Building and Environment 121 (2017) 200–214

Contents lists available at ScienceDirect

Building and Environment

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

Correlating variability of the leakage characteristics with the hydraulic performance of an auxiliary ventilation system

Saad Akhtar, Mustafa Kumral, Agus P. Sasmito*

Department of Mining and Materials Engineering, McGill University, 3450 University Street, Montreal QC H3A0E8, Canada

A R T I C L E   I N F O

Article history:
Received 24 February 2017
Received in revised form
16 May 2017
Accepted 19 May 2017
Available online 20 May 2017

Keywords:
Auxiliary ventilation
CFD
Duct leakage
Fan operating point
Flow resistance

A B S T R A C T

Ventilation is one of the key factors in controlling underground working environment by providing sufficient amount of fresh air for breathing, dispersing harmful gasses and dust and to some extent for heating/cooling. Insufficient airflow is dangerous for the working face and can lead to fatalities. Duct leakage is the most common reason for the insufficient fresh air in underground working and has been the subject of many studies in the literature. However, the main focus has been on ascertaining its impact on the ventilation requirements of the underground environment. This study aims to identify key variables associated with duct leakage that significantly impacts the power consumption levels of auxiliary fans which form an integral part of the underground ventilation system. A three-dimensional Computational Fluid Dynamics (CFD) modeling approach is undertaken in conjunction with Monte Carlo simulations and multiple regression analysis to quantify the effect of duct leakage on the fan operating point and discharge flow rate towards the working face. Various cases involving the positioning, orientation, and size of the rupture in the ventilation duct are simulated, and their respective effects on fan operating point and power levels are determined. Results indicate that the operating point of a fan for ventilation ducts is strongly correlated with the position and size of the rupture, resulting in reduced delivery of ventilation air towards the working face for different levels of fan power consumption.

1. Introduction

Sufficient fresh airflow is the key to maintaining the good underground working environment as it is used for breathing, dispersing harmful gasses and dust, and thermal comfort. Insufficient fresh air can lead fatalities due to oxygen deficiency, explosion due to flammable gas (such as methane in a coal mine), health problems due to dust and diesel emission. Duct leakage has been the most common reason for the insufficient fresh air. Accurately incorporating the effect of duct leakage in the design process of the auxiliary ventilation system has been the Achilles heel for ventilation engineers [1–4]. Meticulous consideration of the impact of flow leakage from auxiliary ventilation duct is very important in ensuring the optimum levels of fresh air delivered while maintaining acceptable levels of contaminants in the underground environment [5–8]. One direct corollary of this consideration is a selection of appropriate fan for the job. Fan's operating point, which is defined by its operating pressure and corresponding flow rate delivered, is dependent upon the level of air leakage from ducts which in turn is influenced by different factors such as the type of duct material, rupture in ducts, duct joints etc. [9–11]. Hence in order to accomplish a safe and efficient ventilation system, it is pertinent to examine the mechanics of flow leakage in detail [12].

The quantification of air leakage and its effect on the overall energy consumption has been a subject of several experimental and numerical studies. Kalamees et al. [13] and Belleudy et al. [14] conducted experimental and numerical studies, exploring a related issue, where they quantified the average leakage rates from the structures. In another study by Jokisalo et al. [15] the effect of infiltration on the energy consumption of a building under cold environment is evaluated. The study found out that infiltration can cause a spike of up to 30% in the overall energy consumption by space heating equipment.

There are numerous studies in the literature which have attempted to incorporate the effect of leakage while analyzing the performance of auxiliary ventilation systems. Lee et al. [16] investigated the impact of leakage through airways to analyze ventilation efficiency of coal mines. They presented a mathematical formulation to determine the pressure drop and air flow rate while

---

* Corresponding author.
E-mail address: agus.sasmito@mcgill.ca (A.P. Sasmito).

http://dx.doi.org/10.1016/j.buildenv.2017.05.029
0360-1323/

incorporating air leakage in ducts. Wolski et al. [17] presented a numerical study to determine leakage factors in ducts in order to estimate system performance for different fan configurations. The study came up with an optimized system of series of fans in an auxiliary ventilation system. These early studies though valuable needed further model refinement in order to effectively incorporate the effect of duct leakage on overall ventilation efficiency.

Auld [9] studied the difference between auxiliary ventilation characteristics for leak-free and leaky ducts. The study introduced the concept of duct efficiency which takes into the account the variability in duct leakage with the increasing fan operating pressure. It uses the spreadsheet method to compute the fan operating points in a leaking duct. The study concluded that it would be faulty to select fan based on a leak-free system characteristic curve. Furthermore, it posited that the leakage condition of the duct become increasingly important as the duct length increases. In another study, Onder et al. [18] attempted to optimize the operational factors affecting the volumetric flow rate of air reaching the working face. Factors considered in the study include duct diameter, friction factor, air flow rate at the duct inlet, duct length and resistance coefficient of a leakage path. The study developed a computer program to solve for the fan operating point and air flow rate at working face while incorporating Kirchoff resistance analogy to model duct leakage. It was found that duct diameter and fan flow rate significantly affect the ventilation output. The volume flow rate to the working face was less sensitive to duct length, friction factor and resistance coefficient for duct leakage.

In another study, Onder et al. [19] presented statistical model of the volume flow rate reaching the working face using the field data from an underground mine. Multiple regression analyses were carried out to study the effect of variables such as duct diameter, length, friction factor, resistance coefficient of leakage path, fan operating pressure and flow output etc. on the volumetric flow rate at working face. Statistically, significant correlations were developed with acceptable accuracy to carry out optimization studies for ventilation design.

All of the afore-mentioned studies lay their prime focus on ascertaining the overall impact of leakage on the ventilation system characteristics. However, a component-level approach of tackling the leakage issue lacks in the literature. Variables such as rupture positioning, leak orientation, and rupture size are some of the key considerations that must be taken into account while designing such systems. These factors are of paramount importance, especially in the underground mining industry, as in some countries, the flow rate in working space is regulated by law, e.g. in typical Canadian mines in Ontario province requires 0.063 m^3/s of fresh air per kW power of diesel machine. Thus, if the airflow rate at the working face is below the minimum requirement due to leakage, the mine operation can be shut down by law. Also, it is not uncommon in underground operations to have tears and ruptures in the auxiliary ventilation ducts due to the movement of heavy machinery below the ducts. This leads to air leakage and inefficiency of the ventilation system. On top of that, insufficient airflow at the working face can lead to fatalities, especially in the underground coal mines that emit flammable gasses such as methane. This study aims to fill that gap by studying the effect of these parameters on fan power requirement and system volumetric flow rate at the working face through well-established computational fluid dynamics (CFD) approach coupled with Monte-Carlo and regression models. Furthermore, as pointed in the literature [20], the resistance to air flow in a ventilation system is critical to the overall performance of the system. Thus, quantification of the impact of leakage on system's efficiency is crucial. Note that the leakage phenomenon is complex in nature, highly non-linear and is affected by many factors that interacts to each other. Thus far, however, none has attempted to quantify the effect of these parameters from mechanistic and continuum point of view. Therefore, for first approximation, the scope of present study is limited to three leakage factors, i.e. rapture location, rapture size and rapture orientation. The effect of other parameters, such as duct material, duct joint, duct diameter, duct length and so forth will be evaluated in our future study.

The geometrical parameters associated with the rupture are assumed to be stochastic in nature. Consequently, their impact on the output variables is also studied probabilistically. Monte Carlo statistical simulations have been carried out in conjunction with regression analysis to determine the dependency of rupture characteristics on response variables. Based on the analysis of the distribution of the output variables, recommendations are drawn for estimating the likelihood of instances which are critical to the fan performance and ventilation requirements. Resistance analogy for the duct leakage is also developed stochastically to create an easy to follow the method in the context of ventilation professionals.

2. Methodology

This section provides a description of various steps involved in the computational fluid dynamics (CFD) analysis coupled with statistical methods. The flowchart showed in Fig. 1 elucidates the interaction between CFD calculations and statistical methods for this study. The rupture characteristics (size, location, and orientation) are taken to be random variables and are treated as such. Monte Carlo Simulation (MCS) is used to generate a case list of random samples for both independent and combined analysis of the impact of input parameters on output variables. These random samples are fed into the CFD simulation package which then yields the case specific values of output variables such as fan power, fan flow rate, leakage through rupture, flow rate towards the working face and operating pressure of the fan. These results are then analyzed by developing probability distributions for various response variables which give important insights on ventilation system design.

The preceding sub-section discusses the methodology for the CFD study which includes flow model and boundary conditions followed by a description of numerics. The last sub-section expounds the statistical method as tersely outlined afore.

2.1. Computational fluid dynamics (CFD) study

The schematic of the portion of the underground working face modeled along with the rupture in the auxiliary duct is shown in Fig. 2. The schematic shown in the figure comprises of the main drift with a branching tunnel towards working face. It also contains an auxiliary ventilation system as highlighted by the smaller curved duct inside the outer drift structure. The auxiliary duct is taken to be 68.58 m (225 ft) long with a 90° bend. The end of the duct is kept at 15.24 m from the working face. The topology of the rupture and the dimensions of the working face are also shown in Fig. 2.

2.1.1. Mathematical model

The objective of the CFD study is to solve for the flow field for three-dimensional fluid flow taking into account ruptures and bends in the auxiliary duct. To enhance the accuracy of the analysis, surface roughness has also been incorporated into the model. For the full-scale three-dimensional structure, the resolution of the turbulent flow field requires solving continuity and momentum conservation equations. The mathematical equations are presented here in tensor notation for the Cartesian coordinate system. Equation (1) is the mathematical representation of the principle of conservation of mass and is most commonly referred to as