An application of SPIV technique to experimental validation of the turbulence model for the air flow in the intersection of the mining face with the ventilation gallery

Marek Jaszczur\textsuperscript{1}, Remigiusz Nowak\textsuperscript{1}, Janusz Szmyd\textsuperscript{1}, Marian Branny\textsuperscript{2}, Michał Karch\textsuperscript{2}, Waldemar Wodziak\textsuperscript{2}

\textsuperscript{1}Faculty of Energy and Fuels, AGH University of Science and Technology, Poland.  
\textsuperscript{2}Faculty of Mining and Geoengineering, AGH University of Science and Technology, Poland.  
E-mail: marek.jaszczur@agh.edu.pl

Abstract. This paper presents a comparison between results obtained using SPIV experimental technique and numerical simulations approach. An analysis has been performed to validate the turbulent models used in mining ventilation systems. The flow of air across the intersection of the mining face with the ventilation gallery has been examined.

1. Introduction

Computational fluid dynamics (CFD) methods have useful applications in many fields of technology. In mining engineering, they are used to simulate flow issues in solving local ventilation problems. The main constraint limiting the use of solutions obtained by means of CFD models is that they must be experimentally verified. This paper presents the results of experimental and numerical investigations of air flow through the crossing of the mining longwall heading and ventilating gallery. The investigated object consists of airways (headings) arranged in a T-shape.

Maintained for technological reasons, the cave is exposed particularly to dangerous accumulations of methane. A methane-air mixture flows into the cave of 3-5 m length from a goaf space. This flow is not limited, because it should push away the zone of high methane concentrations from the conveyor drive located at the terminal section of the wall. The existing system of duct connections causes this to be the least intensively ventilated part of the longwall. Properly arranged ventilation should assure the maintenance of methane concentrations at a safe level. In mine ventilation, in theoretical investigations of the velocity field in chosen segments of a ventilation network, viscosity turbulence models are used exclusively, both in Poland and abroad, with the standard $k-\varepsilon$ model applied in the majority of cases. Numerous papers have been published on periodic, straight or L-shaped turbulent channel using a large variety of modeling techniques: RANS, URANS, LES, DLES and DNS (Jaszczur & Portela, 2008), (Silvester, 2002), (Wala \textit{et al.}, 2008). Few, however, have taken up the relatively simple but technologically crucial geometry. In practice, for such high Reynolds numbers, viscous RANS
models are used exclusively. Experimental and numerical studies done in the US on mine face ventilation are worthy of mention here. Depending on the considered variant of the exploitation system, various researchers have indicated various models for the best way to reproduce the real flow. The models include the $k-\varepsilon$ model in the RNG version, $k-\omega$ model in the SST version and the single equation Spalart-Allmaras' model (Wala et al., 2001), (Wala et al., 2007), (Aminossadati & Hooman, 2008). The velocity field we have investigated differs significantly from those analysed in the above-mentioned papers as it arranges the connections between the ventilation ducts differently.

2. Experimental setup

The experimental set-up shown in Fig. 1 represents a model of the intersection of the mining face and ventilation gallery. Here assumed that dimensions of the real object are: the cross-section of the duct 4m x 2m, the length of the cavity 5m, the length of inlet section (terminal segment of a mine face) 3.3 m and outlet 5.8m. The mean velocities are usually in the range of 1-2m/s (Reynolds number from 150000 to 300000). The geometrical scale of the physical model was 1:10. It was assumed that the air flow is steady and isothermal. In this case, equality of the Reynolds number in the model and the real object assures the flow criteria are similar.

![Figure 1. Experimental set-up of T-shape channel flow.](image)

The air was used as the experimental fluid in the channel, thus the equality of the Pr number was automatic. Having equal Re numbers ensures the averaged velocity fields are similar. The inlet velocities were $U_1=3.8m/s$ and $U_2=9.85m/s$ and the corresponding Reynolds numbers were equal to $Re_1=57300$ and $Re_2=148600$.

Stereo Particle Image Velocimetry (SIV) method was used to evaluate the velocity vector components. The particles were illuminated with a double-pulse Nd:YAG laser of energy of about 400 mJ per pulse. The digital images were acquired by 4 Mpx monochromatic CCD camera. In each experiment, 1000 double frame images were recorded with the camera recording at a frequency of 3 Hz, which resulted in an overall time of one measure of around 5 minutes. An analysis of the number of images on average value has been show in Fig. 2. Time $\Delta t$ between two subsequent frames varied from about 100$\mu$s to 500$\mu$s. However, the measurements inside the cave were taken in the range of 3000-4000 $\mu$s, because the velocities there are lower than in the other sections. During the calculations, the size of interrogation windows that exhibit satisfying results was set to 32x32 px.
Figure 2. Wall-normal velocity component versus number of samples to average procedure at $Re=148600$

3. Mathematical model

Classical modelling of turbulence is based on the Reynolds concept, which for stationary and incompressible flows leads to the following equations:

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (1)$$

$$\frac{\partial}{\partial x_i} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) - \frac{\partial}{\partial x_j} (-\rho u_i u_j) \quad (2)$$

In this study, two models of turbulence were tested: the standard $k-\varepsilon$ model and a variation of that model, the RNG $k-\varepsilon$ model. The RNG model differs from the standard version because it accounts for the additional source term in the transport equation for the rate of kinetic energy dissipation, thus describing the effects of the rapid rate of strain and streamline curvatures, and also because it uses a different method to compute the effective viscosity and turbulent Prandtl number. Boundary conditions on rigid walls were set in the form of non-equilibrium wall functions. A numerical simulation of the air flow for conditions identical to experimental investigations was performed using FLUENT software. A structured, non-uniform mesh was generated for the computational domain. Local refinement was used in the cross-road region, where large gradients exist in the flow field and in the vicinity of the walls. Grid independence was examined with grid convergence index GCI. The order of magnitude of the GCI index is $10^{-2}$-$10^{-3}$ for the two tested meshes of about 876,000 cells and another of 2,950,000 cells.

4. Results

The measurements and calculations were performed for two flow velocities: 3.8m/s and 9.85m/s. We present here only profiles of stream-wise and wall-normal components of velocities along the horizontal line at the half height of the channel ($z^*=0$) located before and after the cross of the ducts and in the cave. All geometrical dimensions are normalised by channel height. Fig. 3 shows a comparison of the measured and calculated velocity just before the cross of the duct for $y^*=-0.3$. Small differences between the measurements and calculation results for the stream-wise and wall normal components for two considered flow velocities can be seen. In the cave (Fig. 4) the measured and numerically generated flow fields are qualitatively similar, but quantitatively
Figure 3. Flow streamwise (a) and (c) and wall-normal (b) and (d) velocity component for section A at \(z^*=0\) and \(Re=57300\) (top), 148600 (bottom). (—–)\(k-\varepsilon\), (- - -)RNG \(k-\varepsilon\), (∆)Exp.

they differ considerably. The narrow stream flows into the cave along the wall \(y^*=1.0\). Its width is not bigger than 0.5\(h\) in the range of \(z^*=(-0.5,0.5)\). The zone with the return stream with positive values of streamwise components of velocity is considerably larger. This flow pattern is common both for measurements and calculations.

Figure 4. Flow fields obtained in the cave by experimental measurement (left) and numerical calculation (right) at \(z^*=0\) and \(Re=57300\)

Fig. 5 shows the comparison between measured and calculated velocities along the horizontal line at half height and at the entrance section of the cave, \(x^*=-0.3\). The calculated values of streamwise velocity components over-predict the measured ones by about two times whereas the wall normal components differ even in the shape of its profile. Note that the flow field in this
Figure 5. Flow streamwise (a) and (c) and wall-normal (b) and (d) velocity component for section C at \(z^*=0\) and \(Re=57300\) (top), 148600 (bottom). (—–) \(k-\varepsilon\), (- - -) RNG \(k-\varepsilon\), (Δ) Exp.

Figure 6. Flow streamwise (a) and (c) and wall-normal (b) and (d) velocity component for section B at \(z^*=0\) and \(Re=57300\) (top), 148600 (bottom). (—–) \(k-\varepsilon\), (- - -) RNG \(k-\varepsilon\), (Δ) Exp.
zone is complex and even a slight translation of the horizontal line in the $x$ direction changes the profile of the transverse component considerably (Fig. 5). Fig. 6 shows the comparison between the measured and calculated components of velocity behind the cross of the ducts for $x^*=0.3$. The predicted velocities are in good agreement with the experimental results in the main stream but differ in the separating zone. The calculations over-predict the negative streamwise components of velocity in the zone where recirculation occurs.

5. Conclusion

The validation of the particular turbulence model remains a necessary step in the fully justified application. In this paper, a two-equation $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence model was tested with air flowing through the laboratory model representing a fragment of a mining ventilation network. The k-e model was selected mainly because it is a standard, commonly used model in solving local ventilation problems in mines. The examined flow is characterised by such flow features as separation, stream impingement on the wall, stress-driven flow and strong streamline curvature. For such cases, non-equilibrium wall function is recommended together with a turbulence model which includes the particular flow feature. For our investigation we chose the RNG $k-\varepsilon$ model. Based on this study, we can offer the following conclusions. Neither tested model, the standard $k-\varepsilon$ and RNG $k-\varepsilon$, provided satisfactory results for the examined flow in the cave zone. Furthermore, the calculated values of streamwise components are over-predicted in the cave zone. This will cause the ventilation intensity to be overestimated in this area. For the cross section located before and behind the cross of the ducts, the calculations and measurements are in quite good agreement with the experimental results, bearing in mind the accuracy needed in ventilation problems. However, considerable differences are observed in the separation zone.

Acknowledgment

The research was supported by the Polish Ministry of Science and Higher Education (project No. 4653/B/T02/2010/39)

References

WALA, A.M., STOLTZ, J.R. & JACOB, J. 2001 Numerical and experimental study of mine face ventilation system for CFD code validation. Proceedings of the 7th International Mine Ventilation Congress, Krakow, Poland

WALA, A.M., VYTILA, S., TAYLOR, C.D. & HUANG G. 2007 Mine face ventilation: a comparison of CFD results against benchmark experiments for CFD code validation. Mining Engineering 59 10

WALA, A.M., VYTILA, A., HUANG G. & TAYLOR C.D. 2008 Study on the effects of scrubber operation on the face ventilation. 12th U.S. North American Mine Ventilation Symposium 2008 Reno, USA

AMINOSADATI, S.M. & HOOMAN, K. 2008 Numerical simulation of ventilation air flow in underground mine workings.12th U.S North American Mine Ventilation Symposium, Reno, USA

SILVESTER, S.A. 2002 The integration of CFD and VR methods to assist auxiliary ventilation practice. PhD thesis, The University of Nottingham.

JASZCZUR, M. & PORTELA L. 2008 Numerical data for reliability of LES for non-isothermal multiphase turbulent channel flow. In Quality and reliability of large-eddy simulations (ed. J. Meyers, B. Geurts & P. Sagaut), pp. 343–354. Springer.