Study on spray cooling of heading face based on DPM model

Chen Li1, *, Haoyue Zhang1, Luna Sun1, Tianmo Yang1, Teng Zhang1, Haonan Zhang

1College of safety and environmental engineering, Shandong University of Science and Technology, Shandong, China

*Corresponding author: 201801010104@sdust.edu.cn

Abstract. In order to study the treatment of mine heat disaster and ensure the protection of the health of the staff in the heading face. In this study, a physical model was built based on the heading face of a metal mine, and the temperature field distribution of the roadway was simulated by computational fluid dynamics (CFD) software. The results show that spraying can significantly reduce the temperature of the roadway environment. This paper provides theoretical guidance for the application of spray device in underground mines.

Keywords: Thermal hazard, Spray cooling, Discrete phase, Numerical simulation

1. Introduction

With the progress of society and the further development of industry, the demand for mineral resources is increasing. In recent years, the number of shallow resources has been decreasing year by year. In the future, the mining depth of underground mineral resources will be deeper and deeper, and the underground rock temperature will increase with the increase of mining depth. The problem of mine heat damage will be very serious and common. According to the survey, the temperature of rock stratum will exceed that of human body in a deep well of more than one kilometer. For example, the temperature of the 920 meters deep coal mine in Japan is 40 degrees [1]; the maximum temperature of 1000-1200 meters in the Ruhr mining area in Germany is up to 60 degrees [2]; the study on the cooling direction of the heading face mainly focuses on the ventilation field, and Sun Yong and Nie Xiao et al. have carried out numerical simulation research on the ventilation and temperature reduction of the heading face.[3][4] Many experts and scholars at home and abroad pay attention to it. Wang et al. Studied the numerical simulation of single droplet impact.[5]

In this paper, according to the actual situation, the physical model is established by using rhino and ANSYS mesh. The grid is divided. Secondly, the effectiveness of the simulation is tested according to the actual situation. The test results prove that the model is scientific and effective. Fluent is used to simulate the situation of no spray in the laneway, and observe the temperature field distribution in the tunnel. Then the simulation of the spray is carried out, and the results are compared with the results without spray. The results show that the spray is in the high temperature and high humidity drivage lane. The tunnel can cool down the environment in the tunnel.
2. Mathematical model

In this paper, the commercial CFD software (ANSYS fluent 19.2) is used to calculate the air flow, heat and mass transfer in the process of roadway excavation. Single phase air model was used. Turbulence model is an important part to characterize the flow characteristics of underground environment. On the basis of comparing the calculation results of various turbulence models, the standard k-epsilon model which is most widely used in engineering field is selected for prediction. In the tunnel flow, mass, momentum and energy transport occur simultaneously. The conservation equations of mass, momentum and energy can be expressed as[6]

2.1. Continuity equation

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  

(1)

Momentum equation:

\[
\frac{\partial}{\partial t} \left( \rho \mathbf{v} \right) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla \mathbf{p} + \nabla \cdot (\mathbf{t}) + \rho \mathbf{g}
\]  

(2)

Where \( \rho \) is the static pressure, \( \mathbf{t} \) is the stress tensor, and \( \rho \mathbf{g} \) is the gravitational body force.

Energy conservation equation:

\[
\frac{\partial}{\partial t} \left( \rho E \right) + \nabla \cdot \left( \rho \mathbf{v} E \right) = -\nabla \cdot \left( \Sigma_j h_j J_j \right) + S_h
\]  

(3)

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k - \rho \varepsilon
\]  

(4)

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon G_k}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}
\]  

(5)

Where \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients, \( C_{1\varepsilon} \) and \( C_{2\varepsilon} \) are model constants, \( \sigma_k \) and \( \sigma_\varepsilon \) are the turbulent Prandtl numbers corresponding to the k equation and the \( \varepsilon \) equation, respectively, and \( \mu_t \) is turbulent viscosity given by:

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}
\]  

(6)

The values of \( C_{1\varepsilon}, C_{2\varepsilon}, C_{\mu}, \sigma_k, \) and \( \sigma_\varepsilon \) are 1.44, 1.92, 0.09, 1, and 1.3, respectively.

2.2. Discrete phase model

Lagrange method is used to calculate the discrete phase. A large number of computational regions, each of which represents a group of real water droplets with the same characteristics, are tracked through the flow field. The trajectory of each package and the mass transfer from it are calculated by the following instantaneous droplet motion equation. In this model, drag force and gravity are considered.[7]

The particle force equilibrium equation is
\[ \frac{du_{p,x}}{dt} = F_D(u_{x,y} - u_{x,z}) + \frac{g_z(p_x - p_z)}{\rho_p} \]
\[ \frac{du_{p,y}}{dt} = F_D(u_{y,x} - u_{y,z}) + \frac{g_z(p_y - p_z)}{\rho_p} \]
\[ \frac{du_{p,z}}{dt} = F_D(u_{z,x} - u_{z,y}) + \frac{g_z(p_z - p_y)}{\rho_p} \]  
(7)

Where
\[ F_D = \frac{18\mu C_D}{\rho_p d_p^2} \]  
(8)
\[ C_D = \frac{24}{Re} (1 + 0.186Re^{0.626}) + \frac{0.44Re}{7185.353 + Re} \]  
(9)

Re is the relative Reynolds number, defined as
\[ Re = \frac{\rho d |u_p - u_p|}{\mu} \]  
(10)

Trajectory equation of particles:
\[ \frac{dx}{dt} = u_{p,x} \]
\[ \frac{dy}{dt} = u_{p,y} \]
\[ \frac{dz}{dt} = u_{p,z} \]  
(11)

Due to the high humidity in the environment, the heat carried away by droplet evaporation is not considered. Therefore, a simple heat balance can be used to link the particle temperature TP (T) with convective heat transfer and radiation absorption / emission on the particle surface. The expression is
\[ T_p(t + \Delta t) = \alpha_p [T_p(t) - \alpha_p e^{-\beta_p \Delta t}] \]  
(12)

Where
\[ \alpha_p = \frac{h'T_p + \epsilon_c \sigma T_p^4}{h' + \epsilon_c \sigma T_p^4(t)} \]  
(13)
\[ \beta_p = \frac{A_c (h' + \epsilon_c \sigma T_p^4(T))}{m_p c_p} \]  
(14)

The convective heat transfer coefficient \( h' \) is calculated by the empirical correlation of Ranz and Marshall. [8]
\[ h' = k / d_p (2 + 0.6Re^{1/2} Pr^{1/3}) \]  
(15)
3. Modeling establish

3.1. Physical model

According to the actual situation of driving roadway in a gold mine, the corresponding physical model is established in rhino software. The physical model is composed of roadway, scraper and air inlet pipe. The roadway is arched, 40 meters long, 4.6 meters wide and 4.9 meters high, of which the straight wall is 2.6 meters high and the arch is 2.3 meters high. The LHD is 8.10 meters long, 1.82 meters wide and 1.99 meters high. The length of the air inlet is 28 meters, the diameter is 0.8 meters, and the distance between the air outlet and the working face is 12 meters. Figure 1 shows the physical model, and table 1 shows the size of the model, where the positive direction of the x-axis represents the direction from the central axis of the roadway floor to the roadway wall near the air duct, the positive direction of the y-axis represents the direction from the end of the roadway to the head part, and the positive direction of the z-axis represents the direction from the roadway floor to the top of the roadway.

![Figure 1. Three dimensional model](image)

**Table 1. Dimensions of 3D model**

| Property                  | Value              | Property                  | Value   |
|---------------------------|--------------------|----------------------------|---------|
| Size of tunnel            | 40m×4.6m×4.9m      | Length of air duct        | 28m     |
| Size of LHD               | 8.10×1.82m×1.9m    | Distance between inlet and working face | 12m     |
| Diameter of air duct      | 0.8m               |                            |         |

In the simulation of spray, combined with the actual situation in the roadway, we chose stainless steel wide angle cone stainless steel nozzles. The spray height is set to 2.8-3m more suitable. Here we set the spray height of 3M, and the spray nozzle is installed at the top of the middle of the roadway. Except for other locations, the sprinklers with the same parameters are all located at Y=2.3, and the distance from the floor of the roadway is 3 meters. There are 7 nozzles in total, and the interval between each two nozzles is 4 meters, which are respectively located on the plane of x = 28, x = 24, x = 20, x = 16, x = 12, x = 8 and x = 4.

3.2. Boundary conditions

According to the actual ventilation conditions of the heading face, the basic boundary conditions of the wind field can be determined. In order to make the air flow around the end
face of the roadway flow to the surrounding fully, the end face of the roadway is set as the outflow outlet, and the forced air duct and the wall of the roadway are set as the non-slip wall boundary; the standard wall function is selected to correlate the physical quantity of the roadway surface and the corresponding physical quantity of the turbulent core area. In addition, the wind speed of the inlet duct is 12 m/s. The DPM model can be used to simulate the discrete phase in the flow field and track the movement of particles. The result is intuitive, so it can be used to simulate atomized water particle spray and observe the effect of spray on the temperature of heading face.

Table 2. Betting of boundary conditions and related parameters

| Type          | Property                  | Value    | Type          | Property                  | Value    |
|---------------|---------------------------|----------|---------------|---------------------------|----------|
| General       | Solver Type               | Pressure-Based | Water-Liquid | Density               | 998.2kg/m³ |
|               | Time                       | Steady   | Wall          | Heat Flux               | 300W/m²  |
|               | Air                        | Density  | Wall of Scraper | Heat Flux               | 420W/m²  |
|               | Velocity                  | 12m/s    | Solution Methods | scheme               | SIMPLE   |
| Inlet         | Turbulent intensity       | 5%       | Injection Type | cone                  |          |
|               | Hydraulic Diameter        | 0.8m     | DPM            | Diameter Distribution | Rosin-rammler |
| Viscous Model | K-epsilon                 | standard | Total Flow Rate | 0.06kg/s               |          |
|               | Near-Wall Treatment       | Standard wall function | Cone Angle | 60°                   |          |

3.3. Grid generation and independence test

By adjusting the value of scale factor to change the grid density, three different quality grids of high, medium and low quality are generated. For the velocity distribution of x = 25, x = 15 and x = 5, fluent software is used to simulate the velocity field distribution of heading face under different grid quality. The obtained data are derived, analyzed and compared. The results are shown in Figure 2. The results are in good agreement with the measured data. When the grid quality is low, there is a certain deviation between the simulation results and the measured data, which means that the low quality grid is unreasonable. Considering the influence of simulation efficiency and other factors, medium quality mesh is selected for the geometric model, and the final generated mesh is as shown in Figure 2. There are 1415568 meshes in the geometric model, of which the minimum mesh quality is 0.10131, the maximum mesh quality is 1, and the average mesh quality is 0.84007. The mesh has no negative value and good quality, so it can be used in numerical simulation.
3.4. Verification of basic model

In order to verify the feasibility of the established model and boundary conditions, the air velocities of different sections were measured in the field, and the predicted results were verified by CFD. In this study, seven measuring points (x = 5m, 10m, 15m, 20m, 25m, 30m, 35m) are selected at the position of y = 2.3 on the z = 2.5 plane, which can fully represent the air flow characteristics in the roadway, as shown in Figure 3. The wind speed of these nine points was measured by electronic anemometer, and repeated three times. The average results were compared with the CFD prediction results, and the results are shown in Figure 3.

As shown in Figure 3, the wind velocity measured at each point shows a similar trend with the CFD prediction results. Due to the influence of transfer point, the wind velocity of each measuring point predicted by CFD fluctuates to a certain extent. The average error between the measured value and the simulated value is 5.11%. Considering the complexity of operation and ventilation conditions, the CFD prediction is not completely consistent with the experimental results. However, the average error is less than 10%, that is to say, the simulation result of air flow field is acceptable.
4. Results and discussion

4.1. Airflow temperature field distribution without spray

Figure 4 shows y=1.3, y=2.3 when there is no spray. Y = 3.3 three plane temperature nephogram, due to the low temperature of the air blown from the air duct, can absorb the heat existing in the roadway near the head-on area, so that the temperature of the roadway near the head-on area is reduced, the temperature here is about 299k, which can play a certain cooling effect, but due to the limitation of the wind speed and air volume of the ventilation system, the LHD, as a large heat source, can reduce the temperature so that the cold air flowing near itself is heated and expanded, the density is reduced, so it is lifted to the top of the roadway, and finally flows out along the top of the roadway. Because the air is heated and cannot flow through the lower working area, the temperature is higher, slightly higher than 303k. It can be seen that in some cases, the cooling effect can not be achieved only by means of ventilation. The temperature in most areas of the roadway is still very high, which will reduce the labor efficiency of the workers working there, and even endanger their health, which is not conducive to the mine Safe mining work is normal.

4.2. The distribution of airflow temperature field during spray

Take the three planes of y=1.3, y=2.3 and y=3.3 when spraying, make their temperature cloud chart (unit: k), as shown in Figure 5, observe its temperature cloud chart to compare, and compare the cooling effect of adding spray as auxiliary means. It can be observed that after adding the spray device, the temperature near the frontal surface is almost the same as that without spray. The temperature rising from the head area away from the frontal area will not increase as the distance increases. The overall temperature is about 302k. The temperature in the local area is slightly lower due to the nearer spray or the area flowing through the spray, indicating the obvious effect of spray. The temperature did not exceed 30 ℃, which was suitable for workers. Because the humidity in the roadway is too high and the relative humidity is close to 100%, the heat dissipated by evaporation and heat absorption is almost negligible. The atomized water particles ejected from the spray are relatively low in temperature, only 278k. There is a temperature difference with the roadway ambient temperature. According to the first law of thermodynamics, the heat can be transferred from the higher temperature

![Figure 4. Temperature cloud map without spray](image-url)
air to the atomized water particles with low temperature, so the ambient temperature in the roadway is low. The degree of stress decreased.

**Figure 5.** Temperature cloud map with spray

**4.3. Comparison between the use of spray and without the use of spray**

Fig. 6 is the temperature nephogram (unit: k) of the plane with $Z = 1.6$, which is roughly equal to the height of human head. Fig. (a) is the temperature nephogram of case 1. In the area near the roadway, the temperature is about 300K, and the cooling effect of the ventilation system is obvious. However, due to the heat exchange between the LHD and the surrounding rock and the air, the environment in the roadway behind the LHD is poor. The temperature rise is about 304K, which is not suitable for workers. In case 2 of B, the thermal environment in the working area is roughly the same as that in the case. However, due to the presence of spray, the temperature behind the scraper is low, most of the area is about 300K, and the temperature around the spray and passing through the area is about 298K, which is suitable for workers to carry out construction work. Therefore, it can be thought that adding spray device can effectively reduce the temperature in the environment.

**Figure 6.** Temperature comparison
5. Conclusion
1. This paper studies the use of ventilation to cool the temperature and not meet the required temperature, the use of spray can effectively reduce the temperature far away from the face area, and can improve the working environment in the roadway. In this paper, when the ventilation temperature is used only, the temperature inside the roadway can reach up to 305K, and the temperature in the tunnel after spraying is about 302k. It is beneficial to improve the labor efficiency and protect the health of workers.

2. Because the spray scope is small, it is not enough to affect the cooling effect of the whole tunnel. The influence of spraying is lower than that of the spray, and the temperature is higher. This requires the production of heat at certain locations during spray installation, and nozzles should first be installed in areas with higher heat production and higher temperature.

References
[1] Greth, A.; Roghanchi, P.; Kocsis, K. A review of cooling system practices and their applicability to deep and hot underground US mines. In Proceedings of the 16th North American Mine Ventilation Symposium, Golden, CO, USA, 17–22 June 2017; Volume 11, pp. 1–9.
[2] Kong, B.; Li, Z.H.; Wang, E.Y.; Yang, Y.L.; Chen, L.; Qi, G.S. An experimental study for characterization the.
[3] Sun Yong, Wang Wei. Numerical simulation of ventilation thermal environment of heading face based on fluent [J]. Coal science and technology, 2012 (07): 34-37
[4] Nie Xiaoye. Numerical simulation research on ventilation and cooling of heading face based on fluent [J]. Mining technology, 2015, v.15; No.76 (06): 36-38
[5] L. Wang, X. Huai, Y. Tao, Flow and heat transfer of micro-droplet impact on thin liquid film during spray cooling, J Eng Thermophys 31 (2010) 987–990.
[6] Hu S , Wang Z , Feng G . Temporal and Spatial Distribution of Respirable Dust After Blasting of Coal Roadway Driving Faces: A Case Study[J]. Minerals, 2015, 5(4):679-692.
[7] Hou Y , Tao Y , Huai X , et al. Numerical simulation of multi-nozzle spray cooling heat transfer[J], international journal of thermal sciences, 2018, 125:81-8.
[8] W.E. Ranz, W.R. Marshall, Evaporation from drops, Part II, Chem Eng Prog 48(1952) 173–180.