Numerical simulation of the performance of a low-head prototype Kaplan turbine

F N Chen¹, H M Lei¹, S H Ahn¹, H G Fan¹, X P Jiang², J W Hu², J Zhang¹, Y Y Luo¹ and Z W Wang¹

¹State Key Laboratory of Hydroscience and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China
²School of Mechanical Electronic & Information Engineering, China University of Mining & Technology (Beijing), Beijing 100083, China

E-mail: wzw@mail.tsinghua.edu.cn

Abstract. The opening degrees of guide vane and runner blade are adjustable for Kaplan units. Therefore, the prototype unit performance is evaluated by cam curves to ensure the maximum efficiency points at different runner blade or guide vane angles. In this paper, the numerical simulation method is used to obtain the cohesive curves. The gravity force term is not ignored in the calculation, of which the effects are analysed not only on prototype performances, but also on internal flow characteristics such as the streamlines around runner blades, the pressure distributions and the head losses of each part.

1. Introduction
For Kaplan turbine, the opening of runner blades and guide vanes can be adjusted. Thus, the Kaplan turbine can achieve high efficiency at both partial loads and overloads [¹]. Therefore, it’s important to find the relationship between the runner blade opening and guide vane opening, which can be obtained from cam curves. Currently, the cam curve used in hydropower station is based on model test. Model test can help obtain the cam curve of the model turbine, and then it can be used in hydropower station after similar conversion. However, there is difference between that and the cam curve of Kaplan turbine prototype. Test can also be performed on the Kaplan turbine prototype, and thus the cam curve can be obtained directly. However, it’s difficult to test enough runner blade openings and guide vane openings, so the cam curve can’t be gained accurately. Through the numerical simulation method, it’s possible to predict the cam curves of the turbine prototype precisely. Many researchers have studied the cam curves from the perspective of airfoils and design [²-⁵].

The internal flow characteristics of the Kaplan turbine can be changed when the opening of runner blades or guide vane blades changes. And thus the differences in internal flow characteristics lead to differences in efficiency and power. Liu et al. [¹] simulated the unsteady turbulent flow throughout the whole flow passage of a prototype Kaplan turbine and discussed the mechanisms of generation and transmission of the main pressure fluctuation in the prototype turbine, and they also compared the pressure fluctuation on-cam and off-cam operation conditions of the prototype turbine. R. Maddahian et al. [⁶] investigated the rotating vortex rope in the draft tube of a Kaplan turbine at part load condition. They also found that the tip vortex was created due to the runner and shroud.

Froude number is an indicator which reflects the effect of gravity acceleration on the flow field. Gravity can cause vertical pressure gradients in the flow field, and affect the cavitation performance of
the axial-type hydraulic turbines \cite{7}. For low-head Kaplan turbines, due to the low head, the velocity of the fluid is quite small. If the runner diameter is large, the Froude number will be low. Then gravity will have an impact on the Kaplan turbines which can’t be ignored \cite{8-10}. And the cam curve of the turbine will also be influenced.

In this paper, the numerical simulation method is used to obtain the cam curves. And the gravity force term is not ignored in the calculation. In order to make it clear how the gravity force influences the cam curves, the simulation without gravity taken into account was also performed. The two results were compared in this paper. The effect of gravity was analyzed not only on prototype performances, but also on internal flow characteristics such as the streamlines around runner blades, the pressure distributions and the head losses of each part. In addition, the possibility of cavitation is discussed by analyzing the low pressure zones on runner blades.

2. Kaplan turbine prototype
In this paper, the Kaplan turbine prototype is studied here. It consists of casing, draft tube, 4 runner blades, 24 guide vanes, 12 stay vanes and 1 special stay vane, as shown in figure 1. And they will be called CAS, DT, RV, GV and SV as shown in figure 1. The runner diameter is 3.3 m designed by the rated head of 4.7 m, and the rated rotational speed is 125 rpm. The runner blades and the guide vanes can be adjusted in a wide range.

![Figure 1. The flow passage of the Kaplan turbine prototype](image)

3. Numerical method
3.1. Governing equations and turbulence model
The incompressible continuity equation and Navier-Stokes equations are used as the governing equations. A general time averaging filter is used to solve the turbulence problems and the equations are changed as follows:

\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)
\]

\[
\rho \left( \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \rho \frac{\partial}{\partial x_j} (u'_i u'_j) - f_i \quad (2)
\]

where \( \bar{u}_i \) is the time-averaged velocity, \( u'_i \) is the velocity fluctuation, \( \bar{p} \) is the time-averaged pressure, \( \rho \) is the fluid density, \( \mu \) is the viscosity, and \( f_i \) is the body force term, which is defined as \( \rho g \) when gravity acceleration is considered in simulation.

The SST k – \( \omega \) double-equation-turbulence is applied to close the equations above.

3.2. Numerical model and boundary conditions
The flow path is shown in figure 2. It consists of 5 computational domains: casing, stay vanes, guide vanes, runner, and draft tube. The runner domain is discretized with high quality structured mesh. Other domains are discretized with unstructured mesh which are generated based on the grid density in the runner. At the interface between two domains, the grid density in different domain is almost the same. The number of the grid elements in all computational domains is 7,333,717, and the number of the nodes is 2,272,261.

![Calculation grids](image)

**Figure 2.** Calculation grids (upper: total turbine; lower: runner)

In this paper, 2 numerical methods are used to get the cam curves of the Kaplan turbine prototype. These 2 methods are as follows:

- Method 1): steady simulation without the gravity force term.
- Method 2): steady simulation with the gravity force term.

Method 1 is a typical simulation method often used for high-head turbines [11-13]. For this method, the body force term in equation (2) is set to zero. Therefore, the gravity force term is ignored. The boundary conditions are as follows:

The total pressure is set at inlet of the casing, and the static pressure is set at the outlet of draft tube. Here, the velocity at the inlet is assumed uniform and normal to the cross section of the inlet. At all walls, the no-slip condition is set.

For Method 2, the body force term in equation (2) is set as $\rho g$ (for water, $g = 9.81 \text{m/s}^2$, $\rho = 997 \text{kg/m}^3$). Then the gravity acceleration is considered. Therefore the total pressure set at inlet is not constant, but variable. It’s a function of height. The static pressure at outlet is also a function of height. At all walls, the no-slip condition is set.

4. **Numerical results**

4.1. *Comparison of cam curves obtained through different methods*
The cam curves indicate that when runner blade opening and guide vane opening meet a certain relationship, the turbine can work at high hydraulic efficiency. In this paper, the hydraulic efficiency can be calculated according to the following equation:

$$\eta = \frac{T \omega}{\rho g H Q}$$  \hspace{1cm} (3)

where $T$ is the torque of the rotating part, $\omega$ is the angular velocity, $\rho$ is the density of water, $g$ is the gravity acceleration, $Q$ is the discharge, and $H$ is the head.

$Q$ is expressed as follows:

$$Q = \int_A v_n dA$$  \hspace{1cm} (4)

where $v_n$ is the velocity normal to the boundaries at the inlet, $A$ is the area of inlet.

In Method 1, $H$ is defined as follows:

$$H = \left(\frac{p_t}{\rho g}\right)_{in} - \left(\frac{p_t}{\rho g}\right)_{out}$$  \hspace{1cm} (5)

where $p_t$ is the total pressure. The subscripts $in$ and $out$ express the inflow plane (inlet of CAS) and the outflow plane (outlet of DT).

In Method 2, $H$ is given by

$$H = \left(\frac{p_t}{\rho g}\right)_{in} - \left(\frac{p_t}{\rho g}\right)_{out} + z$$  \hspace{1cm} (6)

where $z$ is the height difference between the geometric center of inlet and outlet.

The cam curves under rated head and the relative runner blade opening of 83% is shown in figure 3. The upper curves show the relationship between efficiency and unit discharge. And the bottom curves correspond to the upper one, and shows the guide vane opening. It can be seen from figure 3 that for a certain RV opening, the efficiency of the unit increases first and then decreases as the opening of GV increasing. When the relative guide vane opening is 81.59% (Method 1) or 89.11% (Method 2), the efficiency of the unit is higher than other guide vane openings, and this case is called on-cam condition.

The differences between the cam curves obtained through Method 1 and Method 2 can be seen in figure 3. Method 1 causes larger discharge at the same GV opening than Method 2, thus the head loss is less, and the hydraulic efficiency is higher than Method 2, as shown in figure 3. At the same time, the opening of GV at on-cam condition obtained by Method 1 is 81.59%, less than that by Method 2, which is 89.11%. Therefore, if gravity is considered, the turbine will achieve the highest efficiency at larger GV opening.
Figure 3. Cam curves under the relative RV opening of 83%
For Method 2, the cam curves under other runner blade openings can be seen in figure 4. Similarly, each RV opening corresponds to an on-cam condition. A thick solid line is used to connect all the on-cam conditions, which is a complete cam curve under rated head, as shown in figure 4. It indicates that the turbine achieves the highest efficiency at the RV opening of 33%.

Figure 4. The complete cam curve gained from Method 2
Figure 5 shows the complete cam curves obtained from different methods. It indicates that Method 1 can cause higher efficiency than Method 2. However, the optimal operating condition predicted by the 2 methods is almost the same. It’s predicted that when the relative openings of RV and GV are 33% and 56% respectively, the unit reaches the highest efficiency. And at the optimal operating condition, the efficiency divergence is smaller than at other conditions. But as mentioned above, there is difference in on-cam condition prediction at partial loads or overloads.

4.2. Comparison of internal flow characteristics
In this section, the difference in internal flow characteristics is analysed. Table 1 shows the relative head loss in each part of the turbine, which is defined as:

\[ \Delta H_{rel} = \frac{\Delta H}{H} \]

where \( \Delta H \) is the head loss in each part, and \( H \) is the head of the turbine. The sum of all the relative head losses and efficiency is 1. It can be seen that the head loss in RV is higher than in other parts. And Method 1 causes less head loss in RV than Method 2, which result in higher efficiency than Method 2. The difference between the head loss in RV obtained by the 2 methods can be seen more clearly in figure 6. For Method 1, as the opening of GV decreases, the head loss in RV also decreases, thus the efficiency increases. Therefore, the on-cam condition appears at small opening of guide vanes. For Method 2, the head loss is large at small GV opening, so the efficiency is much lower than Method 1, and the on-cam condition appears at larger guide vane opening.

Table 1. Relative head loss in each part of the turbine (unit: %) (relative RV opening is 83%)

| Relative opening of GV | 96.30% | 87.04% | 79.63% |
|------------------------|--------|--------|--------|
| \( \Delta H_{rel-CAS} \) | 0.61   | 0.73   | 0.58   |
| \( \Delta H_{rel-SV} \) | 0.99   | 1.00   | 1.05   | 1.05   | 1.08   |
| \( \Delta H_{rel-GV} \) | 1.99   | 1.15   | 1.75   | 0.92   | 1.72   | 0.90   |
| \( \Delta H_{rel-RV} \) | 5.02   | 7.46   | 4.31   | 6.67   | 3.68   | 8.15   |
| \( \Delta H_{rel-DT} \) | 2.74   | 2.39   | 2.48   | 2.38   | 2.94   | 3.08   |
| Efficiency             | 88.65  | 87.27  | 89.81  | 88.24  | 90.03  | 86.07  |
5. Conclusion

In this paper, the 3D steady flow in a low-head prototype Kaplan turbine is simulated. The cam curves of the turbine under the rated head are obtained by use different simulation methods. The internal flow characteristics are also analyzed. The results indicate that the efficiency of the turbine will be lower if gravity is considered into the governor equations. And there are differences in the on-cam condition prediction between the 2 methods except the prediction of optimal operating condition. At overloads, the turbine will achieve the on-cam condition at larger guide vane opening if gravity is considered.

Acknowledgement

Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), the National Natural Science Foundation of China (No. 51479093), the National Key Research and Development Program of China (No. 2017YFC0404200), for supporting the present work.

References

[1] Yulin Wu, Shuhong Liu, Hua-Shu Dou, Shangfeng Wua and Tiejun Chen 2012 Numerical prediction and similarity study of pressure fluctuation in a prototype kapan turbine and the model turbine Computers & Fluids 56 128 – 142
[2] Houbin Song and Zhenggui Li 2010 Alteration of cognate curve and its influences on bulb tubular turbines Journal of Gansu sciences 22(4) 145-149
[3] Shijie Yan, Fengqin Han and Youqing Wang 2010 Numerical prediction of on-cam openings of adjust-ble guide vanes and runner vanes for bulb turbine Journal of Engineering Thermophysics 3 137-140
[4] Jiasheng Wu, Qingshu Ou and Fengqin Han 2009 Numerical prediction of combined operation condition of tubular turbine Water Resources and Power 27 (3) 141-143
[5] Linming Zhao, Weiliang Li and Guimian Zhang 2007 The joint relationship of kapan turbine considering hydraulic vibration characteristics Journal of Hebei University of Engineering (natural science edition) 24 (4) 74-75
[6] R Maddahian, M J Cervantes and N Sotoudeh Numerical investigation of the flow structure in a kapan draft tube at part load, IOP Conf. Ser.: Earth Environ. Sci. 49 022008
[7] Y.Z. Liu and J.S. Chang 2008 Influence of gravity on flow field analysis and hydraulic performance evaluation of bulb turbine Shuili Xuebao 39 (1) 96-102.
[8] Y. Luo, Z. Wang, X. Liu, Y. Xiao, C. Chen, H. Wang and J. Yan 2015 Numerical prediction of pressure pulsation for a low head bidirectional bulb turbine Energy 89 730-738.
[9] Soo-Hwang Ahn, Yexiang Xiao, Zhengwei Wang, Xuezhi Zhou and Yongyao Luo 2017 Numerical prediction on the effect of free surface vortex on intake flow characteristics for tidal power station Renewable Energy 101 617-628.
[10] Soo-Hwang Ahn, Yexiang Xiao, Zhengwei Wang, Xuezhi Zhou and Yongyao Luo 2017 Performance prediction of a prototype tidal power turbine by using a suitable numerical
model Renewable Energy 113 293-302.

[11] X. Yexiang, W. Zhengwei, Y. Zongguo, L. Mingan, X. Ming and L. Dingyou 2010 Numerical analysis of unsteady flow under high-head operating conditions in Francis turbine Eng. Comput. 27 (3) 365-386.

[12] Z.W. Wang and L.J. Zhou 2006 Simulations and measurements of pressure oscillations caused by vortex ropes J. Fluid Eng-T Asme 128 (4) 649-655.

[13] X. Yexiang, W. Zhengwei and Y. Zongguo 2011 Experimental and numerical analysis of blade channel vortices in a Francis turbine runner Eng. Comput. 28 (2) 154-171.