Numerical analysis of the bucket surface roughness effects in Pelton turbine

Y X Xiao, C J Zeng, J Zhang, Z G Yan and Z W Wang

Department of Thermal Engineering & State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China

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

Abstract. The internal flow of a Pelton turbine is quite complex. It is difficult to analyse the unsteady free water sheet flow in the rotating bucket owing to the lack of a sound theory. Affected by manufacturing technique and silt abrasion during the operation, the bucket surface roughness of Pelton turbine may be too great, and thereby influence unit performance. To investigate the effect of bucket roughness on Pelton turbine performance, this paper presents the numerical simulation of the interaction between the jet and the bucket in a Pelton turbine. The unsteady three-dimensional numerical simulations were performed with CFX code by using the SST turbulence model coupling the two-phase flow volume of fluid method. Different magnitude orders of bucket surface roughness were analysed and compared. Unsteady numerical results of the free water sheet flow patterns on bucket surface, torque and unit performance for each bucket surface roughness were generated. The total pressure distribution on bucket surface is used to show the free water sheet flow pattern on bucket surface. By comparing the variation of water sheet flow patterns on bucket surface with different roughness, this paper qualitatively analyses how the bucket surface roughness magnitude influences the impeding effect on free water sheet flow. Comparison of the torque variation of different bucket surface roughness highlighted the effect of the bucket surface roughness on the Pelton turbine output capacity. To further investigate the effect of bucket surface roughness on Pelton turbine performance, the relation between the relative efficiency loss rate and bucket surface roughness magnitude is quantitatively analysed. The result can be used to predict and evaluate the Pelton turbine performance.

1. Introduction

The Pelton turbine, which is an impulse turbine. Compared with other impulse turbines, Pelton turbine is more successful and widely used. It is one of the main varieties of hydro turbines among the world together with Francis turbine and axial flow turbine. Pelton turbine can perform under a wide range of water head of 100 m to 3000 m, and its output range from 50 kW to 500 MW. It requires no large dams, so it has small influence on natural environment.

The internal flow of Pelton turbine is quite complex. The objectives dealt with in the hydraulic design of Pelton turbines mainly consist of the jet flows from injectors, the interaction between the jet and rotating buckets of the Pelton wheel as well as the relative flow in the bucket [1-5]. With the improvement of machining precision of bucket surface, choice of appropriate machining precision which impose little influence on turbine performance and can save working hours has been a common concerned problem. Affected by silt erosion, the bucket surface roughness magnitude would significantly increase and thereby lead to a significant efficiency loss [6-7]. Surface roughness is an
important index of surface machining quality, thus it is necessary to take into consideration of effect of the bucket surface roughness on runner performance. So far studies of surface roughness effects on performance in gas turbines and compressors are paid great attention to \cite{8-9}, however similar studies in hydraulic machinery are rarely seen. Soviet scientist Edel studied effects of different bucket surface roughness on Pelton turbine performance with model test. He found hydraulic efficiency of runner has a significant loss with bucket surface roughness increasing. Zhang et al theoretically studied the reason of flow friction loss on bucket surface in Pelton turbine and indicated that the flow friction in the rotating buckets of a Pelton turbine influences the system efficiency in both the direct and indirect ways. The frictional flows in the rotating buckets of a Pelton turbine are non-stationary and of irregular form along the bucket surface\cite{10-11}. In other hydraulic machinery studies, Krishnamachar studied effect of runner surface roughness on hydraulic performance of Kaplan turbines \cite{12}. Li Long et al discussed the influence of wall roughness on the performance of axial-flow pumps \cite{13}. Zhang Lanjin et al studied influence of roughness of runner on the performance and cavitation of pump-turbine in the pump mode \cite{14}.

So far research about the effect of flow surface roughness on flow field and hydraulic performance in Pelton turbines is not sufficient. This paper studied high head Pelton turbine based on comparative analysis of numerical simulation of inner flow characteristic and field test performance parameters\cite{15}. Adopted SST turbulence model to simulate free water sheets flow patterns on bucket surface with different roughness, this paper quantitatively analysed effect of surface roughness on turbine output power and efficiency.

2. Prototype Pelton turbine and numerical simulation method

2.1. Prototype Pelton turbine and mesh
A high head vertical axis 19-bucket Pelton turbine is investigated, structure of nozzle and buckets is shown in Figure 1. The runner has 19 buckets so the period is 18.95 degrees of runner rotation. The bucket inner breadth $B$ and the reference runner diameter have a ratio of $B/D_{ref}=0.25$. It has a rated speed of 500 rpm with the design head of 456 m. The turbine optimum unit speed $n_{11}=41$ rpm with one injector. In the case of higher water head, gravity has litter influence on turbine performance. Thereby, the computational domain of Pelton turbine can be simplified to the half because of the structure symmetrical characteristic of nozzle and buckets, as shown in Figure 2.

\begin{figure}[h]
\centering
\includegraphics[width=0.45\textwidth]{figure1.png}
\caption{Nozzle and buckets of Pelton turbine}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=0.45\textwidth]{figure2.png}
\caption{The computational domain}
\end{figure}
Inner flow of nozzle, jet flow and free water sheet flow in the rotating buckets are taken into consideration in the numerical simulation. Thus, the computational domain is divided into three sub-domains, i.e., a nozzle, a free jet and a rotor with the runner. A transient rotor-stator sliding interface is setting between the two sub-domains, as shown in Figure 3. The shape of bucket with cutout is very complicate, an unstructured mesh was used to refine the local grid of critical area in jet flow and bucket surface, while the stator region has a relatively coarse mesh. The analysis was done for a prototype at a jet velocity of about 100 m/s. For such high velocity values, the grid near the buckets was still not refined enough due to the limitations of hardware. The total mesh has about 1,160,000 elements, 260,000 nodes.

In the same configuration of operating parameters and inlet and outlet boundary conditions, the unsteady numerical simulations were conducted on the whole interaction process of jet and bucket at four operating conditions with different surface roughness of bucket internal surface. Surface roughness was characterized by the parameter $Ra$, with unit of mm. The simulation were performed with $Ra$ of 0, 25 $\mu$m, 0.15 mm and 1 mm ($Ra=0$ means smooth surface).

2.2. Numerical model and numerical method

The multiphase flow model adopted the volume of fluid model (VOF), which can be used predict the interface shape between two or more mutually non-penetrating fluids. The model introduces the volume fraction for different phases, $\alpha_w$ for water and $\alpha_g$ for air, and determines the interface location by solving the volume fraction in each control cell. In a control cell, the volume fraction of water and air is 1, that means $\alpha_w+\alpha_g=1$. Then $\alpha_w=0$ denotes cells filled with air; $\alpha_w=1$ denotes cells filled with water; $0<\alpha<1$ denotes that the cell contains an interface between water and air. The air was set as primary phase and water secondary phase in simulations. The unsteady flow was simulated by applying the Reynolds-averaged Navier-Stokes equations together with the SST turbulence model.

Inlet boundary condition of nozzle was set as total pressure inlet with the value of design water head. Volume faction of inlet two-phase flow was 1, which means it is all water at the inlet. The peripheral face of case was set as free outflow outlet. The numerical simulation of the flow is carried out with the ANSYS Fluent 13 code in the paper. The continuity and momentum equations are discretized using an upwind scheme with the physical advection terms weighted by a gradient-dependent blend factor, providing a good trade-off between diffusion and dispersion. A first-order backward Euler scheme is used for the transient terms. According to the flow condition in jet flow and rotating bucket, the unsteady simulation time step was set as $1.58 \times 10^{-4}$ s. For obtaining a higher calculation precision to satisfy the practical requests, convergence residual was set as $10^{-4}$.

3. Numerical results and analysis

3.1. Numerical results of performance parameter
Unsteady numerical results of torque variation for runner and each bucket were generated for four different cases. The numerical results shown that bucket surface roughness has a direct impact on torque of buckets and output of runner. The smooth surface case has the maximum torque results. With the surface roughness increasing, the torque and output of same bucket at the same time step will diminish. The numerical torque results of smooth bucket and bucket with $Ra=1$ mm was shown in Figure 4, along with their difference. The horizontal axis shows the computation time. The vertical axis shows the torque of same bucket with different surface roughness, with unit of N·m. The blue diamond dotted line shows the torque variation of bucket with smooth surface, the red triangle dotted line shows the torque variation of bucket with $Ra=1$ mm, the olivine round dotted line shows the torque difference of two cases.

![Figure 4. Numerical results of torque variation for different cases](image)

The horizontal axis of time step represents the working process of single bucket, setting the moment of bucket coming into contact with jet flow as the original time step, as shown in Figure 4. As shown in the working process of single bucket, the bucket surface roughness has different impact on torque at the different phase of working process. At the beginning phase of jet flow entering the rotating bucket, the torque results of buckets with different surface roughness were almost the same. The main reason was that while jet entering the bucket along the splitter, the jet flow was not fully developed at the bucket surface and has relatively smaller contact surface with bucket. Thus, bucket surface roughness has small impact on flow patterns and torque of bucket. Then, the jet flows into bucket rapidly and spreads into thin water sheet flow with free moving boundary on the internal surface of bucket. Its contact surface with bucket expands to the whole internal surface of bucket. At this phase, surface roughness has the most significance impact on water sheet flow patterns and torque of bucket. With the surface roughness increasing, the torque of bucket at the same time step diminishes, so does the output of runner. In the end, no jet flows into bucket, while water sheet begins evacuating from the outlet of bucket. At this phase, the contact surface gradually decreases and the torque difference of two cases relatively decreases. Because the larger surface roughness has retardation effect on water sheet flow process, the working process of jet was relatively shorten.

The efficiency was calculated from torque averaged over time. Influence of bucket surface roughness on turbine efficiency loss was shown in Figure 5. The horizontal axis was $Ra$ representing the bucket surface roughness. The vertical axis was efficiency loss rate of runner with unit of %. Same trend as the influence on torque and output of bucket, with the surface roughness increasing, the efficiency loss rate of turbine increases. Relative to the reference runner efficiency of hydraulically smooth bucket, the efficiency loss rate was 0.5% while $Ra=25\mu m$, increases to 1.32% while $Ra=0.15mm$ and even surpasses 3.5% while $Ra=1mm$. The numerical results indicates that bucket surface roughness directly influences the hydraulic performance parameter, especially reduce the runner efficiency. With the surface roughness increasing, both the torque of bucket and output of runner decrease and runner efficiency loss rate has a significant increase. Thus the smooth surface of bucket can ensure the unit operating performance parameter.
3.2. Flow patterns on bucket surface

Fig. 6 and 7 shows the total pressure distribution on runner buckets and evacuating sheets for three cases of different bucket surface roughness. Flow pattern results of hydraulically smooth bucket, rough bucket with $Ra=0.15\text{mm}$ and $1\text{mm}$ are listed. High pressure on regions where the jets impact the buckets and low pressure at the water sheet boundary can be clearly distinguished. A detailed explanation of pressure distribution and water sheets during a single period is included in [15], therefore it is not repeated here.

At the first stage with jets just impact the bucket the total pressure distribution is different for different roughness cases, as shown in Figure 6, also the evacuating water sheets. The smooth bucket has biggest high total pressure area, while $Ra=1\text{mm}$ the smallest. Thus the area gradually decreases with surface roughness increasing, so as the water sheet cover on bucket surface. However, at the whole beginning phase, surface roughness has relatively smaller influence on flow patterns. The difference of total pressure distribution and the evacuating water sheets for different roughness cases is relatively smaller. In the Figure 7, it shows the total pressure distribution for different roughness cases with flow fully developed in the bucket while the jet was perpendicular to bucket location. The torque of bucket reaches the maximum, water sheet flows over the bucket and spreads to the whole internal surface. Total pressure is maximum at the splitter location and decreases along the water sheet flow direction. The smooth bucket has biggest high total pressure area relatively, while $Ra=1\text{mm}$ the smallest. Because of the accumulated influence of surface roughness on flow pattern, the flow patterns of three cases has relatively significant difference.
According to the numerical results of torque of bucket and the working process, roughness of bucket surface not only induces friction loss of water sheet flow, consumes part of the jet kinetic energy, reduces the torque of each bucket, but also has a retardant effect on water sheet flow, significantly influences the flow patterns of internal surface.

4. Conclusions

Unsteady flow simulation of bucket with different surface roughness, it can be found surface roughness will significantly decrease output and efficiency of Pelton turbine. Analyzed its specific hydraulic reason, the conclusion can be drawn as follows:

(1) Bucket surface roughness influence the working process of each bucket. With surface roughness increasing, the torque of bucket decreases. And at the different phase of working process, it has different influence. At the first stage with jet just impact the bucket, because of the small water sheet contact surface, surface roughness does not reduce the torque of bucket significantly. With the rapidly inflow of jet and expand on the internal surface, the contact surface expands and therefore, the torque of bucket decreases sharply influenced by surface roughness.

(2) According to the quantitative analysis of unit efficiency, the efficiency loss rate increases with surface roughness increasing.

(3) Roughness of bucket surface not only induces friction loss of water sheet flow, consumes part of the jet kinetic energy, reduces the torque of each bucket, but also has a retardant effect on water sheet flow, significantly influences the flow patterns of internal surface, increases the flow loss in the bucket. Therefore, it is very important to well machine the flow surface of Pelton turbine.

References

[1] Xiao Y X 2006 Numerical analysis method of unsteady interference for jet and for water sheet (Guangzhou, China: South China University of Technology)

[2] Xiao Y X, Han F Q and Kubota T 2006 Journal of South China University of Technology (Natural Science) 34(4) 75-79, 90 (in Chinese)

[3] Xiao Y X, Zheng A L, Han F Q, Ding E X and Kubota T 2007 Journal of South China University of Technology (Natural Science), 35(3) 66-70(in Chinese)

[4] Zhang Z 2007 Proceedings of the Institution of Mechanical Engineers 221 343-57.

[5] Han F Q, Xiao Y X and Kubota T 2007 Journal of South China University of Technology (Natural
[6] Padhy M K, Saini R P 2011 Energy 34(1) 141-47.
[7] Padhy M K, Saini R P 2012 Energy 39(1) 286-93.
[8] Bons J P 2010 Journal of Turbomachinery 132 021004-1-16.
[9] Hobson G V, Song S J and Millsaps K T 2012 Journal of Turbomachinery 134 051013-1
[10] Zhang Z 2007 Journal of Power and Energy 221 1173-80
[11] Zhang Z 2009 Journal of Power and Energy 223 597-608.
[12] Krishnamachar P 2008 International Journal on Hydropower and Dams 15 102-107
[13] Li L and Wang Z 2004 Transactions of The Chinese Society of Agricultural Engineering 2004(1) 132-135. (in Chinese)
[14] Zhang L J and Chang J S 2008 Journal of Hydroelectric Engineering 27(2) 125- 129 (in Chinese)
[15] Xiao Y X, Cui T, Wang Z W, Yan Z G 2012 Numerical simulation of unsteady free surface flow and dynamic performance for Pelton turbine The 26th IAHR Symposium on Hydraulic Machinery and Systems(Beijing, China, 19-23 August 2012).