Improvement on Anti-cavitation Performance for a 120 MW Kaplan Turbine

Yunzhe Li¹, Qilin Liu¹

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

Abstract: The model test results show that the initial runner developed for LB HPP has the problem of clearance cavitation, which does not meet the design requirements of no cavitation in the whole operating range. In order to find out the reasons of clearance cavitation, and put forward the corresponding optimization method, the numerical simulation method is used to calculate the clearance flow inside runner on the basis of the entire passage calculation. By analyzing the pressure distribution on blade surface and the flow characteristics inside runner blade clearance, the reasons of clearance cavitation have been found, the initial runner blade has been optimized, and the secondary model test has been carried out. The results of the secondary model test show that the optimized runner blade can meet the design requirements.

1. Introduction
The prototype turbine in the LB HPP is a Kaplan turbine, and it has a diameter of 7.0 m, rated speed of 107.1 r/min, rated power of 120 MW, maximum head of 36 m, rated head of 34.4 m and minimum head of 26.5 m. The turbine model test was conducted and the results show that energy characteristics and pressure fluctuation characteristics can satisfy design requirements. However, when the model turbine operates under the rated output and maximum head condition, the incipient Thoma number (σ_i = 0.572) tested by model test is larger than the plant Thoma number (σ_p = 0.519), which indicates that the cavitation performance can't satisfy design requirements of no cavitation in whole operating range. With the help of stroboscope observation, it is found that the bubble mainly occurs near the front side of clearance at tip side and the hub side of the blade, and it is confirmed that the cavitation generated here belongs to clearance cavitation, as shown in Figure 1. It is necessary to simulate the runner blade clearance flow to determine the reasons of clearance cavitation and to propose an optimization scheme.

Cavitation is a common phenomenon in turbine, which will affect turbine's performance seriously. In Kaplan turbine, airfoil cavitation and clearance cavitation are the main factors. Now, the research of flow and cavitation inside the runner clearance by numerical simulation technology has been paid more and more attention. For example, Xiuli Han² analyzed the flow inside runner tip of Kaplan turbine, and compared the efficiency calculated by CFD with the test results. Weili Liao³ simulated the flow on the outer edge side of runner blade under the condition of negative impact angle, and focused on the secondary flow and cavitation characteristics caused by the flow inside the tip. All these studies show that the numerical simulation method is accurate and reliable for the analysis and calculation of clearance cavitation in the Kaplan runner.

In this paper, 3D clearance flow in model runner is analyzed and calculated by numerical simulation. By analyzing the influence of clearance flow on the pressure distribution of the blade surface, the cause of clearance cavitation is found, and then the optimal design of the initial runner
blade is carried out, and the experimental verification is carried out.

![Model Kaplan turbine, Cavitation at tip side, Cavitation at hub side](image)

**Figure 1. Cavitation test results under rated output and maximum head condition (with Thoma number is 0.519)**

2. **Geometric and numerical methods**

In this paper, the whole flow passage from spiral case inlet to draft tube outlet of model Kaplan turbine is taken as CFD simulation domain. The model runner diameter is 0.35 m, and calculation head is 20 m. The whole calculation domain is shown in Figure 2. In the calculation, all flow components are dispersed by structured grid.

![Model Kaplan turbine domain of JC HPP](image)

**Figure 2. Model Kaplan turbine domain of JC HPP**

In order to accurately simulate the reasons of clearance cavitation by CFD simulation and find out solution method, clearance around the runner blade is comprehensively simulated, and its geometry and grid are shown in Figure 3. Minimum width of the clearance at tip side is 0.15 mm, which is consistent with actual size of the model machine. Maximum width of the clearance at tip side is 2.36 mm, and maximum width of the clearance at hub side is 2.06 mm.

![Geometric of runner blade, Grid at hub side, Grid at tip side](image)

**Figure 3. Geometric of runner blade and grids at the tip**

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 CFD calculation, the boundary conditions are described as follows. The discharge condition is given at inlet of spiral case, the discharge is converted according to the unit discharge of operation condition. The static pressure condition is given at outlet of draft tube, the pressure is converted
according to the 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. CFD results and blade optimization

When pressure inside turbine is lower than vapour pressure, cavitation will occur\(^{4-5}\). Therefore, analysis of pressure distribution inside turbine can help and guide prediction and optimization of turbine cavitation. The head change range is small enough, so it is possible that cavitation can be avoided in the whole operation range by blade optimization.

When the model turbine operates under the rated output and maximum head condition, the pressure distribution on the runner blade is shown in Figure 4. The results show that there are two areas on the suction side of runner blade where the pressure is obviously lower than vapour pressure, and this two areas in the figure are shown in dark blue. They are located in the front of clearance at tip side and hub side of runner blade. Compared with the cavitation test results, it can be seen that these two low-pressure areas are the main reason for clearance cavitation under this condition.

![Flow distribution on suction side of initial blade under the rated output and maximum head condition](image)

Based on the above analysis, optimal design of clearance cavitation should start with optimizing and improving the blade pressure distribution, and reduce blade load near the clearance to reduce the velocity inside the clearance\(^{4-7}\). According to the structural characteristics of runner blade clearance, the following two methods are used for blade optimization. First, the clearance width at tip side of blade is larger at the leading edge and tailing edge of blade, but smaller in the middle of blade. And clearance cavitation at tip side mainly occurs at the front of blade. Therefore, the load on the tip side of blade should be reduced, especially the load at the front. Second, in the hub side, the improvement of clearance cavitation should be based on the reduction of the load at front of blade. Third, the load distribution on other parts of blade should be adjusted reasonably to ensure that turbine performance remains unchanged. The change of blade shape with optimization is shown in Figure 5. In can be found that deflection of airfoil at tip side and hub side of blade is reduced, especially at the front of tip side airfoil. Through CFD calculation, it can be found that performance of the optimized turbine is slightly better than that of the initial turbine. Comparison of efficiency is shown in Table 1.
Figure 5. Difference of runner blade before and after optimization

Figure 6 shows the pressure and velocity distribution changes near the blade surface and tip after optimization under the rated output and maximum head condition with plant Thoma number. It can be seen that with the blade load charges, the two low pressure area disappear. So it means that the method is useful for clearance cavitation under the rated output and maximum head condition.

Figure 6. Pressure distribution on suction side of optimized blade under the rated output and maximum head condition (with Thoma number is 0.519)

With blade optimization, the pressure distributions at the tip side and hub side of blade are compared as shown in Figure 7. Compared with initial blade, the load at the tip side of the optimized blade decreases slightly, and the load on the middle of the blade decreases greatly. On the hub side, the load on the front of blade decreases significantly, mainly due to a significant increase in the minimum pressure on the suction surface and much higher than vapour pressure. The pressure on the blade increases about 30 kPa, which is equivalent to a possible decrease in incipient Thoma number of about 0.15. This means that the cavitation problem under this condition has been eliminated.

Figure 7. Pressure changes on the runner blade before and after optimization

In order to ensure that the optimized blade will not cavitate under the minimum output and minimum head condition due to the charge of blade shape, internal pressure under this condition are
simulated by CFD simulation. The pressure distribution on blade are shown in Figure 8, and it shows that no cavitation will occur.

![Figure 8](image_url) Pressure distribution on suction side of optimized blade under minimum output and minimum head condition (with Thoma number is 0.519)

The comparison of energy characteristics of model turbine before and after blade optimization is shown in Table 1. The test results show that the hydraulic efficiency of the new blade is slightly better than the initial model. This shows that the blade optimization method described in this paper can improve the tip cavitation inside the runner blade and has no adverse effect on the hydraulic efficiency of the turbine.

| Condition Name     | CFD calculation efficiency | Model test efficiency |
|--------------------|----------------------------|-----------------------|
|                    | Initial blade | Optimized blade | Initial blade | Optimized blade |
| Optimal condition  | 93.65%        | 93.73%            | 93.41%        | 93.53%          |
| Rated condition    | 93.37%        | 93.46%            | 93.25%        | 93.31%          |

After blade optimization, the secondary model test was carried out. The cavitation test results of the optimized runner blade is shown in Figure 9. Under rated output and maximum head condition, the incipient Thoma number decreased from 0.572 to 0.508, which is slightly different from the CFD results. And under minimum output and minimum head condition which is shown in Figure 9 (b), no cavitation occur. So the optimized blade can satisfy the requirement of cavitation free in whole operating range.

![Figure 9](image_url) Secondary Cavitation test results of optimized blade

4. Conclusions
In this paper, the numerical simulation technology is applied to simulate the internal flow of Kaplan turbine, and clearance cavitation around the runner blade are analyzed. A blade optimization method which can be used to improve the blade clearance cavitation is described, and the optimization results are verified by model test. Model test results show that this method is effective.
References

[1] Mikael Grekula, Goran Bark. Cavitation Observations and Noise Measurements in a 9.3 MW Kaplan turbine [A]. Proceedings of the Hydraulic Machinery and Systems 21th IAHR Symposium [C]. Sept.12-16, 2002, Lausanne, pp.: 491-498.

[2] Xiuli Han, Wangjiang Liu, Lefu Zhang. Analysis of Clearance flow in Kaplan turbine [J]. Large Electric Machine and Hydraulic Turbine, 2009, (3): 44-47.

[3] Weili Liao, Kun Tan, Xingqi Luo, Peng Lu. Study on mechanism of cavitation erosion on blade of Kaplan turbine [J]. Water Resources and Hydropower Engineering, 2006, 37(5): 89-92.

[4] Guoyu Wang, Inanc Senocak, Wei Shyy, et al. Dynamics of attached turbulent cavitating flows [J]. Progress in Aerospace Sciences, 2001, 37.

[5] Inanc Senocak, Wei Shyy. A pressure-based method for turbulent cavitating flow computations [J]. Journal of Computational Physics, 2002, (176): 363-383.

[6] Takaya KITAHORA, Ryouji SUZUKI, Tomitarou TOYOKURA, Takashi KUBOTA, Junichi KUROKAWA. Study on blade tip vortex of Kaplan turbine. Reported at XVII IAHR SYMPOSIUM Beijing, China, 1994.

[7] Weili Liao, Shengzhu Liu, Lefu Zhang. Experimental investigation of the tip clearance cavitations on blades flange of Kaplan turbine [J]. Journal of hydroelectric engineering, 2005, 24(4): 67-72.