Experimental and numerical analysis of air flow in a dead-end channel

M Branny, M Jaszczur, W Wodziak and J Szmyd
AGH University of Science and Technology, Krakow, 30 Mickiewicza Ave. Poland
E-mail: branny@agh.edu.pl

Abstract. This study summarises the results of experimental testing and numerical simulations of airflow in a laboratory model of a blind channel aired by a forced ventilation system. The aim of the investigation is qualitative and quantitative verification of computer modelling data. The components of the velocity vector are measured using Particle Image Velocimetry. Two turbulence models, the standard k-ε model and the Reynolds Stress Model, were used in a numerical calculation. A comparison of the magnitude of the velocity vector and the kinetic energy of turbulence obtained by the experimental methods and numerical calculations shows that in qualitative terms, the predicted velocity field correlates well with the measurement data. The mean relative error between the results of the calculations and measurements for the magnitude of the velocity vector and the kinetic energy of turbulence is about 29% for the Reynolds Stress Model and 33% for the standard k-ε model.

1. Introduction
Mining excavations with the working faces of blind headings are typically ventilated with fan and duct systems. In Polish coal mines the forcing, fan and duct system is frequently used to ventilate blind drifts and headings. The distance from the jet inlet to the blind wall of a heading does not exceed 10m, 8m in methane mines. The velocity field generated by the air supply stream is fully three-dimensional and the flow is turbulent. Currently, the parameters of 3D air flows are determined using the CFD (Computational Fluid Dynamics) approach. In engineering applications RANS models (Reynolds Average Navier-Stokes) are most commonly used.

This paper presents the results of experimental testing and numerical predictions of airflow in a laboratory model of a blind channel aired by a forced ventilation system. The geometrical scale of the physical model is 1:10. The model walls are smooth, the channel cross-section is rectangular and the presence of the mining machines and equipment in the channel is disregarded. The measurements were performed for steady and isothermal flow conditions. In this case, an equal Reynold’s number in the model and the real object assures that the flow criteria are similar. The components of the velocity vector were measured using the Particle Image Velocimetry (PIV) technique. The numerical procedures presented in this paper use two turbulence models: the standard k-ε model and the Reynolds Stress Model (RSM). Numerical calculations with boundary conditions similar to those in the experimental studies are supported by the Ansys Fluent program. The main objective of this research is the qualitative and quantitative verification of computer modelling results.

Studies on the validation of turbulence models used in mine ventilation were done in [1], [2]. The authors compared the results of the measurements of the velocity field [1] and the concentration of methane [2] with numerical prediction for airflow in face headings. They estimated that the results
obtained with the Spalart-Allmaras and SST $k$-$\omega$ models agree with the experimental data. The experimental validation of the turbulence models for the airflow in the intersection of the mining face with the ventilation gallery was presented in [3]. The airflow investigated in this paper differs from those analyzed in [1], [2], [3] where the connections between the ventilation ducts have been arranged differently. Validation of turbulence models used in computer simulations of airflows in blind workings was done in works [4], [5], [6] and [7]. The authors [4] compared the results of numerical simulations with the results of measurements of longitudinal components of velocity obtained in a real object. In the numerical calculations they used the Spalart-Allmaras model. An error in the predicted values of the longitudinal components of velocity was estimated at about twenty percent. The authors [5], [6] and [7] used the results of the measurements obtained by Parra [4] to validate four models of turbulence namely the models of Spalart-Allmaras, $k$-$\varepsilon$, $k$-$\omega$ and RSM. Considering the computation time and the relatively small relative error of up to 15%, the authors [5] found that the velocity field obtained using the Spalart-Allmaras model was determined with reasonable accuracy for practical purposes. On the other hand, according to authors [6] and [7] the error in the predicted values of the longitudinal components of the velocity was 36-72% for the Spalart-Allmaras model, 15-51% for $k$-$\varepsilon$, 37-93% for $k$-$\omega$, and 5-70% for RSM. The results of the validation of two turbulence models ($k$-$\varepsilon$ and RSM) in the case of the airflow through a long blind chamber was presented in [8]. The subject of this paper is a continuation of the research described in [9]. Unlike the works presented above, experimental verification of the selected turbulence models in this paper was performed based on the measurements of three (mean and fluctuating) components of the velocity vector and for three different Reynolds numbers.

2. The experimental setup

Experimental measurements were conducted in a laboratory setting consisting of a rectangular channel 3.35m in length and a cross-section area of 0.4m $\times$ 0.2m. Figure 1 shows the setup for the measurement. The air supply system to the laboratory stand was the same as in [9]. The distance between the axis of the air supply pipe from the top and side walls of the channel was 0.08m and 0.75m from the jet inlet to the blind wall. The experiment was conducted to obtain the velocity field in the measurement section of the test facility, using the PIV technique. In real flow the Reynolds number (based on the hydraulic diameter of the supply duct as a characteristic dimension and mean velocity) usually exceeds 100 000. Measurements were performed for the airflow velocities in the inlet duct of 15.4m/s, 21.2m/s and 35.4 m/s, which gave Reynolds numbers of about 78 000, 108 000 and 180 000, respectively.

![Figure 1. Schematic diagram of the test section.](image-url)
3. Comparison of experimental and numerical results

Figure 2 shows the velocity field contours obtained by measurements and numerical simulations in the horizontal cross-section located on the level of the air-supply duct and 5cm above the channel’s bottom. The predicted velocity field with the use of both turbulence models qualitatively agrees well with the experimental data.

![Figure 2](image)

**Figure 2.** Contours of the longitudinal component of velocity \( V_x \) in the horizontal plane on the level above-in the axis of the supply duct, below – 0.05m from the channel’s bottom.

The experimental and numerical data are compared quantitatively in figure 3. This figure shows the distributions of the magnitude of the velocity vector along the selected lines of the channel cross-section. Horizontal lines are defined by coordinates \( x \) [-0.2m,0.2m], \( z=0.02m \) (the axis of the supply duct) and \( z=-0.05m \) (5 cm from the bottom of the channel) and located at distances of \( y=0.25m \) and 0.65m from the inlet. The mean relative error between the measured and calculated magnitudes of the velocity was 29% for the RSM model and 33% for the \( k-\epsilon \) model. The most significant discrepancies between the measurement and calculation data are obtained in the bottom section of the channel \( (z=-0.05m) \). In this zone, locally, the maximal relative error is about 100%.
Figure 3. A comparison of the magnitude of the velocity vector $|\mathbf{V}|$ along selected lines on the level (a) the axis of the supply duct; (b) 0.05m from the bottom of the channel.

Figure 4. A comparison of the kinetic energy of turbulence $k$ along selected lines on the level (a) the axis of the supply duct; (b) 0.05m from the bottom of the channel.
One of the parameters characterizing the turbulent flow is the kinetic energy of turbulence. Figure 4 shows a comparison of the measured and calculated values of the kinetic energy of turbulence. There is a visible qualitative agreement between the numerical and experimental data. Only in the area of 0.25m away from the inlet and at ¼ height of the channel, are the calculated values of the kinetic energy of turbulence significantly underestimated. The mean relative error between the measured and calculated values for the kinetic energy of turbulence is 27% for the RSM model and 31% for the standard \( k-\varepsilon \) model. The comparison of numerical results using the turbulence models: \textit{Raelizable} \( k-\varepsilon \) and \textit{SST} \( k-\omega \) with experimental data has been presented in [10]. The mean relative error between the measured and calculated magnitudes of velocity was close 30% and the mean relative error between the measured and calculated kinetic energy of turbulence was 42% [10].

4. Recirculating flow
Figure 5 shows the velocity field on the horizontal plane 0.05m above the channel’s bottom obtained from the experiment and calculation. The presented velocity field pattern covers a zone with recirculating flows. Due to the measurement technique employed, the area under the air supply duct was invisible to the CCD camera, hence no measurement results are available for this area \( x \in [0.08m, 0.2m] \). It appears that the predicted range of a zone with recirculating flows in the direction of the outlet is overestimated. The measured length of the flow recirculation zone in the outlet direction is nearly 0.2m, and the predicted values are: 0.6m for the RSM and 2.2m for the \( k-\varepsilon \) model. Fig. 6 shows a comparison of the measured and predicted longitudinal components of the velocity vector along the lines defined by the coordinates \( x \in [-0.2m, 0.08m] \), \( y = -0.1m \) and \( y = -0.35m \) and for \( z = 0.02m \) (the axis of the air supply duct) and for \( z = -0.05m \) (1/4 of the channel height). In this area of flow, the calculated values of the streamwise components of the velocity vector are overestimated. This means that the predicted volumetric airflow rate in this zone tends to be overestimated, too. In this section the results obtained by the RSM model better approximate the measurement data.

![Figure 5](image-url)
Figure 6. The distribution of the longitudinal component of the velocity along horizontal lines, on the level left - the axis of the supply duct, right - 0.05m from the bottom of the channel.

5. Similarity of velocity fields
Measurements were performed for three mean velocities of the inlet air stream: 15.4m/s, 21.2m/s and 35.4m/s. The Reynolds numbers corresponding to these velocities are 78 000, 108 000 and 180 000, respectively. Figure 7 shows the distributions of the normalized magnitudes of the velocity vector for the selected measuring lines. The magnitude of the velocity vector \( \vec{V} \) was normalized by the mean air velocity \( V_0 \) at the inlet. The mean relative error between the normalized values of the velocity is 10.1% for the flows at smaller Reynolds numbers (78000 and 108000) and 3.8% at Reynolds numbers of 108000 and 180000. The distribution of normalized kinetic energy is shown in figure 8. The mean relative error between the normalized values of kinetic energy of turbulence for the flows at Reynolds numbers 78000 and 108000 is 13.4%, while for Re=108 000 and Re=180000 this error is 7.0%. The degree of curve-fitting for the normalized magnitudes of velocity and the kinetic energy of turbulence increases with an increase in the Reynolds number. The normalized fields of velocity as well as the kinetic energy of turbulence are qualitatively consistent within the range of Reynolds numbers 100 000÷200 000. Also, the results of the numerical simulations are consistent within this range of Reynolds numbers. In real conditions, ventilation of dead-end workings is carried out also at higher Reynolds numbers than those considered above. On the basis of this research it can be assumed that the error in the forecast values of the velocity vector will be close to the calculated ones also in the case of larger Reynolds numbers.
**Figure 7.** The normalized distribution of the magnitude of velocity along horizontal lines on the level: left - the axis of the supply duct, right - 0.05m from the bottom of the channel.

**Figure 8.** The normalized distribution of kinetic energy of turbulence $k$ along horizontal lines on the level: left - the axis of the supply duct, right - 0.05m from the bottom of channel.
6. Conclusions
When planning blind workings ventilation it is necessary to take into account a number of factors, which include methane and dust hazards above all. Currently, the parameters of air flow are often determined using the CFD approach. The validation of the turbulence model remains a necessary step in the data verification process. In mining conditions validation of the results of numerical simulations is extremely difficult due to the environmental conditions (high humidity and dust concentration of air) and measurement capabilities together with the applicable safety regulations. Alternatively, the measurements are made on physical models. The comparisons presented here show that results obtained using both turbulence models (k-ε and RSM) capture the real flow behavior with similar accuracy. In qualitative terms, the predicted velocity field correlates well with the measurement data. The mean relative error between the results of the calculations and measurements of the magnitude of the velocity vector and the kinetic energy of turbulence is about 29% for the RSM and 33% for the k-ε model. The calculated range of the recirculating zone in the direction of the outlet is overestimated. In this zone, the predicted values of the longitudinal components of the velocity vector are also overestimated. That means that the virtual amount of recirculating air in this zone is also overestimated. Both measured and calculated the normalized (by the mean velocity at the inlet) fields of velocity as well as the kinetic energy of turbulence are qualitatively consistent within the range of Reynolds numbers 100 000÷200 000. On this basis it can be assumed that the error in the forecast values of the velocity vector will be close to the evaluated ones also in the case of larger Reynolds numbers.

Acknowledgments
The present work was supported by the Polish Ministry of Science (Grant AGH 11.11.100.774 and Grant AGH No. 11.11.210.312).

7. References
[1] Wala A M, Stoltz J R and Jacob J D 2001 Proc. 7th Int. Mine Ventilation Cong. Krakow 411
[2] Wala A M, Vytla S, Taylor C D and Huang G 2007 Mine face ventilation: a comparison of CFD results against benchmark experiments for CFD code validation Mining Engineering 59 10
[3] Szmyd J, Branny M, Karch M, Wodziak W, Jaszczur M and Nowak R 2013 Experimental and numerical analysis of the air flow in T-shape channel flow Arch. of Mining Sciences 58 333
[4] Parra M T, Villafruela J M, Castro F and Mendez C 2006 Numerical and experimental analysis of different ventilation systems in deep mines Building and Environment 41 2 87
[5] Sasmito A, Birgesson E, Ly H C and Mujumdar S 2013 Some approaches to improve ventilation system in underground coal mines environment – A computational fluid dynamic study Tunnelling and Underground Space Technology 34 82
[6] Kurnia J, Sasmito A P and Mujumdar A S 2014 Dust dispersion and management in underground mining faces Int. J. Mining Science and Technology 24 39
[7] Kurnia J, Sasmito A and Mujumdar A Simulation of a novel intermittent ventilation system for underground Tunnelling and Underground Space Technology 42 206
[8] Branny M, Karch M, Wodziak W, Jaszczur M, Nowak R and Szmyd J 2014 An experimental validation of a turbulence model for air flow in a mining chamber J. Phys. Conf. Series 530 012029
[9] Branny M, Szmyd J, Jaszczur M, Nowak R, Filipek W and Wodziak W 2013 Model research of air flow in blind headings with fun and duct system of ventilation Gornictwo i Geologia 8 5 (in polish)
[10] Wodziak W 2015 Experimental and numerical study of air flow in a laboratory model of blind channel aired by a force ventilation system PhD Thesis AGH University of Science and Technology Krakow (in polish)