Experimental validation of the transport phenomena in T-shape channel flow

M Jaszczur¹, R Nowak¹, J S Szmyd¹, M Branny² and W Wodziak²

¹Dept. of Fundamental Research in Energy Engineering, Faculty of Energy and Fuels, AGH-University of Science and Technology, Krakow, Poland
²Faculty of Mining and Geoengineering, AGH-University of Science and Technology, Krakow, Poland

E-mail: janusz@agh.edu.pl

Abstract. This paper presents the results of experimental and numerical investigations of air flow through the crossing of a mining longwall and ventilation gallery. The object investigated consists of airways (headings) arranged in a T-shape. Maintained for technological purposes, the cave is exposed particularly to dangerous accumulations of methane. The existing system of duct connections causes part of the crossing to be insufficiently ventilated. Properly arranged ventilation should assure methane concentrations are maintained at a safe level. The geometrical scale of the physical model was 1:10. It was assumed that the air flow is steady and isothermal. Stereo Particle Image Velocimetry (SPIV) was used to evaluate the velocity vector components. 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 examined flow is characterised by such flow features as separation, stream impingement on the wall, stress-driven flow and strong streamline curvature. Velocity fields and scalar (concentration or temperature) fields obtained in the cave by experimental measurements and numerical calculations for different Reynolds numbers are presented and discussed in this paper.

1. Introduction
Flow and mixing at a T-shape channel is encountered in many industrial applications such as mining ventilation systems, chemical reaction processes, combustion processes, and piping systems of the plant. The piping systems of nuclear power plants have many T-junctions where high and low temperature fluids meet at several types of flow conjunctions with various velocity ratios. The temporal and spatial temperature fluctuations that occur due to the mixing of hot and cold water induce high-cycle thermal fatigue of the structure around the T-junctions [1]. The thermo-fluid behaviors of cross-flow type T-junctions have been experimentally/numerically investigated [1-3]. Numerous papers have been published on a periodic, straight or L-shaped turbulent channel using a large variety of modeling techniques: RANS, URANS, LES, DLES and DNS [4 – 6]. However, there are a number of detailed experimental studies related to the single phase air flow and gas-particulate two-phase turbulent flow L-shaped channels [7], [8]. The application of interest in the present paper concerns air flow through the crossing of the mining longwall and ventilation 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 ventilated part of the longwall. Properly arranged ventilation should assure the maintenance of methane concentrations at a safe level.

2. Experimental setup and procedures

The experimental set-up shown in figure 1 represents a model of the intersection of the mining face and ventilation gallery. The dimensions of the real object are assumed to be: 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 the outlet 5.8m. The mean velocities are usually in the range of 1-2m/s (Reynolds number from 150,000 to 300,000). The geometrical scale of the physical model was 1:10. It was assumed that the air flow is steady and isothermal. In this case, the equality of the Reynolds number in the model and the real object ensures that the flow criteria are similar.

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 that the averaged velocity fields are similar. The inlet velocities were $U_1=3.8$m/s and $U_2=9.85$m/s and the corresponding Reynolds numbers were equal to $Re_1=57,300$ and $Re_2=148,600$. Stereo Particle Image Velocimetry (SPIV) was used to evaluate the velocity vector components and the particles were illuminated with a double-pulse Nd:YAG laser of energy of about 400 mJ per pulse. The digital images were acquired using a 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 per average value is shown in figure 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 the interrogation windows that exhibit satisfying results was set at 32x32 px.

![Figure 1. Experimental set-up of T-shape channel flow](image-url)
3. Mathematical model and numerical procedures

Classical modeling of turbulence is based on the Reynolds concept, which for incompressible and Newtonian fluids yields the following equations [11]

\[
\frac{\partial U_i}{\partial x_i} = 0
\]

\[
\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial U_i}{\partial x_j} - u_{ij} \phi \right)
\]

\[
\frac{\partial C}{\partial t} + U_i \frac{\partial C}{\partial x_i} = \frac{\partial}{\partial x_i} \left( D \frac{\partial C}{\partial x_i} - u_{ij} \phi \right)
\]

\[
-u_{ij} = \nu_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}
\]

where \( \nu_i = C_\mu \frac{k^2}{\varepsilon} \) - turbulent viscosity, \( k \) - kinetic energy, \( \varepsilon \) - dissipation rate of \( k \), \( C_\mu \) - constant, \( \delta_{ij} \) - Kronecker delta.
where $C_1, C_2, \sigma_k, \sigma_\varepsilon$ are constants [11].

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 [9]. 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. The $k-\varepsilon$ RNG model is derived using a statistical technique called the renormalization group theory [10]. Boundary conditions on rigid walls were set in the form of non-equilibrium wall functions. Numerical simulations of the air flow for conditions identical to the experimental investigations were 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 $10^{-2}$ to $10^{-3}$ for the two tested meshes - the first of roughly 876,000 cells, and the other of 2,950,000 cells.

4. Experimental and numerical results

The measurements and calculations were performed for three flow velocities: 3.8m/s, 7.0m/s and 9.85m/s (Reynolds numbers Re=57,300, 107,000, and 148,600). 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 normalized by channel height. Figures 3 and figure 4 show a comparison of the measured and calculated velocity for the cross section A, C and B and for Reynolds number Re=57,300 (Fig.3) and Re=148,600 (Fig.4). Small differences between the measurements and calculated results for the streamwise and wall normal components for two considered flow velocities can be seen for entrance section A and large difference for wall normal velocity components for sections B and C. Also, the numerical predictions themselves differ considerably depending on the model in that area. 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 their profile. Note that the flow field in this zone is complex and even a slight unsteady of the horizontal line in the $x$ direction changes the profile of the transverse component considerably. For section B 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. For the measurements of the ventilation phenomena in the cavity, an experimental test has been performed. At the back wall of the cavity ($x = -0.7$), a droplets generator (PIV seeding atomiser and oil droplets) has been mounted behind the wall and connect with channel with many small holes. This was done to simulate methane injection (distribution) from the wall. After a few injections, the time scale particle concentration in the cavity reaches a statistically steady state. Figure 5 shows an instantaneous concentration field in cavity for the three Reynolds numbers used here (Re=57,300, 107,000, and 148,600). Those locations with very low and very high particle concentration can be seen in Figure 5, and can be easily
distinguished at a low Reynolds number. For a higher Reynolds number, dispersed phase is more spread over the entire cavity but it doesn’t provide better cavity ventilation. High particle concentration can be observed in a relatively large cavity area. As seen in figure 6, as the Reynolds number increases, so too does the fluctuation of concentration. On the border between the high and low concentrations, the fluctuation of concentration reaches its highest value. When the concentration of particles in the cavity enters a statistically steady state, the particle generator is switched off. This allows an analysis of ventilation in the cavity (how fast particle concentration is decreasing) to be made. The mean particle concentration in the cavity at $z^*=0$ and for all Reynolds numbers ($Re=57,300, 107,000, 148,600$) is shown in figure 7.

\[ \text{Figure 3. Flow streamwise (left) and wall-normal (right) velocity component for sections A, C and B at } z^*=0 \text{ and } Re=57,300 \text{ (---) } k-\varepsilon, (- - -) \text{RNG } k-\varepsilon, (\Delta) \text{Exp.} \]
This is shown for dimensional time $t[s]$ and non-dimensional time $t^* = t/t_f$ where $t_f$ represents a flow time scale equal to $H/\bar{U}$. The comparison of numerical results with experimental data is shown in figure 7-right. The time needed to dilute the initial concentration to a certain level resulting from calculations is about 30% shorter than that determined by measurements (see Fig. 7–right). It can be seen that calculations artificially rise with the intensity of ventilation in the zone of the cavity.
Figure 5. Instantaneous concentration profile in the cave (for the statistically steady state) at $z=0$ and for Reynolds numbers $Re=57,300$, $107,000$, $148,600$ (left, middle, right).

Figure 6. Instantaneous rms of concentration profile $C$ in the cave (for the statistically steady state) at $z=0$ and for Reynolds numbers $Re=57,300$, $107,000$, $148,600$ (left, middle, right).

Figure 7. -Left: Mean concentration in cave due to ventilation process at $z^*=0$ and for Reynolds numbers, $Re=57,300$, $107,000$, $148,600$ (horizontal axis is presented as $t^*$- non-dimensional time and $t$ real time) –Right: the comparison of numerical results with experimental data.
5. Error analysis
According to Huang et al. [12], the errors that may occur during the PIV measurements include error caused by outliers and root-mean-square error (RMS). Outliers (spurious vectors) are the velocity vectors that are easy to detect thanks to their corresponding velocity value, which is mostly several times larger than the velocity calculated for properly distinguished particles. Spurious vectors are randomly distributed and always appear in the calculation domain, though practically, in the properly prepared measurement, there are no more than 5% of these vectors [13] which, additionally, may be completely removed after a post-processing stage. In the DPIV technique, there is a large group of parameters that may influence the RMS error value, including seeding density, velocity gradients, out-of-plane particle motions, non-uniform illuminations and data acquisition noise [14]. One way to minimize the RMS error during the calculation process is to properly specify the interrogation window size and overlap parameter. The influence of both those parameters on the RMS value are presented in figure 8.

![Figure 8](image)

**Figure 8.** The influence of (a) the interrogation window size, (b) the various overlap factors on the accuracy of the velocity field calculations in the DPIV technique

6. Conclusions
The validation of the particular turbulence model remains a necessary step in the fully justified application. In this paper, two-equation $k - \varepsilon$ and RNG $k - \varepsilon$ turbulence models were tested with air flowing through the laboratory model representing a fragment of a mining ventilation network. The $k - \varepsilon$ 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 $k - \varepsilon$ nor the 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 T-junction, 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.

Acknowledgments
The research was supported by the Project No. 4653/B/T02/2010/39.
References

[1] Nakayama H, Hirotta M, Shinoda K and Koide S 2005 Thermal Science and Engineering 13 17
[2] Kelso R M, Lim T T and Perry A E 1996 Journal of Fluid Mechanics, 306 111
[3] Haven B A and Kurosaka M 1997 Journal of Fluid Mechanics, 352, 27
[4] Jaszczur M and Portela L 2008 in Quality and reliability of large-eddy simulations ed. J Meyers, B Geurts and P Sagaut, pp. 343 (Berlin : Springer)
[5] Wala A M Vytla S Taylor C D and Huang G 2007 Mining Engineering 59 10
[6] Kuan B Yang W and Schwarz M P 2007 Chemical Engineering Science, 62 2068
[7] Humphery J A C Whitelaw J H and Yee G 1981 J. Fluid Mechanics, 103 443
[8] Taylor A M K P Whitelaw J H and Yianneskis M 1982 J. Fluids Engineering, 104 350
[9] Yakhot V and Orszag S A 1986 J. Scientific Computing, 1, 1
[10] Choudhury D 1993 Introduction to the renormalization group method and turbulence modelling, Fluent Inc. Technical Memorandum, TM-107
[11] Rodi W 1979 Turbulence models and their application in hydraulics p. 27 (Delft: IAHR)
[12] Huang H, Dabiri D and Gharib M 1997 Meas. Sci. Technol. 8 1427.
[13] Westerweel J 1994 Exp. Fluids 24 236.
[14] McKenna S P and McGillis 2002 Exp. Fluids 32 106.