Tunnelling and Underground Space Technology 41 (2014) 88–94

Contents lists available at ScienceDirect

Tunnelling and Underground Space Technology

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

Shock losses characterization of ventilation circuits for block caving production levels

Juan P. Hurtado [a], Nicolás Díaz [a], Enrique I. Acuña [b], Joaquín Fernández [c,*]

[a] Universidad de Santiago de Chile, Santiago, Chile
[b] CODELCO, Proyecto Nuevo Nivel Mina, Santiago, Chile
[c] Universidad de Oviedo, Asturias, Spain

ARTICLE INFO

Article history:
Received 29 December 2012
Received in revised form 23 October 2013
Accepted 26 November 2013
Available online 25 December 2013

Keywords:
Block caving
Shock losses
Mine ventilation
Computational Fluid Dynamics
Efficiency improvement

ABSTRACT

With the tendency of world ore deposits to be deeper and of lower grade, block caving is becoming an increasingly important mining method. As with any underground method, an adequate mine ventilation system is essential to control operational costs while achieving the development and production schedule. This paper presents results obtained from a simulation based on Computational Fluid Dynamics (CFD), in combination with a mine ventilation network solver, to determine the head losses for the ventilation circuits of the production level drifts within the block caving layout at the El Teniente mine. In order to simulate the ventilation network, the airflow path is characterized in terms of shock losses from the intake ventilation sublevel through the drifts to the exhaust ventilation sublevel. Subsequently, these simulated head losses are used for the ventilation model of the drift and solved using a commercial ventilation program, to assess the energy loss. Simple energy saving modifications can then be proposed, generating 25% savings. This methodology assists in the design of production levels having efficient mine ventilation systems.

1. Introduction

Block caving is a highly cost-effective underground mass mining method. With the trend towards progressively deeper ore deposits and gradually decreasing ore grades, saving energy to improve both profitability and environmental sustainability will be essential in future projects.

Mechanized block caving involves activities on many levels which are necessary to production. The production level drifts are the main ventilation concern, with pollution by gases and dust due to load, haul and dump activities (Hurtado et al., 2010). For this reason, in many mines ventilation sublevels are developed exclusively to supply fresh air and to exhaust polluted air to surface. However, the circuit the air must follow is very tortuous, with singularities which generate significant shock losses.

Traditionally, shock losses are proportional to velocity pressure, P_v, ($P_s = X\,P_v$), where the constant of proportionality, X, is the shock loss factor and is influenced by the geometry of the shock loss sources. Usually, the shock loss factor characterizes each singularity, but does not vary with geometrical scale changes. Many authors have studied singularities experimentally to obtain the shock loss values presented in extended works (Miller, 1984; Idelchik, 1989), but such studies for mine ventilation are few (McElroy, 1935; Hartman, 1960; Jade and Sastry, 2008), with shock loss values usually taken from piping and HVAC tables.

Most of the singularities in the complex geometries that make up a block caving ventilation circuit are not presented in these tables and often shock loss factors from the most similar geometries are used, resulting in inaccurate values.

In this study, shock loss factors of the main singularities of the ventilation circuit of production levels in a mechanized block caving mine are characterized. The geometries studied are based on the El Teniente mine layout.

The methodology consists of the numerical study of a scale CFD model to determine initial and boundary conditions, turbulence model and mesh size, to allow setting up a real size CFD model. The scaled geometries are not necessarily identical to the real geometries, but should generate representative fluid dynamic phenomena. In this way, real geometry is rectangular with the roof in arch shape and the scale models were made of circular shape in PVC piping, taking in account the geometrical scale relationship from Gerhart and Gross (1985) of hydraulic diameter (D_h):

$$D_h = \frac{4A}{P} \tag{1}$$

where A is the total area and P is the wet perimeter. Thus, the simulation scale dimensions were maintained to the real geometries to conserve the main flow phenomena. The results of this

* Corresponding author. Tel.: +34 985 458 145.
E-mail address: jffrancos@uniovi.es (J. Fernández).

0886-7798/$ - see front matter
http://dx.doi.org/10.1016/j.tust.2013.11.010

characterization have been applied in an elemental block caving ventilation circuit to supply fresh air and remove polluted air in the production level drifts, with significant energy savings.

Previous work has studied the fluid dynamic phenomena in the Elbow-Split/T-joint geometry (Hurtado et al., 2010) by means of CFD techniques. In the present study the CFD results are validated with experimental data and more accurate results.

2. Experimental stage

The scale experiment considers three main singularity geometries, namely Elbow-Split/T-joint, Fan-Chamber-Raise and crosscut, which are shown in Fig. 1 (Díaz, 2011), where arrows indicate the path of the air from the intake to the exhaust airway. The characteristics of the experimental stage for the mentioned geometries are as follows (Maya, 2011):

(a) Elbow-Split/T-joint geometry (scale 1:17). This geometry corresponds to a chamber that connects a raise with a production drift, composed of one elbow and one split/T-joint (depending on the direction of flow), a chamber 2.6 m high, 3.0 m wide and 3.0 m deep, a 1.5 m diameter raise and a production drift section 4.0×4.0 m. For the intake mode (Elbow-Split) the fresh air enters toward the drift from the raise and for the exhaust mode (T-joint-elbow) the polluted air exits from the drift to the raise. Fig. 2 shows the experimental set up, where the arrow in the figure indicates the exhaust case; in the intake case the arrow would be reversed. The main branch is labeled 3 and the identical legs are labeled 1 and 2. The measurement procedure was based on establishing a flow rate relation Q_1/Q_3 to obtain the static and dynamic pressures needed to calculate the shock losses. The pressure loss factor X_{ij} can be calculated from (Miller, 1984):

$$X_{ij} = \frac{\text{(total pressure in leg i—total pressure in leg j)}}{\text{Mean velocity pressure in 3}} \qquad (2)$$

The measurements were made with a Pitot tube, following the standard procedure (AMCA-ASHRAE, 1985) and two orifice plates, according to ISO 5167-1 (2003) and ISO 5167-2 (2003). A double exhausting centrifugal fan at 1300 rpm, 270 mm wide with a 275 mm diameter impeller was used in the experiments.

(b) Fan-Chamber-Raise geometry (Scale 1:16). This geometry corresponds to a chamber located in the ventilation level, and connects the fan with the raise (see Fig. 1). The chamber has approximately the following dimensions: 6.0 m high, 4.5 m wide and 5.0 m deep. The raise and the fan have 1.50 m and 1.20 m diameters. For the intake mode, the fresh air is drawn by the fan into the chamber, where it is conducted to the raise due to pressure elevation. For the exhaust mode, the polluted air is taken from the chamber and conducted into the raise, to be pushed to the exhaust ventilation drift. The measurements were made with a Pitot tube, following the standard procedure (AMCA-ASHRAE, 1985) and similar experimental conditions described above. Fig. 3 shows the experimental set up for intake mode, where 1 is the centrifugal fan which push the air, 2 is the flow straightener, 3 is the Pitot tube, 4 is the singularity, 5 is a Pitot tube and 6 is the nozzle. Note that 2 and 3 can be exchanged with 5 in the exhaust mode and the inlet of centrifugal fan is connected to the set up to pull the air.

(c) Crosscut geometry (Scale 1:46). This geometry represents a production drift with extraction points (draw bell drifts) at a 60° angle, according to the El Teniente layout. The number of crosscuts between an intake and an exhaust raise is four, and must be considered an interference factor which is analyzed in (Maya, 2011). Fig. 3 shows the experimental set up, where 1 is the centrifugal fan which push the air, 2 is the flow straightener, 3 is the Pitot tube, 4 is a Pitot tube, 5 is the nozzle and 6 is the closing cone. Note that 2 and 3 can be exchanged with 5 in the exhaust mode and the inlet of centrifugal fan is connected to the set up to pull the air (see Fig. 4).

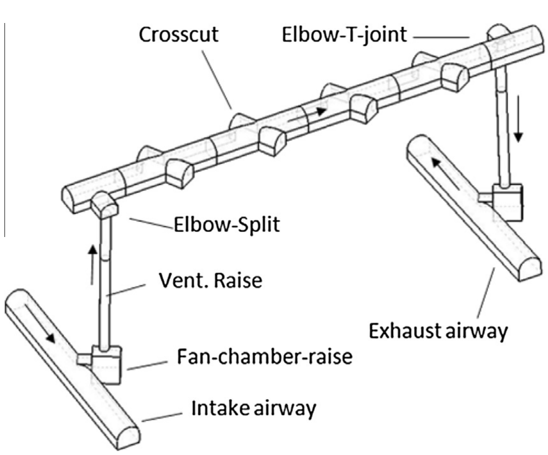

Fig. 1. General scheme of production drift ventilation system.

3. Numerical stage: simulations with CFD and results

Experimental scale models were built as described above. The experimental results of the scaled model allowed for estimations of the accuracy of the simulations results, which then provided the input parameters to simulate the real geometry (scale 1:1).

A three-dimensional simulation was made of the steady state flow with the commercial software package Ansys-Fluent®. This software uses the finite volume method to solve the Navier–Stokes equations and the turbulence model equations on an unstructured tetrahedral grid. The flow is described by the basic equations of the conservation of mass and conservation of momentum. The convergence criterion was a maximum residual of 10^{-5}. The initial conditions used for the three geometries were Velocity Inlet, to control the flow, and Outlet Vent, to adjust the operation point. A literature analysis and the authors' experience from previous work indicated that the viscous models k-epsilon RNG, Reynolds Stress Model (RSM), k-omega and k-epsilon Standard were the more appropriate to be analyzed. After various tests and grid prototypes, the final main characteristics for setting up the numerical scale model are described in Table 1, according to (Díaz, 2011).

All these obtained parameters were compared to the experimental results and were used to carry out the simulation of the real geometry. The aim of the CFD scale simulations was to obtain the main parameters to set up the simulation of the real geometry. The main results are separated by each of studied geometries.

4. Elbow-Split/T-joint geometry

Fig. 5 shows the shock loss factor vs. Reynolds number for the Elbow-Split geometry at the intake mode, which follows a good correlation of results between numerical and experimental scale models and the real numerical geometry. Fig. 6 shows velocity vectors for the numerical scale model (a) and the real numerical geometry (b), where flow follows a similar behavior with the same pattern of turbulence. At the same way, the behavior of flow rate