Numerical simulation study on Optimization of spray cooling mode in high temperature operation

Ningning Zhang*, Cong Zhang and Shuo Yuan
College of Safety and Environmental Engineering, Shandong University of Science and Technology, Qingdao, China

*Corresponding author: ningzhangn@163.com

Abstract. Taking a single mine roadway in a metal mine as the research object, the distribution of wind temperature and wind speed in a single heading is studied by numerical simulation method under the condition of different ventilation in the roadway. The influence rule of different spray angles on the temperature distribution of the roadway is discussed, and the conclusion that the spray angle of the case reaches the optimum cooling effect is 15 degrees. Numerical simulation results show that spray cooling can make up for the defects of traditional pressing ventilation mode and achieve better cooling effect. It should be widely applied in non coal and gas outburst roadway.

Keywords: Mine heat damage, Spray cooling, numerical simulation, Heading face.

1. Introduction
With the gradual reduction of shallow resources in China, coal mining is gradually entering into deep mining [1]. The resulting high ground temperature problem has brought great negative impact on the safety production of mine engineering. With the improvement of mechanization of metal mines, the thermal environment in this area is extremely bad. Gao Jianliang [2], Wang Haiqiao [3], Huang Shouyuan [4], foreign scholars such as Nakayama, Parra, sasmito, etc. have carried out a large number of numerical simulation studies on the thermal and humid environment, ventilation flow field and temperature field of driving face in domestic and foreign coal mines or metal mines. However, there is still some room for improvement in the research on the influence of the whole heat source of single heading on the thermal environment of the working face. This kind of absolute heat source is not fully considered in the numerical simulation. McPherson and Yang Deyuan and other scholars have proposed the calculation formula of this kind of heat source [5,6], but the key parameters in the formula are difficult to determine in the field.

In this paper, the thermal environment of a single heading is taken as the research object. The distribution of wind temperature in the working area under different spray angles is studied, and the optimal layout of spray angle is determined. Based on this, a new cooling scheme for solving the heat hazard problem is put forward by numerical simulation. The application of this scheme can achieve better cooling effect.
2. numerical simulation

In this study, Fluent software of ANSYS company is used, which is specially used to solve the ventilation system in the space environment. The built-in controls can quickly establish the physical model and grid generation. The finite volume method algorithm is used to calculate the flow field and temperature field by using Fluent solver. The Tecplot post-processing software is used to comprehensively check the wind temperature, wind speed and air humidity in the environment. It can be used to evaluate the temperature of human skin.

2.1. geometric model

The physical model of the working face is established. The model consists of driving roadway, spray system and air inlet pipe. The details are as follows: the excavation roadway is a cuboid of 40 m × 4 m × 3.1 m; the compressed air duct is an air duct with a diameter of 0.6 m, in which the mandrel is 2.1 m above the ground, 0.1 m away from the nearest tunnel wall, and the height of the forced air inlet is 10.0 m, which is the common height in field operation; the reserved loader and belt conveyor are located behind the fully mechanized roadheader. Figure 1 shows the physical 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 compressed air duct, the positive direction of the y-axis represents the direction from the head-on part to the roadway end, and the positive direction of the z-axis represents the direction from the roadway floor to the roadway top.

Fig. 1 Physical Model

2.2. mathematical model

In view of the three-dimensional steady flow field and temperature field in single heading working face, the mass conservation equation, momentum equation, energy conservation equation and turbulence model equation are solved by using solver. Before the numerical simulation of single heading roadway, the wind speed and wind temperature calculated by several common turbulence models, such as zero equation model, standard k-ε model and RNG K-ε model, are compared and analyzed with the measured results by anemometer and thermometer. The results show that the RNG K-ε model is more reliable and accurate than the standard k-ε model in the motion of large streamline curvature, which can describe the effect of small-scale turbulence. Therefore, RNG K-ε model is selected in this study.

The governing equations are as follows:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}(\alpha_i \mu_{ij} \frac{\partial k}{\partial x_i}) + G_k - \rho \varepsilon \tag{1}
\]
\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} (\alpha \varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_i}) + C_{1e} \frac{\varepsilon}{k} G_k - C_{2e} \rho \frac{\varepsilon^2}{k} - R_e
\] (2)

In the simulation software, \(\alpha_k = \alpha \approx 1.393\), \(\mu_{eff}\) is the effective viscosity coefficient \((\text{PA} \cdot \text{s})\), \(C_1 \varepsilon\) and \(C_2 \varepsilon\) are model constants, \(G_k\) is the generation term of turbulent energy \(k\) caused by average velocity gradient \((\text{PA} / \text{s})\), and \(R \varepsilon\) is an additional term.

Underground coal mining face, the airflow flows through the spray area and heat exchange. The water mist ejected from the nozzle is composed of water droplets with micron diameter. The heat transfer area of the air and water increases greatly, so as to accelerate the heat exchange efficiency of the droplet and the underground hot air. Because of the small size of fog particles, the heat exchange with air flow can be regarded as a transient heat and mass transfer process. In this process, the heat is transferred from the high temperature gas to the low temperature droplets, and the water vapor formed by evaporation is released into the ambient gas in the form of convection and diffusion.

The convective heat transfer between droplet and air is a short stage before droplet evaporation. The droplet in convective heat transfer stage is in a quasi steady gas phase environment, and the gas boundary layer around the droplet has uniform physical properties. The heat exchange between water mist and ambient gas is caused by the temperature difference between them. The heat transfer rate is proportional to the temperature difference \((T_G - T_P)\) and the surface area \(\pi D^2\). The proportional coefficient \(h\) is the heat transfer coefficient and the mass transfer number \(B_M\) is defined.

\[
B_M = \frac{Y_{ps}}{1 - Y_{ps}}
\]

\[
Q = h \pi D^2 (T_G - T_P)
\]

A dimensionless Nusselt number \(Nu\) in heat transfer is introduced to express the strength of heat transfer,

\[
Nu = \frac{hD}{\lambda_g} = 2 \left( \ln \left( \frac{1 + B_M}{B_M} \right) \right) \frac{hD}{\lambda_g}
\]

The convective heat transfer between droplet surface and ambient gas is as follows,

\[
Q = 2 \pi D \lambda_g (T_G - T_P) \left( \ln \left( \frac{1 + B_M}{B_M} \right) \right)
\]

In the steady-state evaporation process, the heat flux of droplet evaporation is as follows,

\[
Q = m_p L
\]

Where is droplet evaporation,

\[
L = 2502535259 - 2385.76424(723.16).
\]
The difference between $Q$ and $Q_E$ is the heat absorbed by the droplet in the unsteady evaporation process,

$$Q - Q_E = 2\pi D \lambda_g \ln(1 + B_M \left(\frac{T_g - T_s}{B_M} - \frac{L}{c_{pe}}\right))$$

### 2.3. boundary conditions

In order to verify the consistency between the simulation results and the measured results, the boundary conditions are determined according to the measured values and the calculated values.

1. It is assumed that the wind flow is a low velocity incompressible gas, the density conforms to the approximate assumption, the heat distribution on the surrounding rock surface is uniform, and the thermal physical parameters are constant, and the wall roughness is uniform;

2. Inlet and outlet boundary: the inlet boundary wind temperature is set as the same as the actual situation, the initial velocity of air flow is $v = 12.1$ m/s, the supply air temperature is $t = 20 ^\circ$C, and the outlet boundary is set as free reflux;

3. Wall boundary: the nonsliding boundary condition is applied to all wall surfaces, and the heat dissipation of surrounding rock is 134.8 W/m², assuming that the surface temperature is evenly distributed.

### 2.4. grid generation and grid independence test

Finite element mesh generation is very important in numerical simulation, which directly affects the accuracy of subsequent simulation results. In mesh generation, the hexahedral network with high efficiency and good quality should be selected as the mesh type. In addition, considering that the current three-phase flow problem requires a lot of calculation, the hexahedron network is more desirable. In addition, the fluid region is divided into three parts: fluid zone-01, fluid zone-02 and fluid zone-03 by using hexahedral partition method. The grid tool in ICEM CFD is used to generate block structure from top to bottom, and the grid density and node distribution are adjusted reasonably. Generally speaking, increasing the network density can improve the calculation accuracy, but it will also increase the calculation cost. In this study, three kinds of grids with different densities are generated, i.e. fine grid, very fine grid and medium grid. Secondly, the independence of grid is tested by comparing the wind velocity of different sections in the roadway, as shown in Fig. 4. It can be seen that the wind speed error of medium grid is 10% larger than that of fine grid, while the wind speed error of ultra fine grid is about 2% lower than that of medium grid, which indicates that the numerical simulation with very fine grid is feasible. Finally, 191128 grids were generated. Through the quality test, the overall quality of the network is higher than 0.45. Fig. 5 shows the result of mesh generation.

![Grid](image_url)

Fig. 2 Grid
3. Simulation results and discussion

(a) No spray

(b) The nozzle direction is the negative direction of Z axis
The nozzle direction is the positive direction of Z axis

Nozzle tilt down 45 degrees

Fig. 4 Temperature distribution on the section with different nozzle placement

Fig. 4 (a) shows the temperature field distribution on the section A without the spray cooling in the traditional ventilation mode. The coal mine safety regulation stipulates that the air temperature of the mining face of the production mine should not exceed 299K. From the chart, we can see that most of the sections of A on the cross section are above 300K when the spray cooling is done without ventilation and cooling. That is to say, most of them are areas where the temperature is not up to the standard. The traditional cooling method is to increase the wind speed or reduce the supply air temperature through the air cooler. A nozzle is installed at the first support of the air inlet roadway, that is, the nozzle coordinates are (2.2, 7, 3.5), the nozzle is set with a pressure of 2MPa, and the water temperature is 282k. Fig. 4 (B-D) shows the temperature field distribution on section a under three different sprinkler arrangements.

It can be seen that after installing the sprinkler head, no matter which layout form is adopted, the high temperature environment under the mine can be improved to varying degrees. When using the traditional downward injection method, as shown in Fig. 4 (b), there is an obvious cooling area below the sprinkler. The spray will be exchanged with the air inside the working area to heat the hot air in other areas until the temperature rises, and the effective cooling distance is about 60m.

In order to prolong the time of spray in the air, it is found that the temperature distribution of the sprinkler arrangement is changed to the upward spray, as shown in Fig. 4 (c), which has a larger influence range compared with the traditional downward spray phase, which is due to the longer spray spray particles in the air. However, because of the small particles of high pressure spray water mist, the spray particles after spraying are mainly affected by the wind flow, and the gravity is less affected. The spray is constantly conducting heat transfer during the downward process. The temperature of the working area where the workers are not yet reached has been increased, and the cooling effect has not been cooled. It can not effectively improve the high temperature condition of the floor area of the roadway, resulting in the below level of the sprinkler. The high temperature in the working area is not alleviated.

After analyzing the advantages and disadvantages of upward spray, it is found that when the nozzle direction changes to a certain angle, the spray particles have higher longitudinal exit velocity. Under
the action of gravity, the trajectories of spray particles change from short straight line to parabolic motion, that is, the distance between water particles moving in space is longer, and the time is longer. The heat exchange of the environment is more sufficient, so the cooling effect is more obvious. As shown in Fig. 4 (d), the direction towards the air inlet, i.e. the negative direction of Y axis, is 45 ° to the horizontal line. Compared with the traditional downward spray direction, it has a long influence range, about 75m, and the vertical direction is more uniform, and the area with poor cooling effect behind the nozzle at the same height is smaller. Obviously, the placement of nozzle is an important factor to be considered in the installation of spray cooling system for coal face, which is especially important for the optimization of spray cooling system.

4. Conclusion
In conclusion, the temperature field distribution of different placement methods of a nozzle in the selected roadway is comprehensively analyzed, and the safety of underground operators is considered. The conclusion is that the best nozzle placement mode for the selected coal mining face is 45 ° against the wind. Therefore, it is suggested that the spray method should be adopted in mine cooling, and the best angle is 45 degrees. The spray cooling proposed in this study provides a new idea for mine thermal hazard prevention and control.

References
[1] Gu Desheng, Zhou Keping. The development theme of modern metal mining [R]. Review and prospect of China's mining technology for ten years.
[2] Gao Jianliang,Xu Wen,Zhang Xuebo. Treatment of water evaporation during temperature and humidity calculation of surrounding rock heat dissipation[J].Journal of China Coal Society,2010,35(06):951-955.
[3] Wang Haiqiao, Shi Shiliang, Liu Ronghua, Liu Heqing. NUMERICAL SIMULATION STUDY ON THE FLOW AND FLOW FIELD IN THE TUNNEL TUBE WITH TUBE[J]. Journal of China Coal Society, 2004, 29(4): 425-428.
[4] Huang Shouyuan. Analysis of deep heat damage and thermal environment of heading face in Zhouyuanshan Coal Mine [D]. Changsha, Hunan: Hunan University of Science and Technology, 2011.
[5] McPherson, M.J. The analysis and simulation of heat flow into underground airways[J]. Int. J. Min. Geol. Eng,1986(4):165-196.
[6] Yang Deyuan, Yang Tianhong. Mine Thermal Environment and Its Control [M]. Beijing: Metallurgical Publishing House, 2009.