Analysis of hydraulic performance for Kaplan turbine components based on CFD simulation

Yunzhe Li¹, Qilin Liu¹

¹. Zhejiang Fine Institute of Hydraulic Machinery, Hangzhou, 311121
li_yunzhe@zhefu.cn

Abstract: Kaplan turbine is a kind of reaction turbine which is widely used in medium and low head section in China. Generally, flow passage of Kaplan turbine is composed of five components: semi-spiral case, stay vane, guide vane, runner and draft tube. The hydraulic performance of these components has an important impact on turbine efficiency. In this paper, 3D internal flow of about 15 Kaplan turbines is simulated by CFD method, the hydraulic performance of each component is predicted, and the calculation results are statistically analyzed. It can provide reference and direction for the designers involved in the design and research field of Kaplan turbine.

1. Introduction
Kaplan turbine is widely used in medium and low water head section hydropower stations because of its advantages such as large flow rate and rotatable blades. In the low head range, compared with the Francis turbine, Kaplan turbine can adopt smaller diameter and higher speed, which can reduce the investment of hydropower station. With the world energy shortage and the concept of energy saving and emission reduction put forward, improving the performance of Kaplan turbine has more important economic significance.

The core work of turbine hydraulic development is to reduce hydraulic loss and improve the efficiency by optimizing the geometric parameters of flow components. Therefore, carrying out the calculation and statistical analysis of hydraulic loss of each component is helpful for designers to quickly analyze and judge the problems that hinder the improvement of the turbine performance, so as to carry out targeted optimization design and shorten design and development cycle. Development of optimization design theory and CFD technology of hydraulic turbine provides technical support for the development of turbine hydraulic performance, and the development of numerical simulation analysis method has been more mature.

In this paper, a series of excellent Kaplan turbine models are selected, and numerical calculation within whole flow passage is carried out by CFD simulation. The internal flow and hydraulic loss characteristics of each components under the optimum operating conditions are obtained. The relationship between hydraulic loss values of each component and the optimum unit flow is obtained by statistics. The statistical results can provide some reference and direction for the design and research of Kaplan turbine.

2. Numerical simulation for Kaplan turbine

2.1. Model Kaplan turbine of CFD simulation
Kaplan turbines are mostly used in low head hydropower projects. In addition to a small part of Kaplan
units with head higher than 40m using metal spiral case as diversion component, the flow passage forms of conventional Kaplan turbine are relatively similar, as shown in Figure 1. In this paper, about 15 Kaplan turbines with maximum head in the range of 10 m to 50 m are modeled and simulated, and statistical curves of hydraulic performance of each component are obtained. The object of numerical analysis is model turbine, and model runner diameter of each turbine is 0.35m, and calculated head is 20m.

Figure 1. Model Kaplan turbine domain of CFD simulation

2.2. Grids of CFD simulation
In order to accurately simulate the internal flow of each components of the turbine and minimize the error in the numerical calculation, the structural grid is used to discretize each components, as shown in Figure 2. In order to reduce the influence of grid size on the calculation results, the total number of grid nodes in all the models is controlled over 6 million, and the extension structure of grid and the thickness of boundary layers consistent with each other.

Figure 2. Structured grids of each components in Kaplan turbine

2.3. Boundary condition of CFD simulation
In order to simulate the flow inside the turbine more accurately and capture the flow separation inside the turbine, the SST k-ω turbulence model is selected to close the 3D N-S equations to carry out the numerical calculation of the flow inside the turbine.

In the CFD calculation, the boundary conditions are described as follows. The mass flow condition is given at the inlet of the spiral case, the discharge is converted according to the unit discharge of operation condition. The static pressure condition is given at the outlet of draft tube, the pressure is converted according to plant Thoma number. In addition, it is assumed that the solid wall is smooth and free of sliding, and the coupling between the rotating domain and the stationary domain adopts the frozen rotor method. The calculated fluid medium is water at 25 degrees Celsius.

3. Comparison and analysis of CFD simulation results
In the optimal condition, the CFD analysis results of the flow inside the turbine are shown in Figure 3. The internal flow of the turbine is smooth, and the flow out of the spiral case is even in the circumferential direction. The stay vane and guide vane can realize no impact at the inlet. The internal flow of the runner is smooth, the internal flow of the draft tube is good, and the streamline is smooth.
3.1. Analysis of hydraulic performance of turbine components

According to the numerical simulation results, the relationship between hydraulic loss of each components and optimum unit discharge is calculated as shown in Figure 4 and Figure 5.

Figure 4 (a) shows the statistical curve of relative hydraulic loss of spiral case, and Figure 4 (b) shows the statistical curve of relative hydraulic loss of guide vane. It can be seen that the relative hydraulic losses of spiral case and guide vane account for a small proportion of the total hydraulic losses of turbine. Relative hydraulic loss of spiral case is only about 0.1% to 0.3%. In particular, one of the statistical items in this paper is an Kaplan turbine which uses metal spiral case as the diversion component. Although its discharge is smaller than others, the hydraulic loss is the largest. The relative hydraulic loss of guide vane is about 1%, but with the increase of discharge, the increase of the hydraulic loss in guide vane is relatively small. With the increase of discharge, the opening of guide vane will also increase, which makes the velocity inside guide vane not change greatly, and the hydraulic loss basically remains unchanged.

For spiral case and guide vane in Kaplan turbine, relative hydraulic loss is very small, and in actual project, external dimension of spiral case is often limited by layout of power station and can't be adjusted. Therefore, in process of turbine optimization, reducing the hydraulic loss of these two components is not the first goal of optimization.

Elbow draft tube is widely used in Kaplan turbine. The hydraulic friction loss and diffusion flow loss are the main forms of hydraulic loss in draft tube. Figure 5 (a) shows the statistical curve of relative hydraulic loss of draft tube. The analysis shows that the relative hydraulic loss in the draft tube is about 1.0 % to 2.0 %. For the unit with small discharge, the hydraulic loss of draft tube is basically equal to that of guide vane. However, with the increase of discharge, the hydraulic loss of draft tube
increases rapidly, and becomes the largest one among all flow components. So optimizing and reducing the hydraulic loss of draft tube is one of the important work of turbine hydraulic development.

On the other hand, according to the statistical results, the relationship between the relative draft tube height and the relative draft tube hydraulic loss is shown in Figure 5 (b). The relative draft tube height here is defined as the ratio of the draft tube height to the runner diameter. It can be seen from the analysis that the relative draft tube hydraulic loss decreases with the increase of draft tube height, which indicates that increasing the draft tube height is beneficial to reducing hydraulic loss in draft tube and improving turbine efficiency. Relative draft tube height of Kaplan unit calculated in this paper is mostly about 2.7, and these units can achieve better hydraulic performance. However, there are two Kaplan units that its relative draft tube height is only about 2.1. Optimum unit discharge of these two units is about 0.9 m³/s, and it can be found that this value is much smaller than that of other units. But the hydraulic loss in draft tube of these two units are similar to others. This shows that relative draft tube height will seriously affect the application of turbine to large discharge condition.

![Graph](image-url)

**Figure 5. Statistical curve of relative hydraulic loss inside draft tube**

### 3.2. Analysis of hydraulic performance of runner

In this paper, the distribution trend of hydraulic efficiency of runner is also counted as shown in Figure 6. The performance of the runner is judged by hydraulic efficiency of runner. Since the hydraulic loss of other components is not included, it only reflects the performance of the runner itself, and is more suitable for separate comparison of runner performance. The specific formula for calculating hydraulic efficiency of runner is as follows.

At present, the number of runner blades of Kaplan unit is four, five and six. Because of the low head and large discharge, the unit with four-blade runner has the same application range as the bulb turbine, so it has less application in actual project. Five-blade runner and six-blade runner are mostly used in newly-built Kaplan unit. From Figure 6, it can be seen that the optimum unit discharge of four-blade runner is about 1.6 m³/s and that of five-blade runner is about 1.3 m³/s. With the further decrease of the optimum unit discharge, the hydraulic efficiency of six-blade runner will be higher than that of five-blade runner. It is also found that the hydraulic efficiency of runners with different blade numbers can reach above 96.8 % even though the optimum unit discharge is different. Therefore, the relationship between the optimum unit discharge and the number of runner blades can be determined, which provides basis for the selection of turbine parameters.
Figure 6. Statistical curve of hydraulic efficiency for Kaplan turbine runner

Figure 7 shows the relationship between hydraulic efficiency of runner and optimum unit speed. It can be found that the unit speed range of 6-blade runner is 115 r/min to 125 r/min. The main application range of 5-blade runner is about 130 r/min, while 4-blade runner is used at higher speed, about 140 r/min.

4. Conclusions

In this paper, the relationship between the hydraulic loss of each flow component of Kaplan turbine and the optimal unit discharge is described, and the following conclusions are obtained.

- The hydraulic loss in spiral case is smaller than that in guide vane and the draft tube.
- When the discharge is small, hydraulic losses in guide vane and draft tube are equivalent. However, with the increase of discharge, hydraulic loss in draft tube increases obviously. It becomes one of the most important factors for hydraulic optimal design.
- The difference of the blade numbers directly determines the optimal unit discharge, so the selection of turbine design should be based on the design requirements of the unit.

References

[1] Zhengwei Wang, Lingjiu Zhou, Yanguang Cheng, Ming Ding, Guodong Cheng. Hydraulic loss analysis in bulb turbine [J]. Large Electric Machine and Hydraulic Turbine, 2004, (5) : 40-43.

[2] Shangfeng Wu, Guoren He, Yulin Wu, Shuhong Liu. Turbine flow simulation and performances prediction on the Kaplan turbine model with open casing [J]. Journal of engineering thermophysics, 2004, 25(6) : 959-961.

[3] Pengcheng Guo, Xingqi Luo, Wuke Liang, Xiaobo Zheng. Numerical simulation of 3D turbulent flow through Kaplan turbine based on mixing plane approach [J]. Journal of hydrodynamics, 2005, 20(2) : 161-166.

[4] Fengqin Han, Ying Yu, Jiasheng Wu, KUBOTA Takashi. Dynamic flow interference between runner outlet and draft tube inlet of francis turbine [J]. Journal of engineering Thermophysics,
2009, 30(1) : 65-68.

[5] Song Zhu, Zhenyu Wang, Genhai Mao, Guohua Liu. Study on flow field in an elbow draft tube by 3D numerical simulation. Journal of hydroelectric engineering, 2006, 25(1) : 76-80.

[6] DENTON J D. The calculation of three-dimensional viscous flow through multistage turbomachines [J]. ASME J. Turbomachinery, 1995, 114(1) : 8-17.