STUDY ON VENTILATION EFFECT FOR A LARGE SPACE IN AN UNDERGROUND HYDROPOWER STATION

Zhihe Su[1], *Yanfeng* Li[1*], and *Wei* Tian[1]

[1] Beijing Key Laboratory of green built environment and energy-efficient technology, Beijing University of Technology, Beijing, China.100124

Abstract. Due to the urgent need for reducing carbon emissions, an increasing number of pumped storage power stations have been constructed and used considering its obvious advantages of energy saving. However, relevant design guidance and technical research failed to meet the development demand. There is only a small number of studies focusing on the prediction and evaluation of ventilation system in single-story stations, however, little on the integral connected plant floors. Thus, it is significant to study safety operation technology of deep underground pumped storage power stations, especially in terms of thermal and humidity environment. A reduced-scale (1/30) model based on an engineering project was set up, and numerical simulations were carried out to study the temperature and velocity distribution in the large underground space under the summer design condition (L=21.1×10^{4}m^3/s, T=19℃, Q=997KW). Results show that the temperature on the right side of the powerhouse is slightly higher than that on the left side, and the gap is approximately from 1℃ to 4℃. Moreover, local overheating may occur in some places. The side air supply velocity and location should be paid more attention to in designing air distribution.

1 Introduction

With the global demand for greenhouse gases reduction and environmental protection, hydro became one of the most important sources of energy. More than 40,000 underground hydropower stations have been built in China [1–3]. Good ventilation and airflow will directly affect the staff's health, work efficiency, and the normal operation of the equipment in the underground hydropower station [4]. Therefore, it is very important to ensure the effectiveness of airflow ventilation and air conditioning.

The CFD (Computational Fluid Dynamics) numerical simulation and model experiment are usually used in the study of the ventilation of underground, large-space buildings [5]. Li et al. [6] adopted the CFD and PIV methods to analyse the air movement in the generator floor and proved that the RKE turbulence model had a good performance in the large-space jet. Li et al. [7] studied the air distribution in HOHHOT power station by model experiments. Liu et al. [8] proposed a reformed RNG k-ε model, and then the accuracy and reliability of CFD for prediction of ventilation system in large-space was verified.

Overall, factors affecting the air distribution in underground, large-space buildings and ventilation effect have been widely studied by model experiments or numerical simulations attributing to economic conditions, whereas the number of studies focusing on the effect of ceiling air supply on the generator floor and studies on the integral connected plant floors are relatively fewer. Therefore, numerical simulation and reduced scale experiments were carried out in this study, to investigate the air distribution in each floor of the main powerhouse on the basis of many factors such as ceiling air supply, side air supply, staircases and corridors.

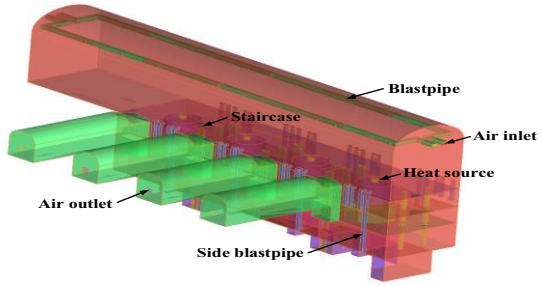
2 Methodology

2.1 Numerical simulation

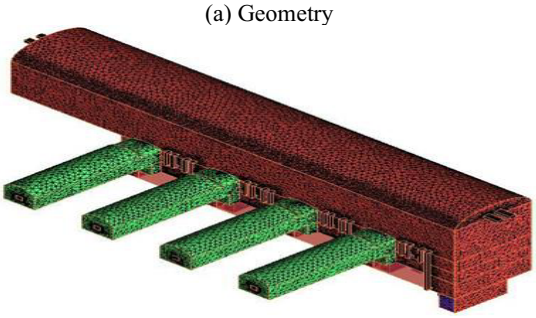
2.1.1 Physical model

In this paper, a full-scale hydropower station is selected as the research object. As shown in Fig. 1(a), the structure mainly includes the following sections: generator floor; busbar floor; turbine floor; volute floor; tailwater gallery floor; and four corridors. The dimension of the powerhouse is 138.7m (L) × 24m (W) × 54.2mm (H). Four corridors with same dimension of 39.6m (L) × 9.7m (W) × 10.1m (H) are evenly distributed along the sidewall. Meanwhile, the ICEM mesh generator associated with the solver has been used to mesh to figure of the powerhouse in Fig. 1(b).

* Corresponding author: liyanfeng@bjut.edu.cn



(a) Geometry



(b) Grid

Fig. 1. Numerical model and grid of the main plant building with four corridors.

The air system comprises two parts: supply and exhaust system (Fig. 2). Thirty-six circular supply inlets (d^*=630mm) are installed at the top of the powerhouse, and totally four circular exhaust outlets (d^*=1980mm) are located at the end of the transport passages to supply and remove ventilated air, respectively. At both sides of the main powerhouse, 36 ducts are used for the side air supply.

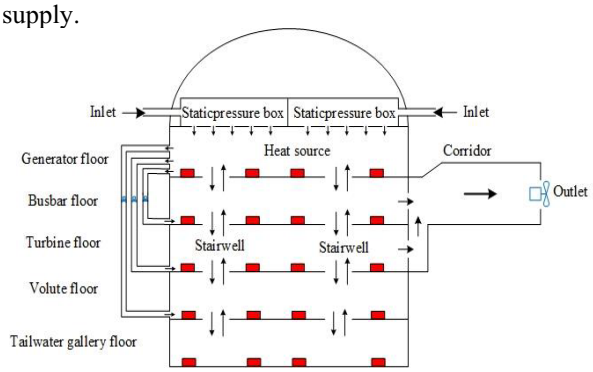


Fig. 2. Air distribution in the main powerhouse and transport passages.

2.1.2 Numerical method and boundary conditions

ANSYS Fluent commercial software was used for the numerical simulation. In this study, FLUENT 19.0 was used to solve the fluid flow governing equations. The governing equations of mass and momentum can be written as the following generic form:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho u_i\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_i}{\partial x_j}-\rho u_i' u_j'\right)+\rho g_i \tag{2}$$

A realizable k-ε turbulence model was applied, which had a good performance in rotating flow [8], boundary layer flow, separation flow, and secondary flow. At the same time, the SIMPLEC algorithm was selected considering that it has been proved to accelerate convergence in problems [6].

The velocity boundary condition was applied to the inlet; the outlet was defined as the exhaust fan. A fan was arranged inside the side air duct. The turbulence intensity and hydraulic diameter were used as the inlet and outlet turbulence boundaries.

2.2 Experimental setup

A 1:30 scaled model of deep underground pumped storage power station was set up to investigate the air distribution in the underground hydropower station, as shown in Fig.3. The model is made of 100 mm thick wooden material expect for the end side, which is made of 10 mm-thick glasses to ensure the unit operation can be observed. Two static pressure boxes are set up on the top of the powerhouse to ensure that 36 circular supply inlets have the same air supply rates. Four heating tungsten lamps with a power of 300W are used as heat sources on each floor. And the voltages of each layer are adjusted by a transformer to control the heat release rate. The lamps are placed inside the perforated sheet iron to keep heat release well-distributed.

Fig. 3. Designed scale ventilation model for deep underground pumped storage power station.

In the previous study, Archimedes number has been shown to be more important than the Reynolds number [9].

$$Ar=\frac{v^2 T_s}{g d_0\left(T_a-T_s\right)} \tag{1}$$

By keeping the same Archimedes number, the relevant parameters of the model and the full scale are shown in Tab. 1.

Table 1 Scale ratios of parameters.

Name	C_l	C_t	C_v	C_G	C_Q
Value	1/30	1	1/5.48	1/4929	1/4929

There were 36 measuring points in the operating area of each floor located 60mm above the floor, as shown in Fig. 4. In order to take measures conveniently, the hole for velocity measurement was pre-reserved in the wall.

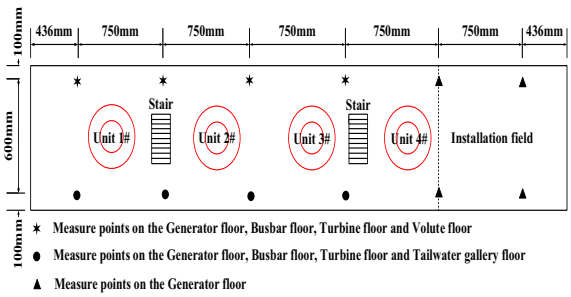


Fig. 4. Measuring points arrangements in each floor.

2.3 Grid sensitivity analysis

Although numerical simulation is widely used in ventilation calculations, to select appropriate calculation parameters, engineers must have a good understanding of the calculation principles behind it. It is necessary to analyse the mesh grid size sensitivity before performing numerical calculations, a suitable grid size can greatly reduce the calculation time while ensuring the accuracy of the calculation. In the paper, four different grid sizes were examined. Fig. 5 shows the temperature attenuation in the horizontal direction on the upstream side of the generator floor under summer condition. There is essentially no difference in the predicted temperature attenuation of the four grids at the dimensionless length of 0.8-1.0. However, the temperature attenuation of 1,506,412 gird numbers and 2,154,261 gird numbers is smaller and more uniform than the other two for dimensionless length less than 0.8. Thus, 1,506,412 grid numbers were finally selected to balance computation time and precision.

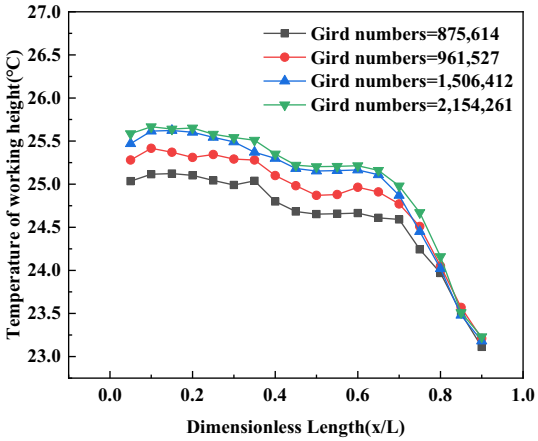


Fig. 5. Comparison of four different grids.

2.4 Model validation

Fig. 6 shows the comparison of the temperature field of the generator floor between numerical simulation and model experiment under the summer condition. It can be seen that the temperature field of numerical simulation results are in acceptable agreement with the results of the model experiment, indicating that the realizable *k-ε* turbulence model is applicable for complex shear flows with heat sources such as occur in room ventilation.

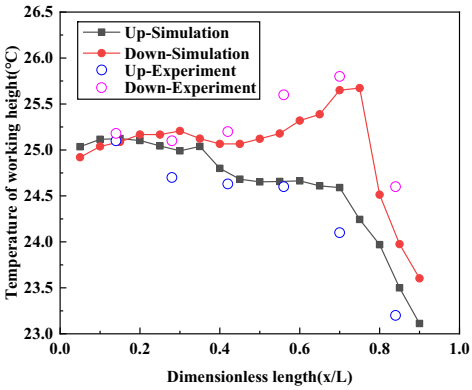
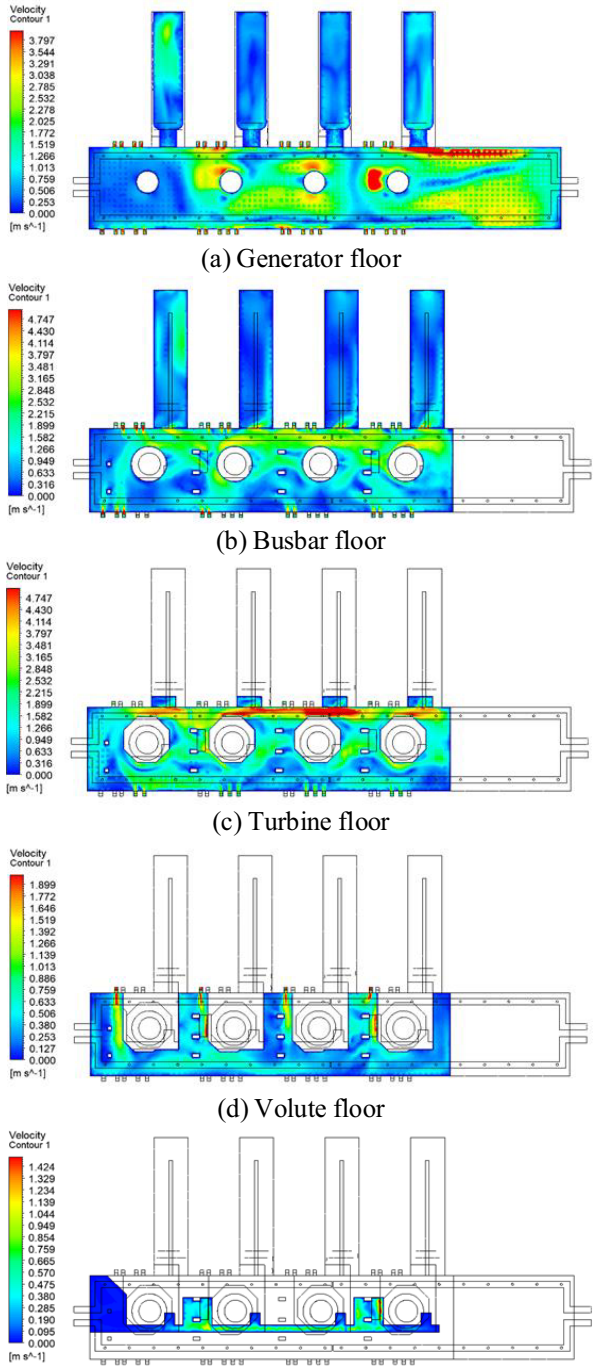


Fig. 6. Comparison of temperature field of generator predicted by FLUENT to those experimental determined by model experiment.

3 Results and discussion

3.1 Velocity field

In practical engineering, due to the height restrictions of special equipment in hydropower plants, staff will often stand to operate the equipment within 2 m height. Therefore, the velocity data under 2 m height were studied to analyse the velocity field in the workspace. The velocity graph of the airflow at each floor is shown in Fig. 7. This graph illustrates that the maximum speed of the airflow occurs in areas near the side air vents, and shield of the units. Cycle flow areas exist in the corners formed by the vertical walls on the volute floor and tailwater gallery floor.



(a) Generator floor

(b) Busbar floor

(c) Turbine floor

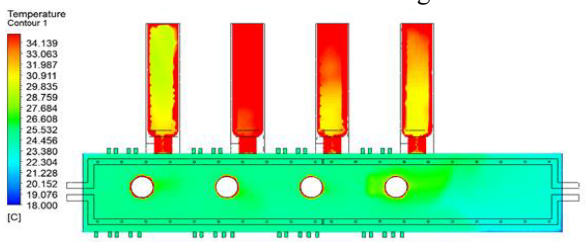
(d) Volute floor

(e) Tailwater gallery floor

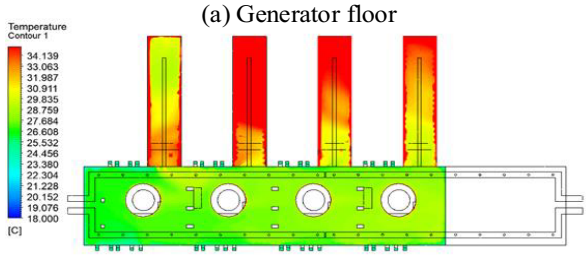
Fig. 7. Velocity contour at each floor.

3.2 Temperature field

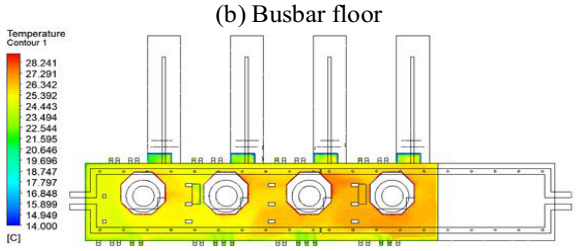
The temperature field distribution in each floor of working height in summer is shown in Fig. 8(a–e), respectively. Fig. 8 shows that the temperature on the right side of the main plant building is slightly higher than that on the left side. Moreover, local over-heating occurs in the volute floor with a maximum temperature of 25.5℃, as shown in Fig. 15(d). This is because the existence of a dead zone was ignored in the air supply design. At the same time, this graph illustrates that the temperature distribution is not symmetrical. Overall, the temperature on the right side of the main plant building is 1℃ to 4℃ higher than that on the left side. However, the installation field of the generator hall was on the right side of the main plant building, in which there was no heating equipment. Hence, the temperature in the installation field was lower than in the generator area.



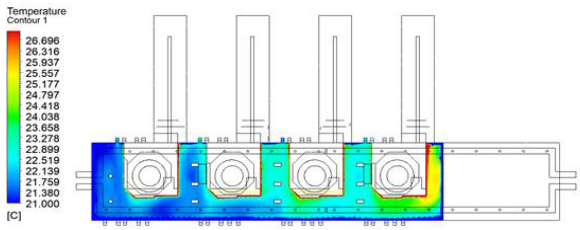
(a) Generator floor



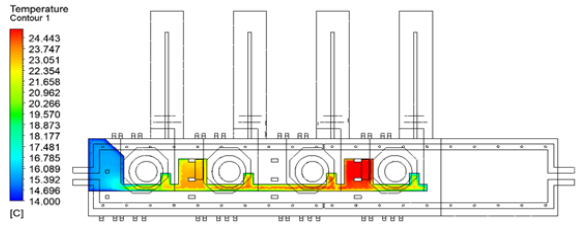
(b) Busbar floor



(c) Turbine floor



(d) Volute floor



(e) Tailwater gallery floor

Fig. 8. Temperature contour at each floor.

4 Conclusion

CFD techniques and 1:30 model experiments are comprehensively used to analyse the flow fields for underground hydropower station on each floor under design jet velocity and heat source. The following conclusions can be drawn:

1) The temperature on the right side of the powerhouse is slightly higher than that on the left side, and the gap is approximately from 1℃ to 4℃.
2) The occurrence of the dead zone shows that a reasonable arrangement of the positions of the side air vents can greatly enhance ventilation.

References

1. Z.G. Sun, L. Zhang, Z.D. Duan, China Water Resour. 7 (2013) 12–13.
2. Y.M. Weng, Water Power 37 (3) (2011) 18–20.
3. A.G. Li, C.Q. Yang, T. Ren, Energy Build. 127 (2016) 259–267.
4. Y.S. Yu, X.T. Li, W.X. Shi, Appl. Therm. Eng. 29 (14) (2009) 3254–3260.
5. Y.L. Ren, Tianjin University, Tianjin, 2008 (in Chinese).
6. Li A, Ren T, Yang C, et al, Building and Environment, 2018, 131(mar.):16-31.
7. Li A, Qin E, Xin B, et al, Energy Conversion & Management, 2010, 51(1):33-41.
8. Liu Y, Wang S, Deng Y, et al, Applied Thermal Engineering, 2016, 105:151-158.
9. T. Kobayashi, M. Sandberg, H. Kotani, L. Claesson, Build. Environ. 45 (2010) 2723–2734.