

Numerical simulation for the ventilation form of the underground main workshop in the hydropower station

LONG Tianyu, YANG Yurong, LUO Qingping

Institute of Urban Construction and Environmental Engineering, Chongqing University, Chongqing, 400045, P.R.China

Abstract: By using a three-dimensional $k-\varepsilon$ turbulent model, the ventilation form of the underground main workshop in a certain hydropower station at non-isothermal condition has been designed optimally by means of a numerical simulation. A model experiment was made for the chosen airflow pattern, and the experiment results verified the validity of the numerical simulation.

Keywords: numerical simulation; ventilation form; main workshop; hydropower station

1. Introduction

In an underground main workshop of a big or moderate hydropower station, a good airflow pattern is helpful to creating a healthy, comfortable environment for working personnel and to raising labor efficiency. In this paper, a numerical model applying the computational fluid dynamics technique has been used to design optimally the ventilation form of the underground main workshop in a certain hydropower station.

The length of the underground main workshop that is in want of designing ventilation form is 204m, its width 24.5m and its height 21.5m. The workshop consists of nine similar set sections, and the structure sketch of one of them is shown in figure 1. Airflow is let vertically into the workshop through the inlets in the ceiling, and discharged from outlets on the ground. In each set section, there are six airflow inlets with same diameter and same flux (Fig. 1), and there are two airflow outlets (Fig. 2), one of which is used as a hoist hole and the other as a stair hole. In order to provide a healthy, comfortable airflow environment for personnel in the workshop, in the process of designing airflow pattern has need of taking into consideration for following several aspects. In the first, the distribution of airflow velocity in the work zone of the workshop ought to be maintained as evenly as possible. In the second, the magnitude of airflow velocity ought to be limited in the range of 0.4~0.7m/s. In the last, the air reflux in the work zone ought to be reduced as full as possible. For that, we have designed optimally the ventilation form of the workshop by means of numerical analysis with respect to some factors such as the minimum flux, the inlet place, the inlet size and so on. A model

experiment was made for the adopted airflow pattern so that we could compare modeling results with experiment results to assure the validity of the numerical modeling.

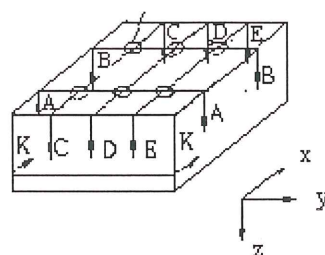


Fig.1 set section

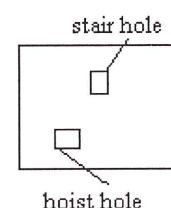


Fig. 2 The ground plane

2. Governing equations

Assuming that the airflow is stable and the heating of generating sets may be regarded approximately as point heat sources, a three-dimension $\kappa-\varepsilon$ turbulent model is chosen to solve the flow field considering that the air flow in the workshop is a three-dimension turbulent flow. As a matter of convenience, the equations in the turbulent model are transformed into the dimensionless ones. Here, the airflow velocity of inlets, w_0 , is taken as the character velocity, the diameter of inlets, d_0 , as the character length, and the ΔT_0 ($\Delta T_0 = T_B - T_0$) is taken as the character temperature. The temperature T_B is the temperature of the ground of the workshop that is assumed to be constant, and T_0 is the airflow temperature flowing into the workshop. The expressions of the dimensionless parameters are defined in Table 1. In this way, the dimensionless equations governing the airflow can be described by the convection-diffusion conservation equations which all have the same form given as follows:

$$\frac{\partial}{\partial x_j}(u_j \Phi) = \frac{\partial}{\partial x_j}(\Gamma_\Phi \frac{\partial \Phi}{\partial x_j}) + S_\Phi \quad (1)$$

where Φ is a variable that represents respectively u_j (the j -th component of the airflow dimensionless velocity), k (dimensionless turbulent kinetic energy), ε (dimensionless turbulent kinetic energy dissipation rate), T (dimensionless temperature) and so on, Γ_Φ and S_Φ represent the diffusion coefficient and the source term depending on the variable Φ . The specific expressions of Φ , Γ_Φ and S_Φ are shown in Table 2.

Table 1 The definition of dimensionless parameters

Parameter	u_i	p	x_i	κ	ε	T	q_T
Definition	$\frac{u_i}{w_0}$	$\frac{p}{\rho w_0^2}$	$\frac{x_i}{d_0}$	$\frac{\kappa}{w_0^2}$	$\frac{\varepsilon d_0}{w_0^2}$	$\frac{T - T_0}{\Delta T_0}$	$\frac{q_T d_0}{\rho c_p \Delta T_0}$

Table 2 The specific expression of Φ , Γ_Φ and S_Φ

Equation	Γ_Φ	Φ	S_Φ
Momentum	ν_{eff}	u_i	$-\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu_{\text{eff}} \frac{\partial u_j}{\partial x_i} \right) + A_r T \delta_{3i}$
κ	$\Gamma_{\kappa, \text{eff}}$	κ	$\nu_i \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \sigma_\mu \frac{\kappa^2}{\nu_i}$
ε	$\Gamma_{\varepsilon, \text{eff}}$	ε	$\sigma_1 \sigma_\mu \kappa \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \sigma_2 \frac{\varepsilon^2}{\kappa}$
Energy	K_{eff}	T	q_T
Continuity	0	1	0

In Table 2,

$$\nu_{\text{eff}} = \frac{1}{\text{Re}} + \nu_t, \quad \Gamma_{\kappa, \text{eff}} = \frac{1}{\text{Re}} + \frac{\nu_t}{\sigma_\kappa}, \quad \Gamma_{\varepsilon, \text{eff}} = \frac{1}{\text{Re}} + \frac{\nu_t}{\sigma_\varepsilon}$$

$$K_{\text{eff}} = \frac{1}{\text{Re Pr}} + \frac{\nu_t}{\sigma_T}, \quad \text{Re} = \frac{w_0 d_0}{\nu}$$

$$A_r = \frac{Gr}{\text{Re}^2}, \quad Gr = \frac{g \beta \Delta T_0 d_0^3}{\nu^2}, \quad \text{Pr} = \frac{\mu c_p}{\lambda}$$

$$\nu_t = \frac{\sigma_\mu \kappa^2}{\varepsilon}, \quad \beta = \frac{1}{T_s}, \quad T_s = \frac{T_B + T_0}{2}$$

The values of $\sigma_1, \sigma_2, \sigma_\mu, \sigma_\kappa, \sigma_\varepsilon, \sigma_T$ are model constants, which are taken as [1]:

$$\sigma_1 = 1.44, \sigma_2 = 1.99, \sigma_\mu = 0.09$$

$$\sigma_\kappa = 1.0, \sigma_\varepsilon = 1.3, \sigma_T = 0.9$$

3. Boundary conditions

The governing equations require to be closed with appropriate boundary conditions. In this particular case, three types of boundary conditions have to be

considered: inlets, outlets, and walls. For selected Cartesian coordinate system (shown in Fig. 1), the boundary conditions of Eq. (1) are given below.

The boundary condition of the walls:

$$u = 0, v = 0, w = 0, \kappa = 0, \varepsilon = 10^{-24}$$

$$T = 1 \text{ (the ground)}, \quad \frac{\partial T}{\partial n} = 0 \text{ (the rest)}$$

where n is the normal direction of wall.

The boundary condition of the airflow inlets [2,3]:

$$u = 0, v = 0, w = 1, \kappa = 0.04, \varepsilon = 0.008, T = 0$$

The boundary condition of airflow outlets:

$$u = 0, v = 0, \quad \frac{\partial \kappa}{\partial z} = 0, \quad \frac{\partial \varepsilon}{\partial z} = 0, \quad \frac{\partial T}{\partial z} = 0$$

The vertical velocity w of the outlets is acquired by the inner velocity distribution [4]. For example, in Fig. 3, assuming that the relative change rates of vertical velocity at the nodes on the cross section of the outlet are constant, so that

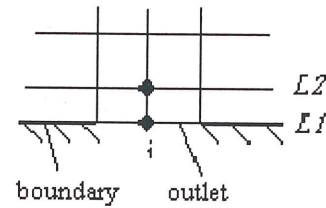


Fig. 3. Outlet boundary

$$\frac{w_{L1,j} - w_{L2,j}}{w_{L2,j}} = \text{const} = A \quad (2)$$

Therefore

$$w_{L1,j} = (A + 1)w_{L2,j} = fw_{L2,j} \quad (3)$$

$$f = A + 1.$$

The condition that the factor f is ascertained is that $w_{L2,j}$ has to meet the mass conservation laws in the whole flow field, then

$$\rho w_{L1,j} A_j = \rho f w_{L2,j} A_j = F_{in} \quad (4)$$

$$f = \frac{F_{in}}{\rho w_{L2,j} A_j} \quad (5)$$

where F_{in} is the mass flux of inlet; A_j is the area of cross section perpendicular to z direction; ρ is the density of airflow.

As the factor f has been fixed on, the vertical velocity on cross section at the outlet may be evaluated by Eq. (3).

4. Numerical simulation procedures

In order to obtain the numerical solution of the above governing equations together with the corresponding boundary conditions, these partial

differential equations need to be transformed into the coordinate discretized equations. Here, equation (1) is discretized employing the finite-volume method [4]. In view of the designed main workshop makes up of nine similar set sections, one of them is chosen as the computational domain (Fig. 1). The computational space is discretized into non-uniform cells, and the discretion equations are solved with the SIMPLE algorithm [4]. The convergence criteria are to ensure the total normalized residual error to be less than 2%.

5. Results and discussion

For different Reynolds numbers, numerical simulations of the ventilation form of the main workshop have been conducted considering some factors such as the inlet place, the inlet diameter, the inlet flux and the like. An optimal airflow pattern is selected out finally after analyzing and comparing many airflow patterns, for which the minimum flux of the inlet is $41.5 \times 10^4 \text{ m}^3/\text{h}$, the inlet diameter 0.5m ; the center distance of neighboring inlet is 7.3 m long and 10 m wide.

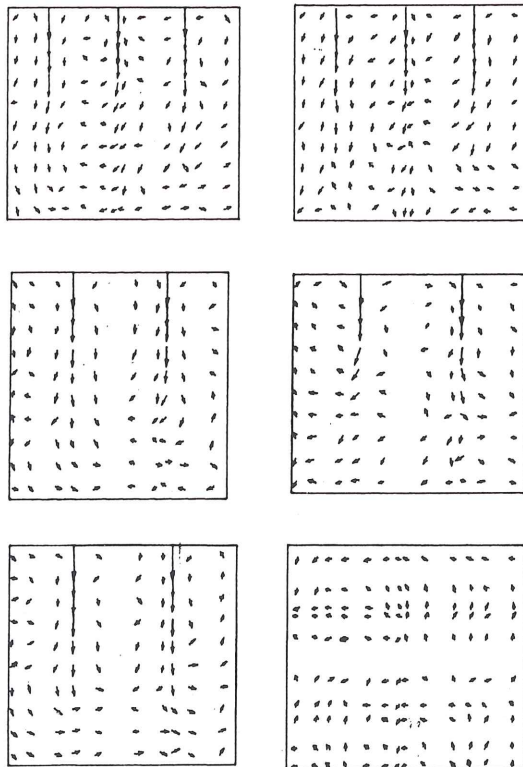


Fig. 4 The section velocity vectors

The calculation results for the chosen pattern are presented partially in Figs. 4 and 5. Fig. 4 gives the views of velocity vectors of the six cross sections labeled in Fig. 1 (here, K-K section is located in the

work zone), and Fig. 5 gives the velocity distribution of the work zone. Figure 4 indicates that the airflow entering the work zone flows out mostly through the exits while flows up nearly invisible. Therefore, the airflow reflux in the work zone is quite small, and the quality of airflow is desire. Fig. 5 indicates that the distribution of airflow velocity in the work zone is relatively uniform, and keeps in the range of $0.4 \sim 0.7 \text{ m/s}$. From the calculating results, it is come to the conclusion that the present design basically ensures a healthy, comfortable air environment for the working personnel in the workshop.

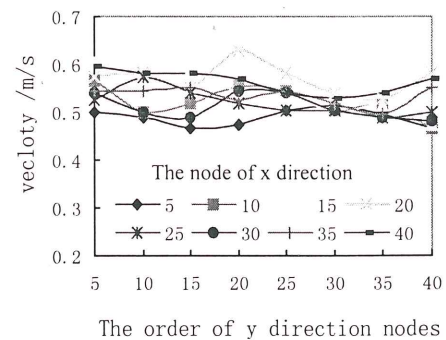


Fig. 5 The work zone velocity

6. Model experiment

To verify the validity of the numerical simulation, a model test for the present airflow pattern was conducted. Figure 6 illustrates the comparison between the test values of velocity and its corresponding calculation values in two locations which errors are relatively bigger. As shown in the Fig. 6, the calculation values are basically identical with the experimental values.

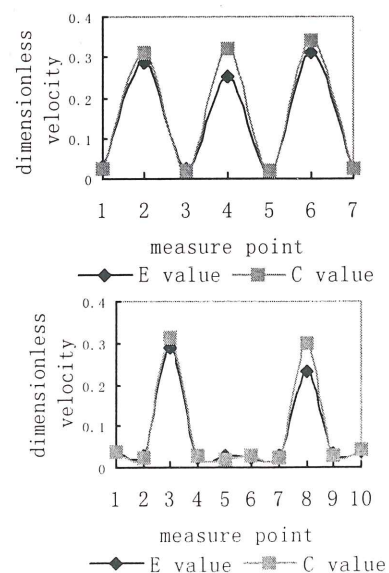


Fig. 6 The comparison of experiment value (E value) with

calculation value (C value)

7. Conclusion

The optimal design of the ventilation form of the underground main workshop in a certain hydropower station under non-isothermal condition has been conducted employing a $k-\varepsilon$ turbulent model. The model test has verified the validity of the design pattern. For the chosen airflow pattern, the working personnel in the workshop could obtain a desire ventilation form and better airflow quality. Presently, this airflow pattern has been applied to the construction of the hydropower station.

Notations

Ar =Archimedes number

Gr =Grashof number

Pr =Prandtl number

Re =Reynolds number

d_0 =diameter of inlets

T =dimensionless temperature

u_j = j th component of the dimensionless velocity

w_0 =velocity of inlets

k =dimensionless turbulent kinetic energy

ε =dimensionless dissipation rate

ν =kinematic viscosity

ν_t =turbulent viscosity

ν_{eff} =turbulent effective viscosity

$\Gamma_{\kappa,eff}$ =effective diffuse coefficient of k

$\Gamma_{\varepsilon,eff}$ =effective diffuse coefficient of ε

K_{eff} =effective diffuse coefficient of T

ΔT_0 =character temperature

References

- 1 Chen Q, Moser A, Suter P. A numerical study of indoor air quality and thermal comfort under six kinds of air diffusion. ASHRAE Transaction, 1992, Vol.98: 203-217.
- 2 Murakami S, Kato S. Numerical and experimental study on room airflow—3-D predictions using the $\kappa-\varepsilon$ model. Building and Environment, 1991, Vol.24 (1): 85-97.
- 3 Foluso L. CFD Applications In the HVAC&R Industry. ASHRAE Journal, 1997, Jun: 44-48.
- 4 Cheng Ck. Computational Fluid Dynamics. Chongqing: Chongqing Publishing House, 1999