Numerical Simulation of Connected Flow in the Inlet and Outlet of the Pumping Storage Power Station

Gaohui Li¹,², Xumin Wu¹, Minjie Yao¹, Ning Liu¹, DanZhen wangjia¹,², WeiJie Cui¹

¹Power China Huadong Engineering Corporation Limited, Zhejiang Province, 311122, China
²College of Water Conservancy and Hydropower Engineering, Hohai University, Jiangsu Province, 210098, China

*Corresponding author’s e-mail: li_gh2@ecidi.com

Abstract: Pumping and storage power stations with different hydraulic units are simultaneously in power generation and pumping operation (same-in-one pumping operation) is an effective means to alleviate the adverse effects caused by the sudden power imbalance of the power grid and adapt to the emergency dispatching requirements of the power grid. In this mode of operation, the water flow in the different inlet and outlet ports is reversed, and water flow communication will be formed in the front pool. Aiming at the problem that the connected flow is difficult to measure in physical model test and prototype observation, a three-dimensional flow field numerical simulation is used to determine the flow connection path and orifice velocity distribution, and the area-weighted average method is used to obtain the connected flow calculation method for different paths. Taking the inlet and outlet of a pumped storage power station as an example, this method is used to analyze the flow rate of the orifice and its proportion. The results show that when the pumped storage power station is running in the same pumping operation, the flow rate of some orifices will exceed 80% of its total flow. It is easy to cause the cooling water of the unit to circulate between the hydraulic units and keep heating up, thus reducing the cooling effect of the unit.

1. Introduction

Pumped-storage power station is currently the most economical large-scale energy storage power source. The unit can be converted in power generation and pumping conditions and no-load spinning reserve. It has the characteristics of flexible operation mode and fast response speed. At present, pumped-storage power plants are mainly responsible for grid frequency regulation, valley filling, phase modulation, and accident backup functions[1]. However, as the troughs are absorbed from outside the area and the installed capacity of new energy increases, sudden power imbalances may occur in the grid. In order to alleviate the adverse effects caused by this phenomenon and adapt to the emergency dispatch of the power grid, the pumped-storage power station has been proposed for the simultaneous generation and pumping operation mode. In this operating mode, different hydraulic units are in power generation and pumping conditions at the same time, so that the power grid can switch quickly when power imbalance occurs, and the stable operation of the power grid can be maintained.

Pumped-storage power station with simultaneous generation and pumping operation means that different hydraulic unit units are in power generation and pumping operation conditions at the same time[2]. In this operation mode, part of the water flow in the reservoir area flows into the inlet/outlet from
the inlet hydraulic unit, while the water flow in the outlet hydraulic unit flows into the front pool, and the water flows at different inlets and outlets are reversed[3]. When the distance between the inflow and outflow hydraulic units is far, and the front pond is wider, the water flow from the outflow unit into the reservoir area will gradually spread in the form of submerged jets, and the reverse water flow will form a backflow at the far end of the reservoir area. When the distance between the inlet and outlet hydraulic units is relatively close, the flow velocity is small, and the diffusion space in the front pool is insufficient, the two water flows will be connected in front of the water inlet/outlet, as shown in Figure 1. In general, the inlet and outlet of the lower reservoir of the pumped-storage power station are relatively close to the unit. Once the flow connection occurs in the front pool, the cooling water of the unit enters the reservoir area from the outlet hydraulic unit, and then enters the hydraulic unit in the form of connected flow. It enters the cooling water system of the unit again, and finally forms a hot water circulation, which continuously heats up, thereby affecting the cooling effect of the unit.

Figure.1 Flow velocity distribution and flow pattern in pumped storage power station

This paper will establish a three-dimensional numerical simulation model of the flow field at the inlet/outlet of the pumped storage power station, analyze and study the flow velocity distribution characteristics of the reservoir area under the same pumping operation mode, and on this basis, combine the physical model verification data to propose a connection flow calculation method. Finally, taking a pumped-storage power station as an example, the flow change rule of the inlet/outlet connection flow under the condition of simultaneous generation and pumping operation is clarified, which provides a basis for the feasibility demonstration of the pumped storage power station’s simultaneous generation and pumping operation.

2. Mathematical model and calculation method

2.1. Basic control equations

Realizable k-ε model is derived from a transient NS turbulence equations, the standard k-ε turbulence model equation similar form, except that the introduction of a new eddy viscosity equation molding process of turbulent viscosity coefficient, To make the calculation results more suitable for separation and shear flow, the basic governing equation is as follows:(1) Continuous equation

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]  

(2) Momentum equation

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\]

Where: t is the time; \(u_i\) and \(x_i\) are the velocity component and the coordinate component respectively; \(P\) is the corrected pressure considering the gravity; \(\mu_t\) is the turbulent viscosity coefficient; \(\rho\) and \(\mu\) are the functions of the volume fraction, which are calculated by the volume fraction weighted average method Obtained by the following formula:

\[
\rho = a_w \rho_w + a_d \rho_d, \mu = a_w \mu_w + a_d \mu_d
\]
Where: \( \rho_w \) and \( \rho_a \) are the density of water and gas respectively; \( \mu_w \) and \( \mu_a \) are the molecular viscosity coefficients of water and gas respectively,

\[
\mu_w = 1.31 \times 10^{-3} \frac{(N \cdot s)}{m^2}, \quad \mu_a \approx 0, \quad a_w + a_a = 1
\]

(3) Realizable k-\( \epsilon \) turbulence equation

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_i} \right] + G_{k,\epsilon} - \rho \varepsilon
\]

(4)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_i} \right] + \rho C_{\mu} \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + \frac{\varepsilon}{k} C_{\kappa} C_{\kappa} \tan \left( \frac{\nu \varepsilon}{\mu} \right)
\]

(5)

Where: \( k \) and \( \varepsilon \) are turbulent kinetic energy and turbulent energy dissipation rate respectively;

Is the turbulent Prandtl number of \( k \). In the high Reynolds number flow, the turbulent viscosity coefficient among

\[
C_{\mu} = 1/(A_{\mu} + A_{\kappa} k U^*/\varepsilon)
\]

\[
A_{\mu} = \sqrt{6} \cos \left( 1 \left( 3 \cos^{-1} \left( \sqrt{6} \left( S_{ij} S_{jk} S_{ki} / \sqrt{S_{ij} S_{ij}} \right) \right) \right) \right),
\]

\[
U^* = \sqrt{S_{ij} S_{ij} + \left( \Omega_{ij} - 2 \varepsilon_{ij} \omega_k \right) \left( \Omega_{ij} - 2 \varepsilon_{ij} \omega_k \right)}
\]

\[
\Omega_{ij} = \left( \partial u_i / \partial x_j - \partial u_j / \partial x_i \right) / 2
\]

\[
S_{ij} \quad \text{is the average strain rate tensor, } S_{ij} = \left( \partial u_i / \partial x_j - \partial u_j / \partial x_i \right) / 2,
\]

\[
A_{\mu} = 4.04, \quad G_{\mu} = -\rho u_i \mu_j \frac{\partial u_j}{\partial x_i}, \quad \text{Indicates the additional item of turbulent kinetic energy due to the average flow velocity gradient; } \nu \text{ is the coefficient of kinematic viscosity; } \nu \text{ and } \mu \text{ are the vertical and horizontal components of the fluid velocity in the control grid, respectively; } \sigma_{\varepsilon} = 1.3 \text{ is the turbulent Prandtl number of } \varepsilon ; \quad C_{1\varepsilon} = 1.9; \quad C_{2\varepsilon} = 1.44.
\]

2.2. Calculation method

The VOF (volume of fluid) method is a numerical method for solving the incompressible viscous free surface flow under a fixed grid. It is suitable for the tracking calculation of the interface between two or more non-penetrating fluids[6]. The model introduces a volume fraction variable for each phase, and determines the phase interface by solving the volume fraction value in each control unit. Suppose the volume fraction of the qth phase in a certain control unit is \( a_q \). When \( a_q = 0 \), there is no qth phase fluid in the control unit, When \( a_q = 1 \), the control unit is filled with the qth phase fluid, When \( 0 < a_q < 1 \), the control unit contains the phase interface, and the sum of the phase volume fractions in each control unit is equal to 1, that is:

\[
\sum_{q=1}^{n} a_q = 1
\]

(6)

\( a_q \) should satisfy the equation

\[
\frac{\partial a_q}{\partial t} + U_i \frac{\partial a_q}{\partial x_i} = 0
\]

(7)

In this paper, the VOF method is used to solve the gas-liquid two-phase flow, so \( n = 2 \). For equation (7), the implicit difference scheme is used to calculate the volume fraction of the control unit, the finite volume method is used to discretize the control equations, and the control equations are transformed into algebraic equations through the second-order upwind equation. The pressure-velocity coupling adopts the PISO algorithm, the discrete equations are solved by the ADI method, and the time difference adopts the fully implicit format.

2.3. Boundary conditions

As shown in Figure 1, the inlet/outlet of the lower reservoir of a pumped-storage power station, the
structure of the transition section, diffusion section, and anti-vortex beam section is shown in Figure 2. The single inlet/outlet of the lower reservoir has a height of 7.0m and a total width of 13.9m. The 1.3m thick branch pier divides it into two holes, each with a net width of 4.8m. The entrance section is 7.6m long, with three 1.0×1.2m (width×height) anti-vortex beams on the top, and the trash rack is close behind the anti-vortex beam. The length of the gradual change section is 17.4m, the longitudinal section changes from a height of 7.0m to 4.4m, and the vertical diffusion angle is 8.3°. The plane changes from a clear width of 10.9m to 4.4m, and the plane spread angle is 22°. The total width of the front of the six water inlets/outlets is 121.8m, and they are arranged side by side outside the excavation slope of 1:0.5. The elevation of the inlet/outlet bottom plate is 285.0m. The upper boundary of the mathematical model is taken to the tunnel section 200m upstream from the end of the diffusion section of the inlet and outlet, and the lower boundary is taken to the channel 500m downstream of the inlet and outlet. In the case of unidirectional pumping (power generation), the upper boundary is the outflow (inflow) boundary, the pressure boundary condition is given, and the lower boundary is the inflow (outflow) boundary, and the velocity boundary condition is given. The upper boundary is the combined condition of flow velocity and pressure under the condition of simultaneous emission and same pumping, and the lower boundary is the solid wall.

3. Connection flow calculation method
To determine the connected flow, firstly, the flow velocity at different positions of the orifice must be known to calculate the flow rate, and secondly, the flow communication path between different orifices must be determined according to the trajectory of the water particles at different positions of the orifice\[5\]. For this reason, the connected flow algorithm shown in Figure 3 is established to study the distribution law of the connected flow of the inlet/outlet when the pumped-storage power station is running at the same time. This algorithm first meshes the front cross section of the inlet and outlet orifices into i×j fluid surface elements, and forms the unit flow velocity matrix V according to the numerical simulation results of the flow velocity distribution at the front cross section of the inlet/outlet under the same flow and pumping operation conditions:

\[
V = \begin{bmatrix} v_{11} & \cdots & v_{ij} \\ \vdots & \ddots & \vdots \\ v_{1i} & \cdots & v_{ij} \end{bmatrix} \quad (8)
\]

Then the flow matrix of each fluid panel is:

\[
Q = sV \quad (9)
\]

In the formula, \( s = S/i/j \) is the fluid surface area, and \( S \) is the cross-sectional area of the front edge of the orifice. Secondly, according to the numerical simulation results, determine the flow communication path between each face element of a certain outlet orifice and other face elements of the inlet orifice, as shown in Figure 3, the element \( n_{ij} \) and the inlet flow on the front cross section of the outlet orifice \( M \) The flow rates of the elements \( m_{ik} \) on the front cross section of the orifice \( M \) communicate with each other. As a result, the corresponding relationship between the communication flow between a certain outflow

![Figure 2 Structure arrangement of the inlet/outlet of a pumped storage power station](image-url)
orifice and each inflow orifice can be established. If there is a face element in the outflow orifice N connected with the flow of the inlet orifice L, and b face elements are connected with the flow of the inlet orifice M, the connected flows are respectively:

\[ q_{NL} = \sum_{NL=1}^{a} Q_{NL} \]  

\[ q_{NM} = \sum_{NM=1}^{b} Q_{NM} \]  

In the formula, \( Q_{NL} \) is the outflow flow rate of a panel, and \( Q_{NM} \) is the outflow flow rate of b panel elements.

Figure 3. Schematic diagram of the connected flow calculation method

4. Connected flow distribution characteristics
Through the aforementioned mathematical model, the flow velocity distribution and the connected flow path of the inlet/outlet forepool of a pumped storage power station shown in Fig. 1 and Fig. 2 are studied in the same generation and pumping operation. For the side inlet/outlet, when the connected flow occurs, the vertical gradient of the flow velocity is smaller than the horizontal gradient. Therefore, when gridding the front edge section of the orifice, the horizontal scale of the panel should be smaller than the vertical scale\(^6\). As shown in the figure, a single inlet/outlet of the power station is 7.0m high, each hole has a net width of 4.8m, a face element width of 0.2m, a height of 0.7m, \( i=20 \), \( j=10 \). In this grid method, the water flow in the reservoir area will be divided into 10 layers, and the connection path of the fluid in this layer can be determined by the elementary streamlines of each flow layer. Figure 4 shows the simulation results of flow velocity distribution and connected flow path in the reservoir area of \( j=1 \) when the inlet and outlet of 1#~3# enter and the outlet of 4#~6# flows out. The layers are similar and will not be repeated here.

![Figure 4: Simulation results of flow velocity distribution and connected flow path](image)

1#Inlet/Outlet Outlet Path  
2#Inlet/Outlet Outlet Path
Figure 4. Flow distribution and connected flow path of the inlet and outlet water in a pumped storage power station

It can be seen from Figure 4 that when the pumped storage power station is running at the same time, a certain proportion of the flow at each inlet/outlet comes from the reservoir, and the flow at the inlet/outlet is not completely connected. In addition, the communication flow path between the orifices is not single, and the communication flow of a certain inlet orifice may originate from multiple outlet orifices. In short, during the operation of simultaneous pumping and pumping, the flow pattern of the water in the reservoir area is relatively complicated, and the connected flow has a certain relationship with the topography of the reservoir area and the distance between the inlet and outlet.

According to the aforementioned connection flow calculation method, the inflow and outflow flow of each orifice is 61.3m3/s, and the connection flow between the orifices is shown in Tab1. According to the data in the table, in this operation mode, there are connected flow rates between the 1# and 2# water inlets and the other three water outlets (4#, 5#, 6# water inlets/outlets), and the 3# water inlet and 4# There is a connected flow between the water outlets. Among them, the 1# and 2# inlet/outlet connecting flow accounts for 44.34% and 80.65% of the orifice longitudinal flow respectively, and the connecting flow comes from the three outflow orifices;4# and 5# inlet/outlet connecting flow accounts for 67.12% and 58.21% of the orifice longitudinal flow respectively, and the connecting flow comes from the three inlet orifices; About 17.00% of the inlet flow of 3# inlet/outlet is taken from 4# inlet/outlet in the form of connected flow; Approximately 11.77% of the outflow flow from the 6# outlet enters the 1# and 2# inlet/outlet in the form of connected flow. The remaining flow of each orifice diffuses after entering the reservoir area or enters the flow channel from the reservoir area.

| Connected flow rate | 1#  | 2#  | 3#  | 4#  | 5#  | 6#  |
|---------------------|-----|-----|-----|-----|-----|-----|
| 4#                  | 4.78| 25.93| 10.43| 67.12%|      |      |
| 5#                  | 15.49| 20.20| 0| 58.21%|      |      |
| 6#                  | 3.90| 3.31| 0| 11.77%|      |      |
| Connected flow rate | 44.34%| 80.65%| 17.00%|      |      |
5. Conclusion
(1) When pumped-storage power stations are running at the same time, a certain proportion of the flow at each inlet/outlet comes from the reservoir area, and the flow at the inlet/outlet is not fully connected. In addition, the communication flow path between the orifices is not single, and the communication flow of a certain inlet orifice may originate from multiple outlet orifices.
(2) When the diffusion space of the reservoir area is small and the water inlet/outlet are close together, the connected flow between the inlet and outlet orifices is more obvious, and the connected flow of some orifices even exceeds 80% of the total flow.

References:
[1] ZHANG, J.J., MAO, G.H., CHENG, W.P., HU, Y.J., (2008) Hydraulic shape optimization on lateral inlet/outlet of pump storage plant. J., Department of Hydraulic and Ocean Engineering, Zhejiang University, Hangzhou, 46-70.
[2] SHA, H.F., ZHOU, H., HUANG, D.J., (2009) Numerical simulation on the side inlet/outlet of pumped storage power station. J., Journal of Hydroelectric Engineering, 84-88.
[3] ZHANG, L.D., (2010) Flow dividing characteristics of intakes/outlets in upper and lower reservoirs of Xiangshuijian Pumped Storage Station. J., Nanjing Hydraulic Research Institute, Nanjing, 48-52.
[4] ZHANG, C.L., ZHU, H.H., ZHONG, W.Q., HUANG, Z.M., Experimenta, (2005) Research on inlet/outlet hydraulics of pump storage power plant. J., Guangdong Research Institute of Water Resources and Hydropower, Guangzhou, 60-63.
[5] WU, S.Q., YU, D.Y., (1990) Calculation and Research on Mathematical Model of Water Diversion System of Xianghongdian Pumped Storage Power Station. J., Nanjing Institute of Hydraulic Research.
[6] GAO, X.P., YE, F., SONG, H.F., (2006) Three-dimensional Numerical Simulation of Water Flow Movement at Side Inlet/Outlet. J., Journal of Tianjin University, 518-522.
[7] ZHANG, C.L., ZHU, H.U., ZHONG, W.Q., (2005) Experimental study on hydraulics of inlet/outlet of pumped-storage power station. J., Journal of Hydroelectric Engineering, 60-63.