency, flow paths, and other related issues.

A CFD approach was used in this study due to the relative simplicity of the experimental apparatus. CFD can resolve details of flow features and tracer distributions. These will help, when we move our test to the field scale, to determine the optimum method and place to release and sample tracer gas. However, it is not practical to apply CFD to the entire mine due to its heavy demand on computational time. Ventilation network modeling is more practical in this situation, but it cannot resolve the detail of tracer gas behavior at the micro scale. Although the focus of this study is CFD modeling, a hybrid scheme will be investigated when this work is applied to the field, which combines the benefits of CFD and network modeling.

In this study, tracer gas (SF_6) was used in an experimental laboratory simplified model mine which was built according to a conceptual mine layout. A CFD model was developed to simulate the laboratory apparatus. Various states of ventilation patterns were controlled by valves in the experimental mine to simulate different ventilation scenarios after incidents. Tracer gas was released to the model mine at a constant rate. Air samples were analyzed to test the tracer concentration at different locations. The aim of this study is to use the experimental data to validate the CFD model, study the relationship between the tracer concentration and the location of incidents, and finally, through analysis of the air sample and the CFD model result, determine the general location of the ventilation damage. A preliminary version of this study was presented by Xu et al. (2011).

2. Experimental setup and measurements

2.1. Experimental apparatus

The laboratory mine model represents a simple conceptual mine shown in Fig. 1, in which the arrows indicate the normal air flow path and this state is referred as Case 1. It has one active panel and one gob panel, two regulators (V2 and V4) regulating the air flow into the panels, one stopping (V1) between the main entries, and five boreholes (P1–P5). Stopping damage at V1, a roof fall in the active panel at V3, and explosion damage in the gob panel at V5 are three possible incidents will be investigated in this study. The air flow to the gob was simplified in both the laboratory experiment and the CFD model in that the air simply flows around the gob; a permeable gob was not studied. It should be noted that this simplified experimental model mine was not used to represent a full scale mine, but rather to validate the CFD model, choose the best sample methods, and test the effectiveness of our methodology, which can later be used to conduct field experiments.

The experimental mine model was built, as shown in Fig. 2, using 0.05 m (2 in.) inside diameter PVC pipes and allows for experiments representing general flow paths of a typical coal mine shown in Fig. 1 under different ventilation statuses. The general dimension of the model is 6.63 m in length, 0.51 m in height, and 0.36 m in width. Air exhausts from the apparatus via a variable speed exhaust fan. Valves were used to represent stoppings, regulators, and damage due to incidents.

Pitot tubes and an electrical manometer were used to measure differential pressure at points P1–P4, and the results were used to calculate velocities at those points using the following equation (Klopfenstein, 1998):

$$V = 44.723 \times \sqrt{\frac{h_{\mathrm{kPa}}}{d_{COR}}} \quad (1)$$

where h_{kPa} is differential pressure measured by pitot tube and manometer in kPa, and d_{COR} is corrected air density in kg/m^3, which can be calculated from the following equation (Klopfenstein, 1998):

$$d_{COR} = 3.4834 \times \frac{P_B}{T_{\mathrm{K}}} \times \left(1 - \left(\frac{0.3783 \times \frac{RH}{100} \times P_s}{P_B}\right)\right) \quad (2)$$

where P_B is barometric pressure in kPa, T_{K} is absolute temperature in Kelvin, RH is relative humidity. P_s is partial pressure of water vapor at T_{K}, and can be calculated using the following equation:

$$P_s = 1.753 \times 10^8 \times e^{-5315.56/T_{\mathrm{K}}} \quad (3)$$

During this experiment, the measured barometric pressure is 94732 Pa, the temperature is 295 K, and the relative humidity is 29%. Therefore, using Eqs. (2)–(5), the calculated air density is 1.114 kg/m^3. This value is used in the experiment velocity calculation and the CFD model input.