

Numerical analysis of a Pelton runner water sheet flow under super-high head

J Zhang¹, Y X Xiao^{1*}, Q R Zhou¹, Z H Gui², B Guo¹, C J Zeng¹, Y Y Luo¹, H G Fan¹ and Z W Wang¹

¹ State Key Laboratory of Hydrosience and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China;

² Technology center state grid Xinyuan company LTD., Beijing 100032, China;

xiaoyex@mail.tsinghua.edu.cn

Abstract. The hydraulic performance of Pelton turbine is different from reaction turbines due to its unsteady flow of rotating buckets with time and space. The water sheet flow in Pelton turbine is also very complex as the jet flow is often interacted with air and itself. This paper numerically investigates the dynamic hydraulic performance of a Pelton under super-high head. Unsteady numerical simulations were performed using SST $k-\omega$. Two-phase homogeneous model and VOF were adopted to model the free surface flow. The evolution history of water sheet flow in the buckets were presented. The distribution of pressure coefficient on the bucket surface were also analysed. The results indicate that the development of water sheet flow on the bucket surface can be divided into three stages, corresponding to the output power variation process. This calculation can be used to illustrate the hydraulic mechanism of water sheet flow evolution.

1. Introduction

China is one of the richest countries with water resources of high hydraulic head. The Pelton turbine is an impulse turbine that can operate under extreme high water head. It does not need to be built on a large dam, which will be very helpful for reducing the construction cost and minimizing the negative effects on the ecology^[1]. It is expected to be one of the most popular impulsive turbines for high water resources in China.

At present, the development of the Pelton turbine is not mature enough to generate a general method for turbine design^[2]. Interactions between the water and air increase the difficulty for understanding the whole flow process in the rotating Pelton bucket^[3]. The water sheet flow in the bucket does not have fixed boundaries, so it is quite complex to fully understand the mechanism of the flow^[4-5]. In recent years, CFD has become an effective tool in simulating the water sheet flow and has been proven to be very effective in predicting the turbine performance^[6-16].

In this paper, a prototype pelton turbine for super-high head of 1100m was designed. The performances of this prototype were predicted through numerical methods. The flow analysis under the design conditions were presented, which will help understand the flow mechanisms for water sheet flow in Pelton turbines.

2. Prototype Pelton turbine and numerical simulation method

2.1. Research model and mesh



Content from this work may be used under the terms of the [Creative Commons Attribution 3.0 licence](https://creativecommons.org/licenses/by/3.0/). Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Based on the design parameters, a prototype Pelton turbine was designed. This turbine has 1 nozzle and 17 buckets. The design head of the turbine is 1100m with a rated rotation speed of 428.6 rpm. The design output power is 200MW. The research model is shown in Figure 1.

In order to improve the simulation precision, the whole flow field is considered in the calculation domains, including a nozzle, a runner with 17 buckets and a stationary region. But due to symmetry of the model, only half the domains were retained in actual simulations. The three dimensional model was built by UG. Due to the complexity of the model, unstructured mesh of tetrahedral cells was generated for the model with ANSYS ICEM. As the global domain and some local areas are significantly different in size, special attention was paid for local details. The cutout and surfaces of the buckets were refined to improve the mesh quality while the stationary region maintains a relatively coarse mesh. The jet domain uses a hexahedral mesh as its shape is relatively simple. In order to ensure the independence of the mesh, cases with different element sizes are performed. After carefully comparison, the mesh chosen at last for all the domains has about 890000 nodes and 4700000 elements.

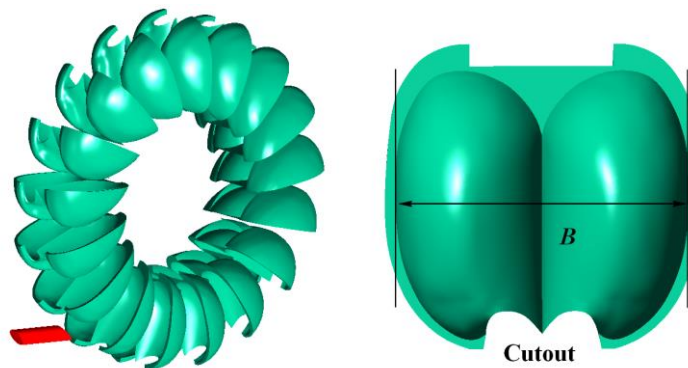


Figure 1. Sketch for the prototype and the bucket

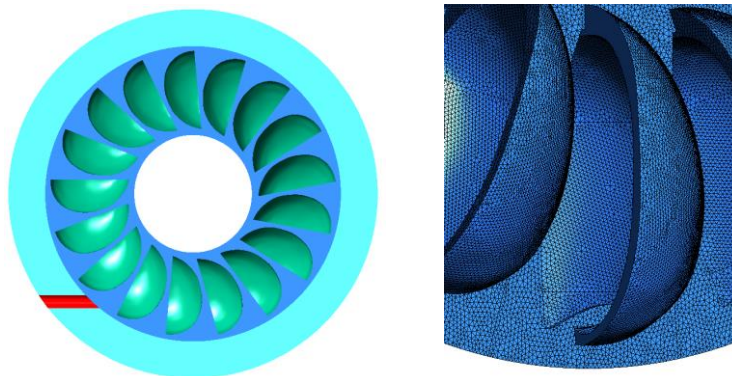


Figure 2. Computational domains and mesh details for buckets

2.2. Numerical scheme and boundary conditions

The water flow is injected under atmospheric condition with the buckets rotating. So water and air should both be considered in the simulations. Homogeneous model is adopted to describe this two phase flow, which means the air and the water share the same velocity. VOF is used to predict the interface shape between the air and water, where the volume fractions of α_a for air and α_w for water are introduced. It is assumed no mass transfer happens between the two phases, and the total fractions for water and air is 1, $\alpha_a + \alpha_w = 1$. The air is chosen for the primary phase while the water is considered as the secondary phase.

The Navier-Stokes equation is discretized by the finite volume method from ANSYS CFX. An upwind scheme was adopted for the continuity and momentum equations, while the transient terms were solved with a first-order backward Euler scheme. As the jet process is fully transient, a proper guess

should be given for the domains as the initial conditions. The whole flow domains are considered full of stationary air in the beginning, which means the volume fraction and velocity for air is 1 and 0 respectively. Given the design parameters into considerations, a velocity inlet full of water is adopted ($\alpha_w = 1$). The outlet boundary conditions are set as the “opening” on the surface of the stationary region. A time-step of 1.03×10^{-4} s was chosen to match the rotating speed, which means the rotor regain its initial position every 1360 timesteps. The simulations were calculated for 4 rotation circles. However, only data from the last rotation cycle is extracted for the results processing.

3. Results and discussions

3.1. Predicted dynamic hydraulic performance.

The bucket rotates for nearly 82 degrees from the moment it gets in touch with the water to the moment that all water is drained completely. Thus, the non-dimensional rotation degree β is defined to illustrate the rotation time, with an equation shown as below:

$$\beta = \frac{\theta}{82} \quad (1)$$

Where θ is the rotation angle for the investigated bucket from the initial position.

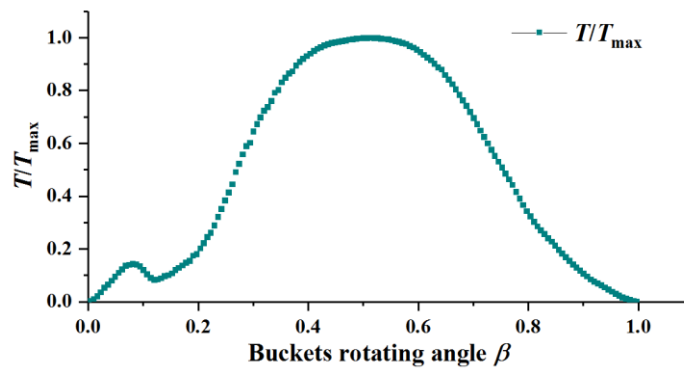


Figure 3. Predicted bucket torque with time under design conditions

The predicted hydraulic performance of the Pelton turbine which varies with time are shown in Figure 3. The maximum value for one bucket is used to process the torque variations of the single bucket. The X axis represents the non-dimensional rotation degree β , while the Y axis represents the non-dimensional relative torque T/T_{\max} for a single bucket.

From $\beta=0$ to 0.4, the bucket torque keeps increasing at rapid pace although a slight drop occurs around $\beta=0.1$. Then the torque maintains at a relatively high level from $\beta=0.4$ to $\beta=0.6$, where afterwards a sharply drop follows. The calculated output power for the turbine is 180MW, 20MW lower than the designed value, which suggests further optimization should be done to satisfy the requirements.

3.2. Developing process for water sheet flow in the bucket

In order to understand the development process of water sheet flow inside the bucket, 3 bucket was specifically studied. The sequence number was named for the 3 buckets, 1, 2, 3 from the clockwise direction. However, detailed analysis was only conducted for the bucket 2 (marked as blue). With a volume fraction of 0.5 for water, isosurfaces of the flow were generated to represent water sheet flow.

The whole jet process for this bucket was subdivided into 6 frames, as shown in Figure 4. As shown in Figure 4 (a), the water was injected to the bucket 1 first. The water is then divided by the cutout of bucket 2 ($\beta=0.1$) and move towards bucket 1 and bucket 2 simultaneously. This is the time when the impulsive force begins to increase for bucket 2. The water sheet flow keeps flowing inside the bucket while no water is able to flow out from the bucket 2 ($\beta=0.19$). As the water sheet flow keeps developing inside the bucket 2, high torque level will be reached when the flow is fully developed.

As the buckets continues to rotate, the jet flow is prevented from entering the bucket 1 completely and the water left begins to decrease. Some water from bucket 1 even flow on the back surface of bucket 2 ($\beta=0.49$), which means the interactions occur between the rear of bucket 2 and outflow from bucket 1. This may exert some negative effect on the turbine efficiency. While the water sheet is developing inside the bucket 2, the bucket 3 also begin to get in touch with the jet flow coming from the nozzle ($\beta=0.49$). At the same time, the water sheet flow covers a large area in bucket 2, though some water begins to flow out from bucket 2. It can be observed some water flows out from the cutout of the bucket, which may also reduce the impulsive force for the buckets. During this short period($\beta=0.49\sim0.74$), water keeps on flowing in and out simultaneously for bucket 2. The impulsive force maintains at a high level as the sheet flow inside the buckets 2 remains relatively stable. From the flow analysis above, it can be found that the interactions exist for the buckets, and the leakage flow also occurs near the buckets, which indicate the buckets can be further optimized to improve the efficiency.

With the further rotation of buckets, no more water is able to enter and replenish the bucket 2 ($\beta=0.79$), the water left in the bucket begins to decrease, leading to a rapid decline of the impulsive force. Buckets keeps working until all the water flows out of bucket 2 ($\beta=0.92$).

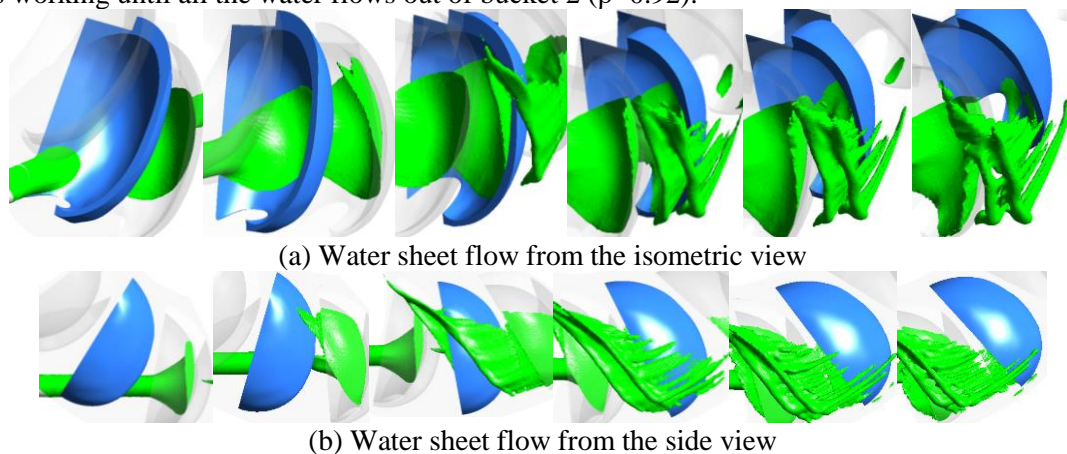


Figure 4. Developing process for water sheet flow in bucket 2 ($\beta=0.1, 0.19, 0.49, 0.74, 0.79$ and 0.92 respectively from the left to the right)

From the analysis above, it can be seen the development of the water jet flow is consistent with the power variation process. The jet flow development of the bucket 2 can be divided into 3 steps: (1) The water begins to flow into bucket 2 with little water being able to flow out. During this process, the repulsive force on bucket 2 keeps increasing. (2) The water sheet flows is fully developed on the surface of the bucket 2, where at the same time the impulsive force maintains at a stable high level. (3) The water sheet flow is pushed and flow out from the outlet edge of bucket 2. As no new water is allowed in to replenish the buckets, the impulsive force drops rapidly at this stage.

3.3. The pressure evolution of water sheet flow on the bucket surface

As the buckets are the main energy conversion components in Pelton turbine, the pressure on the buckets' surfaces can be used to indicate the strength of the torque, thus reflecting the energy conversion efficiency. So a non-dimensional pressure coefficient is defined, where the inlet pressure of the nozzle is defined as 1. The area with pressure coefficient higher than 0.5 is considered as the high pressure zone, as shown in Figure 5.

It can be found that the high pressure zone first appears near the symmetry plane of the buckets, then this zone expands towards the middle area of the bucket. During the powering increasing process ($\beta=0.01\sim0.19$), the pressure gradient is not so obvious, most pressure coefficient lies between 0.8~0.9. However, as the high pressure zone continues to expand($\beta=0.49\sim0.74$), the gradient becomes more obvious, and the pressure coefficient higher than 0.8 only comprise a small portion of the place. The whole pressure area resembles a shape of "butterfly wing". During the power decreasing

process($\beta=0.74\sim0.86$), the pressure coefficient mostly concentrates near the level between 0.2~0.3. From above analysis, it can be seen the pressure zone move along a specific path from the left side to right side. The water sheet flow area first appears near the middle of the buckets but at last disappears from the right side of the region. The high pressure zone does not exist too long for the upper region of the bucket, which indicates that optimization can be conducted for the upper region.

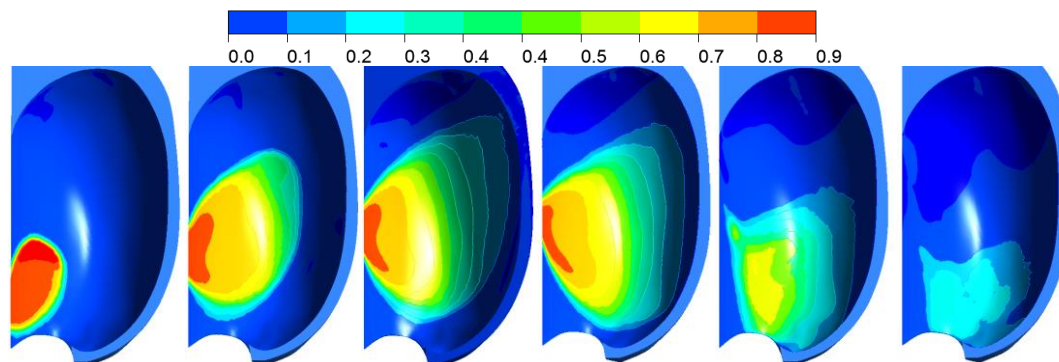


Figure 5. Distribution of pressure coefficient on bucket 2 surface ($\beta=0.1, 0.19, 0.49, 0.74, 0.79$ and 0.92 respectively from the left to the right)

4. Conclusions:

A prototype Pelton turbine for super-high head of 1100m was designed. The dynamic hydraulic performance of the prototype under design conditions were also predicted by CFD. The water sheet flow on a Pelton runner is numerically investigated, where the process of water sheet flow generating and propagation on the bucket surface were analysed. In addition, the distribution of pressure coefficient under different rotation angle were also presented.

The torque for the bucket varies with rotation angle. And the predicted output power for the Pelton turbine has not reached the designed value, which means further optimizations are needed for the super-high head conditions.

The development of water sheet flow on the bucket surface can be divided into three stages from the water contacting the bucket to leaving the buckets completely: Firstly, the jet enters the bucket after being separated by the cutout of the bucket; Then, the jet develops into free water sheet on the working surface of the bucket, where the interaction phenomenon and leakage flow from the cutout are observed during this stage; At last, the water is blocked from entering the bucket, and the left water begins to decrease.

The high pressure zone on the bucket surface first appears near the symmetry plane and at last disappear near the right side of the bucket. The pressure zone moves along a specific path from the left to right. During the power maintaining process, the pressure zone resembles a shape of a “butterfly wing”, which may be due to the shape of the buckets.

Acknowledgements

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

References

- [1] Židonis, Audrius, David S. Benzon, and George A. Aggidis. Development of hydro impulse turbines and new opportunities. *Renewable and Sustainable Energy Reviews*, 2015, 51(16): 24-35.

- [2] Perrig A. Hydrodynamics of the free surface flow in Pelton turbine buckets. *Diss. Ecole Poly Technique Federale de Lausanne*, Lausanne, Switzerland,. 2007.
- [3] Xiao Y X, Wang Z W, J. Zhang, Zeng C J, and Yan Z G. Numerical and Experimental Analysis of the Hydraulic Performance of a Prototype Pelton Turbine. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 2014, 1: 46-55.
- [4] Cobb, Bryan R, and Kendra V, Sharp. Impulse (Turgo and Pelton) turbine performance characteristics and their impact on pico-hydro installations, *Renewable Energy*, 2013, 50(3): 959-64.
- [5] Jean C M. Free surface flows simulations in Pelton turbines using an hybrid SPH-ALE method. *Journal of Hydraulic Research*. 2010, sup1:40-49.
- [6] Zeng C J, Xiao Y X, Zhang J, Ahn S H, and Wang Z W. Numerical analysis of pelton turbine needle erosion characteristics. *Journal of Drainage and Irrigation Machinery Engineering*. 2015, 33(5):407-411.
- [7] Zeng C J, Xiao Y X, Wang Z W, Zhang J, Luo Y Y. Numerical analysis of a Pelton bucket free surface sheet flow and dynamic performance affected by operating head. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 2017, 231(3): 182-196.
- [8] Zeng C J, Xiao Y X, Xu W, Wu T, Zhang J, Wang Z W, Luo Y Y. Numerical Analysis of Pelton Nozzle Jet Flow Behavior Considering Elbow Pipe. 28th IAHR symposium on Hydraulic Machinery and Systems. July 4-8th, 2016. P. 645-652. Grenoble, France.
- [9] Xiao Y X, Wang Z W, Zeng J D, Zheng J T, Lin J Y, Zhang L T. Prototype and Numerical Studies of Interference Characteristics of Two Ski-Jump Jets from Opening Spillway Gates. *Engineering Computations*, 2015, 32(2):289-307.
- [10] Zeng C J, Xiao Y X, Zhu W, Yao Y Y, Wang Z W. Numerical simulation of cavitation flow characteristic on Pelton turbine Bucket surface. ISCM2014. Oct. 20-21, 2014, Beijing. IOP Conference Series: Materials Science and Engineering. V:72, pp: 042043 (6 pp.)
- [11] Zeng C J, Xiao Y X, Zhu W, Yao Y Y, Cao L, Wang Z W. Pelton turbine Needle erosion prediction based on 3D three-phase flow simulation. 27th IAHR Symposium on Hydraulic Machinery and Systems. Sep. 22-26, 2014, Montreal, Canada.
- [12] Xiao Y X, Wang Z W, Zhang J, Zeng C J, Yan Z G. Numerical and Experimental Analysis of the Hydraulic Performance of a Prototype Pelton Turbine. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 2014, 228(1): 46-55.
- [13] Xiao Y X, Cui Tao, Wang Z W, Yan Z G. Numerical Simulation of Unsteady Free Surface Flow and Dynamic Performance for Pelton Turbine. The 26th IAHR Symposium on Hydraulic Machinery and Systems. Aug. 19-23. Beijing.
- [14] Xiao Y X, Han F W, Zhou J L, Kubota T. Numerical Prediction of Dynamic Performance of Pelton Turbine. *Journal of Hydrodynamics, Ser.B*. 2007, 19(3): 356-364.
- [15] Xiao Y X, Han F Q, Kubota T. Mechanism of Unsteady Free Water Sheet Flow on Bucket Surface. *Journal of South China University of Technology*, 2006, 34(4): 75-79+90.
- [16] Han F Q, Xiao Y X, Kubota T, Liu J. Study on the Contraction Mechanism of Free Jet. *Journal of Engineering Thermophysics*, 2004, 25(3): 421-423.