Numerical study on Unstable Hydraulic Factors of Kaplan turbine with semi-spiral case at large flow rate conditions

Yaping Zhao 1, Zhihua Li 1,2※, Mengfan Dang 1, Jianjun Feng 1
1. Institute of Water Resources and Hydro-electric Engineering, Xi’an University of Technology, Xi’an 710048, China
2. Xi’an thermal power research institute co., ltd, Xi’an 710054, China
E-mail: zyp0168@163.com

Abstract. The operation of semi-spiral case Kaplan turbines at large flow rate conditions is restricted by factors such as severe cavitation and severe vibration, which limit the maximum output of the unit. In order to explore the main hydraulic factors that cause the poor operating stability of the unit at large flow rate conditions, and provide a reference for the operation and design of the turbine, the numerical simulation method is adopted to conduct the performance study of a semi-spiral case Kaplan turbine at different operating conditions, by comparing and analysing the results at the optimal conditions and large flow rate conditions, the unstable hydraulic factors at large flow rate conditions is obtained. The research results show that: at large flow rate conditions, the uneven flow distribution in the semi-spiral case cause vortices between the guide vanes, lead to the large fluctuations of blade torque, and generate unstable back-flow and vortices in the draft tube. These phenomena combined with the various gap flows in the runner, make the internal flow of the Kaplan turbine at large flow rate conditions deteriorate, cavitation increase, efficiency decrease, and cause severe vibration, thereby limiting the stable operation of Kaplan turbine at large flow rate conditions.

1. Introduction
Kaplan turbines have obvious advantages in the development of low-head hydraulic resources due to their large flow capacity, high average efficiency, and wide operating range. However, due to its structural characteristics, there are some shortcomings in the operation of the turbine, such as the uneven outflow of the semi-spiral case, the unstable clearance flow in the guide vane and runner, which causes the efficiency loss of the Kaplan turbine to increase and deterioration of cavitation performance and poor stability. It is especially serious at the large flowrate condition, which greatly affects the safe and stable operation of the Kaplan turbine and restricts the maximum output of the unit.

At present, the research on Kaplan turbine design method [1], performance research method [2], flow state and matching of flow components is to focus on improving the performance of the turbine at normal operating conditions. Lingjiu Zhou[3] found that the number of grids, grid form, and grid quality have a great influence on the simulation results of hydraulic turbine characteristics. It is necessary to adopt suitable grid processing methods according to different research problems. Alok [4] performed CFD analysis on the performance of a micro Kaplan unit at the design conditions and three operating conditions at partial load, and based on this, it carried out blade optimization design and performance evaluation. Zoran [5] conducted flow field calculation and strength analysis on Kaplan turbines with different blade thicknesses, and revealed the influence of blade thickness on the stress.
intensity parameters of the turbine. Daoli Zhao [6] studied the cavitation phenomenon caused by the vortex in the rim clearance of the Kaplan turbine, analyzed the strength of the cavitation vortex in the rim clearance and the damage position of the blade surface, and based on this, proposed the prevention of cavitation on the blade surface. Methods of destruction. Fengbin Zhao [7] studied the vibration characteristics of the rotating parts of the unit caused by the periodic interference force caused by the uneven and unstable draft tube incoming flow to the unit. Kenji SHINGAI [8] takes the weighted sum of the estimated efficiency of the three operating conditions as the objective function, and uses the simulated annealing algorithm to optimize the blades of the Kaplan turbine. The numerical analysis on the problem of poor matching between the fixed guide vane and the movable guide vane of the non-snail-shaped part of the spiral case at the optimal working conditions of the Kaplan turbine is carried out in the literature [9], and the flow pattern distribution between the fixed guide vane and the movable guide vane in the optimal working condition is effectively improved by optimizing the fixed guide vane geometry.

From the above study, it should be clear that the current research on Kaplan turbines is mainly focused on conventional operating conditions, and there is limited benefit in expanding the operating area of the turbine. Therefore, the problem of limited operation of turbines at large flowrate conditions still exists. Based on this, this paper uses numerical simulation methods to study the flow field distribution of the Kaplan turbine, by comparing the flow difference between the optimal working condition and the large flowrate condition; the unstable hydraulic factors that cause the poor stability of the unit at the large flow rate condition are revealed. It provides reference for the further optimization design and operation of the Kaplan turbine.

2. Geometric model and operating points
In this paper, a Kaplan paddle turbine model machine which includes 28 vanes, 28 guides, and 5 blades, is considered (figure 1). The runner diameter is 0.4 m, the hub ratio is 0.4375, and the relative guide vane height is 0.4. The main hydraulic factors that affect the stable operation of the turbine at the large flow rate conditions is obtained by comparing the performance of the optimal operating condition and the large flow rate condition, The calculated operating conditions are shown in figure 2.

![Kaplan Turbine Model](image1)

Figure 1. Geometric model of Kaplan turbine

![Operating Conditions](image2)

Figure 2. Operating conditions to calculate

3. Geometric model and mesh generation
Due to the complex geometry of Kaplan turbine, the tetrahedral mesh that is very flexible for irregular areas is used for the computational domain mesh generation of the Kaplan turbine.

In order to save calculation cost and ensure calculation accuracy, this study takes the turbine efficiency as the irrelevant verification parameter, and performs the grid independence verification at the optimum operating conditions, as shown in figure 3. The total number of grid elements selected for
the full flow channel is \(2479 \times 10^4\), and the grid distribution of each flow component is shown in Table 1.

### Table 1. Elements and notes number of grid

| Flow components                     | Number of grid Nodes/10^4 | Number of grid elements/10^4 |
|-------------------------------------|---------------------------|------------------------------|
| Semi-spiral case and stay vane     | 142.77                    | 725.16                       |
| Guide vane                         | 201.95                    | 1021.57                      |
| Runner                             | 109.15                    | 564.17                       |
| Draft tube                         | 29.75                     | 168.09                       |
| **Total**                          | **483.62**                | **2479**                     |

Figure 3. Grid independence verification

Any complex flow process in nature is governed by continuity equations, momentum conservation equations and energy conservation equations. It is generally considered that water flow is an incompressible fluid with a small amount of heat exchange, and energy conservation can be ignored. Therefore, the complex three-dimensional viscous incompressible flow in a turbine can be described by the following two equations:

**Continuity equation:**

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

(1)

**Momentum conservation equation:**

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + S_{ui}, \quad \tau_{ij} = -\rho u_i u_j
\]

(2)

Where: \(S_{ui}\) is the additional source term; \(\tau_{ij}\) is the Reynolds stress term; \(u_i\) is the fluid velocity component; \(\rho\) is the fluid density; \(p\) is the pressure. The momentum conservation equation is also known as the Navier-Stokes (N-S) equation.

In this study, the standard \(k-\epsilon\) turbulence model is used to simulate the flow characteristics in the computational domain. The boundary settings are as follows: the inlet is given a given mass flow rate, and the flow direction is perpendicular to the spiral case inlet; the relative static pressure of the outlet is 0; the solid wall adopts the non-slip wall condition; the data transfer of the interface between the rotating and static domains is carried out by the method of "frozen rotor".

### 4. Result analysis

**4.1 Analysis of flow characteristics in water diversion components**

Figure 4 shows the velocity moment distribution of the middle section of the spiral case. At the large flow rate condition, the velocity moment gradient in the spiral case increases rapidly, especially in the non-snail-shaped part. In the non-snail-shaped of the spiral case, the water near the nose of the spiral case flows into the guide vane radially, and the velocity moment is the smallest. From the nose to the inlet of the snail-shaped of the spiral case, affected by the diversion effect of the buttress, the
water flow in the circumferential direction is gradually obvious, and the velocity moment increases accordingly.

Figure 5 shows the velocity vector distribution in the spiral case at different operating conditions. Since the spiral case has a "F"-shaped section, the water flow into the guide vane area comes from three directions: the downward sloping direction of the top of the spiral case, the front of the seat ring and the upward sloping direction of the bottom of the spiral case. When the water flow enters the guide vane, the flow is redistributed at the influence of the flow parts geometry. The water at the bottom of the spiral case is squeezed into the bottom of the seat ring and forms an obvious high-speed zone, making an uneven flow distribution in the height direction. The uneven flow along the height direction at the outlet of the spiral case propagates downstream, and affects the uniformity of the flow distribution in the downstream flow components.

4.2 Analysis of flow characteristics in the guide vane region

Figure 6 shows the flow state in the guide vane region. At large flow rate condition, obvious cylindrical vortices appear in the guide vane channel at the large wrap angle range. The vortices start from the top cover of the turbine, and develop vertically downward in a spiral rotation, which makes the flow state in the guide vane region sharply deteriorated.

Figure 7 shows the flow state in the triangle clearance of the guide vane. The most of the water flow entering the guide vane area flows around the guide vane and directly enters the runner, a small part of the water flow pass through the lower end gap of the guide vane , flow around the guide vane journal and flow into the triangle clearance as a vortex , then flows into the runner. The mainstream and the vortex in the triangle clearance dope with each other. This is especially obvious at large flow rate conditions.

Figure 8 shows the water torque distribution of the guide vanes at different positions. It can be seen that the hydraulic moment on the guide vane is different at different operating condition. At the large flow rate condition, the uneven of the hydraulic moment distribution along the circumferential direction is more obvious, the resulting non-synchronous adjustment of the movable guide vanes will reduce the flow sharing effect of the movable guide vanes, and lead to the non-uniformity of the flow in the runner, which will affect the efficient and stable operation of the unit.
4.3 Analysis of flow characteristics in the runner

Cavitation is an important factor that limits the operation of Kaplan turbines at large flow rate conditions. The area with lower pressure is more prone to cavitation. Figure 9 shows the easy cavitation position in the runner in the form of isobaric surfaces. At the Optimal operating condition, No cavitation occurs in the turbine; at large flowrate condition, there is a large easy cavitation region in the runner, and the size of easy cavitation region on the blade varies due to the position of the blade. The cavitation inhomogeneity not only causes periodic changes in the size of the cavitation region during the rotation of the blade, but also causes the unevenness of the water flow parameters, which in turn affects the stability of the turbine.

For Kaplan turbine with semi-spiral case, as the flow rate increases, the axial symmetry of the flow at the outlet of the spiral case becomes worse. This uneven outflow still exists until the exit of the guide vane, and is transmitted to the runner, so that the output of the blades at different positions changes. Since the rotation speed of the turbine remains unchanged during operation, the output calculation formula $P=M\omega$ of the turbine shows that the torque of the blades at different positions can be used to characterize the output. Figure 10 shows the blade torque distribution at different positions at different operating conditions.

It can be seen from figure 10 that the torque distribution of the blades at different positions is relatively uniform due to the uniform flow from the runner at optimal operating condition, and the maximum deviation is within 0.5%. The torque deviation can reach 2.7% at large flowrate condition. The largest torque is the No. 3 blade located in the snail-shaped part of the spiral case, and the smallest torque is the No. 1 blade located in the non-snail-shaped part of the spiral case. During the turbine running, the blades will experience torque fluctuations from position 1 to position 5, as shown in figure 11. The uneven torque distribution along the circumference will have a certain impact on the stability of high-speed rotating water turbines. Due to the high speed of the hydraulic turbine, the
cantilevered Kaplan turbine blades will be subjected to high frequency and large dynamic stress at large flow rate conditions, which seriously threatens the strength and safety of the blades.

**Figure 10.** Blade torque at different position.  **Figure 11.** Variation of hydraulic torque in runner

### 4.4 Analysis of flow characteristics in the draft tube

Figure 12 shows the velocity moment and pressure distribution at the draft tube inlet at different operating conditions. At the large flowrate condition, the uneven hydraulic elements in the circumferential direction caused by the water diversion component can still continue to the draft tube. The unevenness of the velocity moment and pressure distribution increases, and the axial symmetry becomes worse. The uneven flow would cause complicated flow pattern in the draft tube.

Figure 13 shows the velocity vector distribution of the middle section of the draft tube at different operating conditions. At optimal operating condition the Backflow in the draft tube is mainly concentrated on the wall of the elbow section and the exit of the draft tube; at the large flowrate condition, the Backflow is mainly distributed in the middle of the straight cone section. It can be seen that at small flow conditions, the backflow in the draft tube is mainly caused by the change of the flow passage, while at large flow rate conditions it is mainly caused by the uneven distribution of hydraulic elements of the flow at the runner outlet.

**Figure 12.** Flow state distribution at the inlet of draft tube

**Figure 13.** Velocity vector distributions on the cross section of the draft tube
5. Conclusion

Through numerical research and Comparative performance analysis on the optimal operating condition and large flow rate condition of the Kaplan turbines with semi-spiral case, the main conclusions drawn are as follows:

The study shows that: when the Kaplan turbine is running at large flow rate condition, the flow state along the circumference and the height at the outlet of the spiral case with the “Γ”-shaped section is obvious uneven due to the discharge ratio decrease in the non-snail-shaped of the spiral case. The non-uniformity causes the vortex in the guide vane region at the snail-shaped part of the spiral case, whose size and length vary with the flow rate; it also causes large fluctuations in the blade torque. The uneven flow field distribution will continue to the draft tube, which will make the flow in the draft tube more complicated, and unstable back-flow and vortex will appear.

Acknowledgments

This research was funded by the National Key R&D Program of China (2018YFB1501900), the National Natural Science Foundation of China (52009105) and the National Natural Science Foundation of China (51879216).

References

[1] Abbas, Ahmad I.; Qandil, Mohammad D.; Al-Haddad, M. Investigation of Horizontal Micro Kaplan Hydro Turbine Performance Using Multi-Disciplinary Design Optimization, Proceeding of 13th ASME International Conference on Energy sustainability, 2019.

[2] Ardalan, J.; Håkan, N. Detailed numerical investigation of a Kaplan turbine with rotor-stator interaction using turbulence-resolving simulations, International Journal of Heat and Fluid Flow, vol.63, pp. 1-13, 2017.

[3] Wensheng Ma, Lingjiu Zhou. The influence of grid on the calculation results of hydraulic turbine flow[J]. Journal of Hydroelectric Engineering, 2006, 1: 72-75.

[4] Alok Mishra, R.P. Saini, M.K. Singhal. CFD Based Performance Analysis of Kaplan Turbine for Micro Hydro Power[C]. International Conference on Mechanical and Industrial Engineering, 2012, Singapore.

[5] Zoran Markov, Predrag Popovski, Andrej Lipej, Vesko Djelic. On the influence of the Kaplan turbine runner blade thickness on its stress parameters[C]. International Conference HYDRO. 2008, Ljubljana.

[6] Daoli Zhao, Wei Ma, Wuke Liamg, Weili Liao, TianhuWan. On 3D Flow Numerical Simulation of Bulb Tubular Turbine Considering and not Flange Clearance[J]. Large Electric Machine and Hydraulic Turbine, 2007, 3: 31-35.

[7] Fengbin Zhao. Shao Li. Research on Cavitation Performance and Stability of Turbine Capacity-increasing Modification of Gezhouba Power Station[J]. Heilongjiang Hydraulic Science and Technology, 2005, 33 (4): 14-15.

[8] Kenji SHINGAI, Kei KATAYAMA, Katsumasa SHIMMEI. Optimization of the Axial Turbine Runner Blade Using a Simulated Annealing Algorithm[C]. 22nd IAHR Symposium on Hydraulic Machinery and Systems, 2006, Yokohama.

[9] Linghua Wang, Weijuan Hu. Hydraulic Optimization Calculation of Flow Field of Guide Vane for Low-head Hydropower Station[J]. Power Engineering, 2012, 30(7): 150-152.