Numerical Simulation of Ventilation Air Flow in Underground Mine Workings

S.M. Aminossadati
The University of Queensland, CRCMining, Brisbane, Australia

K. Hooman
The University of Queensland, Brisbane, Australia

ABSTRACT: In recent years, Computational Fluid Dynamics, CFD, has been commonly utilized in the mining industry to model the fluid flow behavior in underground mine workings. This paper uses CFD modeling to simulate the airflow behavior in underground cross-cut regions, where brattice sails are used to direct the airflow into these regions. Brattice sails are cost effective ventilation control devices for temporary or permanent use in underground mining. They can be used to deflect air into the unventilated areas such as cross-cut regions. Their design and installation is a fundamental issue for maintaining a sufficient supply of fresh air and achieving effective air circulation and contaminant removal. At the same time, they should have little impact on the mine ventilation system. This paper presents the results of a two-dimensional CFD model, which examines the effects of brattice length on fluid flow behavior in the cross-cut regions. The results of this study will assist in understanding the ventilation air behavior and in determining the optimum size of brattice sails, which provide a highly effective contaminant removal from the unventilated mine areas. This, in turn, helps the mine ventilation designers to meet the mine safety requirements.

1 Introduction

In underground mining, it is often necessary to provide fresh and cool airflow in unventilated areas such as cross-cut regions. Ventilating air is employed to dilute and remove undesirable or dangerous gases such as methane and to reduce the air temperature in these areas. Cross-cut regions are frequently used as crib rooms when their one end is sealed and the other end is open to the mine airways. They are also used for accommodating the mining equipment, electrical transformers, or drilling machineries. In many underground mining operations, it is common practice to direct and control the airflow into the unventilated areas such as cross-cut regions by means of hanging a thin, lightweight, and fire resistant cloth known as brattice sail. Brattice sails are hung against the ceiling across tunnel openings and are extended in the cross-cut regions, so that the airflow can be diverted away from the tunnels and flow into the cross-cut regions. Companies are continuously attempting to provide innovative products and solutions for a range of ventilation related applications of brattice, to be cost effective, flexible, durable, and tear and fire resistant. The structure of brattice sails and the influence of various arrangements and qualities of ventilation brattices on the distribution of air through various areas of underground mines has been extensively studied and well documented in the literature. Louw (1974) found a wide variety of airflow patterns for different brattice locations and qualities. Kissell and Matta (1979) described a proper ventilation system by means of a conventional brattice. Standish (1983) investigated the effects of different brattice types on the ventilation characteristics.

In a notable study, Taylor et al. (1992) evaluated the face ventilation for two face ventilation techniques that utilize either blowing brattice or a jet fan. The influence of brattice sails on the reduction of methane concentration in face ventilation was investigated by Smith and Stoltz (1991) and Thimons et al. (1999). They evaluated the effectiveness of a blowing face ventilation system in controlling methane liberation in the face area while ventilation was carried out with brattice sails with different configurations, qualities, and distances from the face. They argued that the methane concentration is affected by the quantity and quality of fresh air reaching the face, as expected. Moreover, it was shown that the selection of the brattice setback distance could significantly improve ventilation to the end of box cut.

Lee et al. (1996) investigated the effects of brattice characteristics on the effectiveness of ventilation on reductions in concentration of diesel particulate matter. They developed computer models to study the generation, transfer, and distribution of diesel particulate matter in coal mine sections. They found that the reduction of diesel particulate matter is significantly influenced by ventilation schemes using vent tubing and brattice, exhaust locations and direction of ventilation.

Literature indicates a number of studies on the effectiveness of brattice sails in controlling respirable dust. Tien (1988) and Potts and Jankowski (1988) studied the airflow pattern in the working face and showed that the usage of a brattice is essential in order to avoid recirculation and control respirable dust in the face area. They argued that the heading airflow was maximized by properly installing the brattice curtain.
Recently, researchers have studied the effects of brattice setback distance on the flow-field behavior in underground workings. Goodman and Pollock (2004) studied the effects of line brattice setback distance and showed that changes in setback distance affected return airway dust levels and untransformed gas levels around the cutting drum. They concluded that changes in the brattice flow quantity and setback distance impacted effectiveness of the face ventilation. Taylor et al. (2005) developed a test system to measure the airflow profiles at locations between the face and the end of the brattice for different brattice setback distances, intake flow quantities, and entry widths. They found that the entry geometry had a significant effect on airflow patterns. Goodman et al. (2006) used a line brattice for a series of laboratory evaluations to examine the impact of brattice setback distance. They showed that increasing brattice setback distance often elevated dust levels, likely due to the reduction of the amount of airflow reaching the face.

Along with the development in experimental studies (some of them have been mentioned above), there have been a growing number of numerical studies addressing the problem. One of the important reasons for using a computational model is the superlative illustrative presentation of the results, which allows designers to have an increased understanding of the problem. A good understanding of the fluid-flow behavior results in improving the accuracy and, consequently, safety of designs with minimum cost of the investigation. However, comprehensive validation process of CFD modeling against actual mining experiments is an important issue in the application of the CFD results in appropriate designing of mine ventilation systems. Simulation of airflow in underground mine workings is one example of the application of CFD modeling in mining industry. For instance, Van Heerden and Sullivan (1993) applied CFD in evaluating and improving environmental conditions in continuous miner and road header sections. Even though, they did not validate their model, the results presented a good understanding of the movement of dust and methane in the predicted flow fields. Srinivasa et al. (1993), however, validated their numerical results, from commercial software, using field trial measurements and obtained a good degree of agreement. Of particular interest were air velocities and the effect of dust control techniques on dust concentrations at a typical longwall face. Comparing their results with analytical predictions, Brunner et al. (1995) used CFD to evaluate the effects of varying the airflow rate in a ventilated airway on the layering along the roof of smoke and hot gases due to a vehicle fire. In an interesting study, Wala et al. (2003) reported that the proper development of the computer code, the application of software, and selection of the grid size and turbulence modeling are crucial when CFD is considered as a tool to evaluate the mine face ventilation. Silvester et al. (2004) used a k-ε model to account for turbulence in their three dimensional model for the ventilation system within an underground crushing installation.

Absent in the above studies; however, is a numerical analysis of the airflow behavior within a cross-cut region of an underground mine. The problem is of particular interest not only for providing the ventilation air to those hard-to-ventilate regions but also for washing the contaminants out of these regions. This paper aims to address the importance of this crucial safety issue in underground mine environment.

2 Model Development

The geometry shown in Figure 1 represents a horizontal two-dimensional mine airway including a cross-cut region, which is being ventilated using a brattice sail, installed in the main heading and is extended in the cross-cut region. To simulate air flow behavior, a fully implicit segregated solver, employing a cell-centered control volume technique is implemented. The standard k-ε turbulence model employed in previous experimentally validated mine ventilation studies such as the study by Wilcox (2006) and Hargreaves and Lowndes (2007) is used in this work. Our main goal is to study the effects of length of the brattice sail on the flow-field within the cross-cut region at various ventilation air quantities. In most practical cases, the air temperature remains almost the same throughout the flow region so that there is no point in doing the thermal analysis of the problem. For the same reason, air property variation (with the temperature) was neglected. A simple scale analysis will also show that, under working circumstances, air can be considered as an incompressible fluid. The fluid inlet velocity (u0) and opening width (H) are used as velocity and length scale, respectively. The flow filed consists of an entrance and exit section, each of which of 2H length, with the cross-cut of width H and depth L=3H, as depicted in Figure 1. The wall-brattice distance is W=H/10.

![Figure 1. A schematic diagram of the channel](image)

...
To observe different combinations of the key parameters, the inlet velocity \( u_0 \) and the length of the brattice inside the cross-cut region \( h \) were varied and the results of the flow field were analyzed. The corresponding Reynolds numbers: 
\[
Re = \frac{\rho u_0 D_h}{\mu}
\]
(1) to three different airflow velocities were \( (Re = 3.2 \times 10^6, 1.6 \times 10^6 \text{ and } 0.3 \times 10^6) \) inline with practical underground mine requirements. In the above equation \( \rho \) is the air density \((\text{kg/m}^3)\), \( D_h \) is the hydraulic diameter of the mine airway cross-section \((\text{m})\) and \( \mu \) is the air viscosity \((\text{Pa.s})\). For every value of Reynolds number, four different scenarios were investigated corresponding to different lengths of the brattice: (A) \( h/H=0 \) (B) \( h/H=1.5 \) (C) \( h/H=2.5 \) (D) \( h/H=2.75 \). This has been illustrated by Figure 2.

\[
\text{Figure 2. Different configurations of the brattice}
\]

3 Governing Equations

The governing equations of all fluid flow equations can be expressed in the following conservation form of the transport equation, Patankar (1980):
\[
\frac{\partial \rho \phi_t}{\partial t} + \nabla (\rho_t \vec{V}_t \phi_t) - \nabla (\Gamma_{\phi_t} \text{grad} \phi_t) = S_{\phi_t}
\]
(2)

Where, \( \phi_t \) represents the variable of interest, \( \rho_t \) is the instantaneous air density, \( \Gamma_{\phi_t} \) is the diffusive coefficient and \( S_{\phi_t} \) is the source rate per unit volume. From left to right these terms are the transient term (which is absent in our study for the steady state condition), the convective term, the diffusive term, and the source term.

4 Numerical Details

Commercially available software CFD-ACE (ESI Software), which uses the SIMPLEC algorithm of Van Doormal and Raithby (1984) to handle the pressure-velocity coupling, is used to solve the mass continuity, momentum, and two additional transport equations for turbulent kinetic energy, \( k \), and its dissipation, \( \varepsilon \).

The computational domain was generated with quadratic mesh using the commercial package CFD-GEOM (ESI Software) that is typically used in conjunction with the commercially available finite volume flow solver CFD-ACE.

Non-uniform grids, with cluster in near-wall regions, were controlled in CFD-GEOM using curvature criterion, transition factor, and maximum and minimum cell sizes. These values were 30 degrees, 1.1, 0.1, and \( 3 \times 10^{-5} \) respectively. The results were found to be accurate when 18942 cells were applied for the most stringent case, being case D with \( h/H=2.75 \). Grid-independence was tested by control runs on a finer grid with 22178 cells that produced consistent results (with a maximum error being less than 3%). Hence, finer grids were not used in reporting the results. The convergence criterion (maximum relative error in the values of the dependent variables between two successive iterations) in all runs was set at \( 10^{-5} \).

The results of this study were cross-validated for case A (no brattice) with the results of another commercially available CFD software FLUENT (Bagheri, 2006). As an example, Figure 3 is presented to compare the stream-wise velocity distribution from two solvers at \( x/H=1.5 \).

\[
\text{Figure 3. Cross-validation of two numerical solvers, ACE (used in the present study) and FLUENT.}
\]

5 Results and Discussions

Figures 4-6 present the stream function \((\psi/\psi_{\text{max}})\) patterns for \( Re= 3.2 \times 10^6, 1.6 \times 10^6 \text{ and } 0.3 \times 10^6 \) respectively. Each figure illustrates the results for four different lengths of brattice. The results show that at all values of \( Re \), installation of a brattice increases the magnitude of stream function within the cross-cut region. This is manifested by the strength of the flow field in the cross-cut as opposed to nearly subsided flow in the cross-cut for the no-brattice case. The results also indicate that for a constant brattice length, varying Reynolds number has little effects on the stream function patterns. This is mainly because the stream function is normalized by its maximum value.
Figure 4. Stream Function patterns ($\psi/\psi_{\text{max}}$) in the cross-cut region at $Re=3.2\times10^6$.

Figure 5. Stream Function patterns ($\psi/\psi_{\text{max}}$) in the cross-cut region at $Re=1.6\times10^6$. 
In the case of no brattice, there is a flow circulation within the cross-cut region and no contaminants can be removed from this region. It is mainly because without the brattice the flow circulates in the cross-cut very similar to a (long) cavity flow. The fluid prefers to enter the cavity from the regions close to the right wall, where stream-wise velocity puts on lower values compared to the left wall. Observe that while the dimensionless peak velocity near the right wall for case A is less than that of other cases, it is almost independent of the Reynolds number.

After introducing the brattice, it is clear that the flow tends to opposite directions. Some portion of the airflow is returned to the mainstream after washing out the contaminants. However, some portion of the airflow forms an anti-clock-wise circulation within the cross-cut at all three different Reynolds numbers. The intensity of this circulation reduces as the brattice extends towards the end wall of the cross-cut. That is to say that as the brattice length increases, the fluid tends to wash out the cross-cut region more effectively. This can be verified by comparing the degree of stream function deflection (from the right wall to the left). This, in turn, implies that at the vicinity of the right wall, the mass flow rate puts on higher values as the brattice length is increased. This behavior is evident at all three different Reynolds numbers.

Due to the presence of the brattice with a circular curvature, the stream-wise component of velocity changes direction to form a y-component in the cross-cut region. Therefore, it is of particular interest to observe the y-velocity in this section. Besides, the y-component of velocity is acting, more or less, like an impinging jet targeting the bottom wall, and washing out the contaminants from this region. It is also very useful to analyze the maximum value of y-velocity as it represents the fluid rotation within the cross-cut region. The normalized y-velocity in the cross-cut region proved to behave similarly at different Reynolds numbers.
As an example, the normalized y-velocity was only plotted at Re= 3.2×10^6 in Figure 7. In this Figure, the normalized y-velocity is plotted outside of the brattice region at a fixed location half way through the cross-cut (y/H=1.5). This study helps to understand and compare the flow field behavior at the same location in the cross-cut for different brattice lengths: h/H=0, 1.5, 2.5, and 2.75. The results confirm that there is a clockwise flow circulation in the case with no brattice and an anti-clockwise circulation with the brattice. In general, the results of normalized y-velocity within the cross-cut region indicate a slight difference for various brattice lengths. A comparison between the maximum absolute values of y-velocity near the right wall of the cross-cut indicates that in case D, the fluid has a less tendency to form a circulation comparing to cases B and C. When the absolute values of maximum velocity near the right wall of the cross-cut are compared, it can be concluded that flow tends to return and forms a circulation within the cross-cut for h/H=1.5 compared to other cases. Similar results can be observed in Figure 8 where, maximum y-velocities near the right wall of the cross-cut are compared for different brattice lengths and Reynolds numbers.

![Figure 8](image8.png)

Figure 8. Maximum y-velocity (v_{max}/u_0) at half-section of the cross-cut region

To illustrate the previous discussion more clearly, the velocity profiles for the highest Reynolds number (Re=3.2×10^6) are plotted at a location at the end of the brattice (i.e., y/H=1.5, y/H=0.5 and y/H=0.25) with different brattice lengths (cases B, C and D) in Figure 9. As seen, a slightly higher value of maximum y-velocity near the right wall is evident for longer brattice case. This is also associated with a lower velocity near the left wall of the cross-cut (next to the brattice). The higher values of y-velocity near the right wall and lower values next to the left wall indicate less flow circulation in the cross-cut region and better ventilation for longer brattice cases. This, once again, gives more credit to the use of brattice in cross-cut regions where the near-wall velocity increases dramatically by the aid of the brattice.

![Figure 9](image9.png)

Figure 9. Velocity profile (v/u_0) along a plane at the end of brattice (Re=3.2×10^6)

Another point that is worthwhile to mention is with the longest brattice, ignoring the near-wall regions, the velocity distribution is more flattened in the shown cross-section, being closer to zero. This is also an indication of less circulation in the center of the cross-cut region and better ventilation for the longest brattice case.

![Figure 10](image10.png)

Figure 10. Brattice mass flow rate ratio at different brattice lengths (Re=3.2×10^6)

Another feature of considerable interest is that with the smallest brattice length, the velocity of airflow in a narrow region formed by the left wall and the brattice levels up compared to the other counterparts. This can be explained by higher mass flow rates through this region for shorter brattice cases due to lower pressure losses. This has also been illustrated in Figure 10, where the normalized mass flow rate of air through the brattice region is plotted against brattice lengths at Re= 3.2×10^6. Thus, it can be said that even though the mass flow rate of air through the brattice region is minimum for the longest brattice length, it provides a better airflow through the cross-cut region.
Conclusions

The flow field within a segment of an underground mine, where a cross-cut is located, was investigated numerically. The study was carried out to compare the flow field within the cross-cut region with and without a brattice. The brattice was used to deflect the airflow into the cross-cut region. In addition, the effects of brattice length on the flow pattern within the cross-cut region were examined at different main airstream velocities. The results showed a slight difference between the presented flow patterns. This was an indication of minor dependency of the dimensionless streamlines to the magnitude of main airstream velocity. With no brattice, the streamlines proved weak compared to that when a brattice was used. The use of a brattice increases the airflow into the cross-cut region, depending on the distance of brattice with the wall. The y-velocity in the cross-cut region behaved differently for various brattice lengths. Higher brattice lengths showed a higher maximum velocity near the right wall of the cross-cut and therefore, a better ventilation mechanism. This study proved to be very useful in predicting the flow behavior in a section of the mine, where a brattice was used to deflect the airflow into the cross-cut. It is recommended to develop the present model to study other configurations of the brattice and different locations of the brattice in the mine workings.

6 Acknowledgement

This work was supported by a start-up grant provided by The University of Queensland. The second author also thanks financial supports of The University of Queensland in terms of UQILAS, Endeavor IPRS, and School of Engineering Scholarship.

7 References

Brunner, D. J., Miclea, P. C., McKinney, D. and Mathur, S., 1995. Examples of the application of computational fluid dynamics simulation to mine and tunnel ventilation. in, Lexington, KY, USA, pp 479-484.
Kissell, F. N. and Matta, J. E., 1979. Face ventilation system for coal mines.
Potts, J. D. and Jankowski, R. A., 1988. Improved ventilation and water sprays reduce auger mining dust levels. in, Berkeley, CA, USA, pp 444-453.