International Journal of Mining Science and Technology 25 (2015) 107–112

Contents lists available at ScienceDirect

International Journal of Mining Science and Technology

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

Numerical simulation for propagation characteristics of shock wave and gas flow induced by outburst intensity

Zhou Aitao [a,b,*], Wang Kai [a,b], Wang Li [a], Du Feng [a,b], Li Zhilei [a,b]

[a] *School of Resource & Safety Engineering, China University of Mining and Technology, Beijing 100083, China*
[b] *State Key Laboratory of Coal Resources and Mine Safety, China University of Mining and Technology, Beijing 100083, China*

A R T I C L E   I N F O

Article history:
Received 20 April 2013
Received in revised form 5 June 2013
Accepted 28 July 2014
Available online 7 February 2015

Keywords:
Coal and gas outburst
Outburst intensity
Shock wave and gas flow
Propagation characteristics

A B S T R A C T

In order to analyze the propagation characteristics of shock wave and gas flow induced by outburst intensity, the governing equations of shock wave and gas flow propagation were put forward, and the numerical simulation boundary condition was obtained based on outburst characteristics. The propagation characteristics of shock wave and gas flow were simulated by Fluent software, and the simulation results were verified by experiments. The results show that air shock wave is formed due to air medium compressed by the transient high pressure gas which rapidly expands in the roadway; the shock wave and gas flow with high velocity are formed behind the shock wave front, which significantly decays due to limiting effect of the roadway wall. The attenuation degree is greater in the early stage than that in the late stage, and the velocity of gas convection transport is lower than the speed of the shock wave. The greater the outburst intensity is, the greater the pressure of the shock wave front is, and the higher the speed of the shock wave and gas flow is.

1. Introduction

Outburst shock wave and gas flow is pulverized coal–gas flow formed instantaneously after coal and gas outburst. It has strong destructiveness and can lead casualties of operating personnel and damage to the underground facilities directly. Gas flow can move at the opposite direction of airflow and fill a tunnel of tens to thousands meters, even the whole mine; destroy the mine ventilation system completely. Since high concentration gas is difficult to dilute and explosible and combustible, when gas concentration reaches a certain range as well as where there is fire in the tunnel, secondary disasters such as gas explosion, fire, and even coal dust explosion are possibly occur and will cause heavy casualties and property losses. Cheng [1–5] etc. investigated the formation and propagation of outburst shock wave by theoretical analysis and mathematical derivation. Otuonye and Sheng [6–9] studied the dynamic process of outburst shock wave and gas flow. The predecessors have done the preliminary exploration about the propagation characteristics of shock wave induced by outburst and gas flow reversal. Based on previous studies, Wang and Zhou [10–12] studied the formation mechanism and propagation characteristics in different types of tunnel systematically. But this study assumes that the area of outburst is full of high pressure gas, and when outburst occurs, the high pressure gas instantly emitted from the area of outburst forms shock wave and air flow. In fact, after the outburst, the gas is continuously desorbed from coal. The greater coal quantity outbursts in unit of time (outburst intensity) are, the greater the quantities of gas desorption are, and the stronger the impact force of shock wave and air flow is. This paper analyzes the effect of outburst intensity on the shock wave and air flow propagation characteristics systematically, and the results of this paper have important theoretical and practical significance on disaster rescue and effective prevention of secondary disasters of outburst mine.

2. Governing equations for outburst shock wave and the flow of gas flow

The governing equation for the flow outburst shock wave is put forward here. It is established based on the universal basic law of conservation such as mass conservation, momentum conservation, energy conservation, conservation of natural components [13–16]. The governing equation in two dimensions is as follow [15,16]:

---

[*] Corresponding author. Tel.: +86 15210567636.
E-mail address: cumtbzat@126.com (A. Zhou).

http://dx.doi.org/10.1016/j.ijmst.2014.12.009
2095-2686/

$$\left.\begin{array}{l}\frac{\partial\rho}{\partial t}+\frac{\partial\rho u_i}{\partial x_i}=0\\ \frac{\partial\rho u_i}{\partial t}+\frac{\partial\rho u_i u_j}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial\tau_{ij}}{\partial x_j}\\ \frac{\partial E_t}{\partial t}+\frac{\partial E_t u_j}{\partial x_j}=-\frac{\partial}{\partial x_j}\left[(P-\tau_{ij})u_i+\tau_{ij}u_j+q_j\right]\\ \frac{\partial C}{\partial t}+\frac{\partial C u_j}{\partial x_j}=\frac{\partial J_{xj}}{\partial x_j}\end{array}\right\} \tag{1}$$

where $\tau_{ij}=\mu\left[\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right]$; $E_t=\rho(C_V T+\frac{1}{2}u_i u_i)$; c_υ is the specific heat at constant volume, J/(m^3 K); C the gas concentration, %; E_t the total energy of the mixture, J; as two-dimensions equation, both i and j can be selected as 1 and 2, where 1 represents the x-direction and 2 represents the y-direction; u is the average velocity, m/s; J_x is the diffusion flux of gas, kg/(m^2 s), and $J_x=\Gamma\frac{\partial C}{\partial x}$; q is the heat flux due to conduction, w/m^2, and $q=-K\frac{\partial T}{\partial x}$; P the pressure of the mixture, Pa; K the thermal conductivity of gas; t the time; T the temperature of a mixture, K; Γ the diffusion coefficient for gases; ρ the density of air flow, kg/m^3; δ_{ij} the Dirac delta function; μ the dynamic viscosity, Pa s; and τ_{ij} the shear stress tensor, Pa.

3. Boundary conditions of numerical simulation

Boundary conditions used in the numerical computation include inlet boundary conditions, outlet boundary conditions, symmetric boundary conditions, and wall boundary conditions. The boundary conditions used in the process of the numerical simulation are analyzed as follows:

(1) Inlet boundary conditions

Once the outburst occurs, because the coal body suddenly releases the pressure, a lot of gas is released from the coal. Gas emission during outburst can be divided into two stages. First stage is the development stage. A lot of free gas is released from the coal body. The absorbed gas also has quick desorption and is emitted from coal at the same time of coal broken. Second stage is the mitigation stage. The broken coal and pulverized coal form. The gas around the coal hole is desorbed and emitted into the tunnel, however, the gas emission rate declines. Because gas emission quantity in the first minute after outburst is difficult to measure, only the law of gas released from coal particle can be analyzed [17–19]. The experimental results show that [20] the gas emission increases rapidly in the initial period, then gradually tends to be gentle. The release amount of gas with time can be expressed by a power function. The gas release rate in 0–3 min and time t represent power function relationship, which can be given as follows:

$$V_t=At^{-6} \tag{2}$$

where V_t is the gas release rate, cm^3/(g s); t the time, s; A and b are the regression coefficients, which can be obtained through experiment. The relationship of gas release rate and the release time is based on the data from Ref. [20].

As shown in Eq. (2), when the intensity of outburst is known, the gas amount released per unit time can be obtained. As shown in Fig. 1, Regard section AB as the quality entrance of numerical computation and the mass flow rate is the gas amount released per unit time. Besides the difference of initial entrance and boundary conditions, calculation conditions based on numerical outburst intensity are consist with the initial boundary conditions based on the gas pressure.

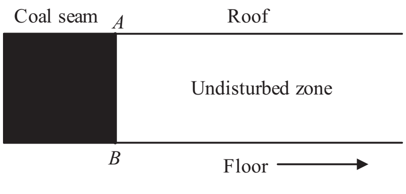

Fig. 1. Physical model of simulation.

(2) Wall boundary condition

The wall boundary is the most common used boundary condition. For calculation of the turbulent, turbulence will evolve into a laminar flow in the nearby wall region. The viscous force is of primary importance. Aiming at the simulation of nearby wall region, the wall function method is used. The viscous flow wall boundary condition is usually taken as the nonslip condition, which can be given as:

$$\rho u_i=0 \tag{3}$$

The wall boundary conditions for pressure can be expressed as:

$$\frac{\partial p}{\partial n}=0 \tag{4}$$

where n is the normal direction to the wall.

The state equation for an ideal gas can be described as:

$$\frac{\partial E_t}{\partial n}=0 \tag{5}$$

Meanwhile, considering the temperature condition at the wall, an adiabatic-wall can be assumed as:

$$\frac{\partial T}{\partial n}=0 \tag{6}$$

Based on the ideal gas law, the relationship of the flow density and gas concentration is as follows:

$$\left.\begin{array}{l}\frac{\partial\rho}{\partial n}=0\\ \frac{\partial C}{\partial n}=0\end{array}\right\} \tag{7}$$

(3) Symmetric boundary condition

If the physical model is symmetric, symmetric boundary condition is applied, so that only half of the model need to be solved, which can shorten the solving time. In symmetric boundary, vertical boundary can be zero and other physical quantities in the inside and outside of this boundary are equal. Hence, the condition can be described by the following Eq. (8):

$$\left.\begin{array}{l}\frac{\partial u_i}{\partial\theta}=0\\ \frac{\partial\rho}{\partial\theta}=0\\ \frac{\partial C}{\partial\theta}=0\end{array}\right\} \tag{8}$$

where θ is the direction normal to the symmetric plane.

(4) Outlet boundary condition

Outlet boundary conditions refer to the flow parameter of the physical model given at export position, including pressure, speed etc. Outlet boundary conditions generally select the places which are far enough from the physical model disturbance regions to apply, so that the flow can be in a fully developed state, and there was no change in the direction. In the model, the outlet boundary is located in distant places because the study area should not be affected by the boundary conditions. Outlet boundary pressure is equal to atmospheric pressure in tunnel, and other parameters gradients are zero, as can be described as follows:

$$\left.\begin{array}{l}\frac{\partial u}{\partial x_i}=0\\ p_{\text{outlet}}=p_a\\ \frac{\partial h}{\partial x_i}=0\end{array}\right\} \tag{9}$$

where i is the normal direction of outlet boundary plane which is perpendicular to the flow direction.