Numerical prediction of hydraulic performance in model and homologous prototype Pelton turbine

C J Zeng¹, Y X Xiao*, J Zhang¹, Z H Gui², S H Wang³, Y Y Luo¹, H G Fan¹ and Z W Wang¹

¹ State Key Laboratory of Hydroscience and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China;
² Technology center state grid Xinyuan company LTD., Beijing 100032, China;
³ The Institute of Seawater Desalination and Multipurpose Utilization, SOA, Tianjin 300192, China.

xiaoyex@mail.tsinghua.edu.cn

Abstract. Pelton turbine is widely used in utilizing the high water head resource. The multiphase flow in the Pelton turbine is too complex to be analysed detailed as the flow in the reaction turbines. So the hydraulic design of Pelton turbine was mainly based on the know-how. The homologous turbine model was the only way to verify its performance in the past. Although an efficiency scale-up equation for Pelton turbines had established in the IEC 60193 code, researches had shown different internal flow phenomena would influence the efficiency scale effects and lead to different efficiency scale for different turbine design. This paper simulates the internal flow of model and homologous prototype Pelton turbine at three relative needle strokes. Homogenous model and SST k-ω model will be adopted to simulate the unsteady two-phase flow in the rotating buckets. Unsteady simulation results would help to numerical investigate the scale effect of Pelton turbine.

1. Introduction

The Pelton turbine has been proven and well-advanced as an impulse type water turbine for more than one century[1]. At present, the hydraulic design of the Pelton turbine is still relying heavily on know-how and extensive experimental testing on the reduced-scale model. Thus, the scale-up equation is vital to predict the prototype efficiencies from the model hydraulic efficiencies[2].

Although a scale-up formula for the Pelton turbine is available in the Annex K of IEC 60193 code[3], practices by GE Hydro and other hydraulic turbine manufacturers have shown that an accurate efficiency prediction of prototype is still challenging and needs lots of experience[4]. The scale-up equation for Pelton turbines in the IEC 60193 code was developed by Grein et al. and utilized the dimensionless numbers of Froude, Reynolds and Weber to characterize the efficiency scaling of Pelton turbine[4]. While, Brekke have pointed out that the proposed formula failed to take into consideration of the losses due to the complex outlet flow in multi-nozzle Pelton turbines and cannot be regarded to be valid for all types of Pelton turbines in general[6]. Han et al. proposed a scale-up method to include the losses in the pipe flow, jet flow and water sheet flow in the runner, but still lack of a general form of scale-up equation[7][8].
This paper simulates the internal flow of a pair of homologous model and prototype Pelton turbines under three similar operating conditions. Hydraulic performances of the turbines are presented along with the unsteady flow results.

2. Computational Methods

2.1. Physical model
Two geometrically similar Pelton turbines with single injector and 17 buckets are analyzed. The sketch of computational domains is shown in figure 1(a). The Pelton runner has a geometrical specific speed $B/D$ of 0.35, where $B$ is the bucket inside width and $D$ is the runner pitch diameter. The geometrically scale difference is $D_P/D_M=4.278$, where the subscript P denotes prototype and M denotes model. Three operating conditions and dimensionless numbers of $Fr$, $Re$ and $We$ are listed in Table 1 for similarity study. The dimensionless numbers of $Fr$, $Re$ and $We$ are defined as

$$Fr = \frac{w}{\sqrt{gB/2}}$$

(1)

$$Re = \frac{\rho w B/2}{\mu}$$

(2)

$$We = \frac{\rho B w^2 / 2}{\sigma}$$

(3)

$$w = v_j - \omega D/2$$

(4)

where $\mu$ is the dynamic viscosity and $\sigma$ is the surface tension of water, $v_j$ is the water jet velocity.

Each condition guarantees scale geometries for the same unit speed $n_{11}$ and the same unit flow rate $Q_{11}$ between the model and prototype. The unit speed $n_{11}$ and the unit flow rate $Q_{11}$ are defined as

$$n_{11} = \frac{nD}{\sqrt{H}}$$

(5)

$$Q_{11} = \frac{Q}{D^2 \sqrt{H}}$$

(6)

![Sketch of Computational domains](image)

(a) Sketch of Computational domains

![Mesh distribution](image)

(b) Mesh distribution

Figure 1. Computational domains

2.2. Numerical setup
This paper adopts commercial codes of Ansys CFX 15.0 to perform the unsteady free surface flow simulation. To reduce the computational domain and save computation resources, symmetry plane is adopted to model only half of the turbine bucket and runner as shown in figure 1(b). An ideal jet...
configuration is assumed for the three operating conditions with a constant velocity profile determined by the specific energy and discharge conditions. The rotating bucket walls are set as no-slip wall. A transient rotor-stator sliding interface is used between the rotor and stator domains. The casing is not considered in the simulation, so the runner worked in atmosphere and the outlet is set as opening with average static pressure of one atmosphere.\[9\]

The two phase flow is modeled by using the homogeneous model. This model assumes different fluids share the same flow field to reduce the computational load and increase the numerical stability.\[10\] Standard \( k - \varepsilon \) turbulence model cannot capture the shear stress flow near the high curvature bucket surface well. Therefore, a \( k - \omega \) based SST turbulence model with automatic wall function and curvature correction (SST-CC) is adopted for high accuracy boundary layer simulations. This model uses the Wilcoxon \( k - \omega \) model in near wall regions and the standard \( k - \varepsilon \) turbulence model in the fully turbulent region which is far away from the wall. The correctness of the transition between the two models is guaranteed by the automatic near wall treatment. Curvature correction takes the bucket curvature and runner rotation into account.\[11\] The continuity and momentum equations discretely use a high resolution scheme with the physical advection terms weighted by a gradient-dependent blend factor. While a second order backward Euler scheme is used for the transient terms.\[12\]

| Table 1. Operating conditions |
|-----------------------------|
| Operating conditions | I | II | III |
| Head, \( H \) (m) | M | P | M | P | M | P |
| 41.97 | 259.13 | 41.97 | 259.13 | 41.97 | 259.13 |
| Rotation rate, \( n \) (rpm) | M | P | M | P | M | P |
| 1033 | 600 | 1033 | 600 | 1033 | 600 |
| Unit speed, \( n_{u} \) (rpm) | M | P | M | P | M | P |
| 41 | 41 | 41 | 41 | 41 | 41 |
| Needle stroke, \( S_n \) (mm) | M | P | M | P | M | P |
| 8 | 34 | 16 | 68 | 26 | 111 |
| Flow rate, \( Q \) (L/s) | M | P | M | P | M | P |
| 8.95 | 407.09 | 15.68 | 712.90 | 20.26 | 921.31 |
| Unit flow rate, \( Q_{u} \) (L/s) | M | P | M | P | M | P |
| 20.9 | 20.9 | 36.6 | 36.6 | 47.3 | 47.3 |
| \( Fr \) | M | P | M | P | M | P |
| 20.96 | 25.18 | 20.96 | 25.18 | 20.96 | 25.18 |
| \( Re \) (10^5) | M | P | M | P | M | P |
| 7.04 | 74.85 | 7.04 | 74.85 | 7.04 | 74.85 |
| \( We \) (10^5) | M | P | M | P | M | P |
| 1.21 | 32.02 | 1.21 | 32.02 | 1.21 | 32.02 |

2.3. Mesh Scheme
Based on the different flow characteristics on the three subdomains, both hexahedral structured and tetrahedral unstructured meshes are used. Usage of hexahedral structured meshes in jet and stator domains with simple geometry and single flow direction could help to achieve high numerical accuracy and stability.\[15\] Unstructured meshes used to discretize a single bucket domain could better capture the unsteady sheet flow in the rotating bucket with curvature fluctuating widely on smooth surface. The whole rotor runner domain is formed by the 17 buckets. Meshes of jet and stator domains are aligned along the jet flow direction and refined near the interface between water and air to ensure the jet stays stable before the rotor–stator interface as shown in figure 1(b).

3. Simulation results

3.1. Hydraulic performance predictions
Torque coefficient (\( C_T \)) is introduced to assess the torque (\( T \)) of buckets and runner for the model and prototype Pelton turbines, which is defined as

\[
C_T = \frac{T}{P_i/\omega}
\]

where \( P_i \) is the input power, \( \omega \) is the angular velocity of the runner.
The time histories for torque coefficient of a single bucket are shown in figure 2 under three operating conditions. The predicted torque coefficients of model and prototype turbines have similar time histories for the similar operating condition. In the torque rising stage, the predicted coefficients of model and prototype turbines are almost the same. While, in the high torque and torque descending stages, the predicted torque coefficients of prototype turbine are a little bigger than that of model turbine. This indicates that the water flow transforms more energy to the bucket in the prototype turbine than the model turbine. Also, the time histories reveal that the energy exchange for a single bucket lasts about 510 time-steps or 90° rotating angle under the three operating conditions.

![Time histories of torque coefficient for a single bucket](image)

(a) $C_T$ of I operating condition  (b) $C_T$ of II operating condition  (c) $C_T$ of III operating condition

**Figure 2.** Time histories of torque coefficient for a single bucket

Time histories of torque coefficient for the whole runner also could be obtained. Therefore, the predicted hydraulic efficiencies of the runner under three operating conditions could be derived from the time-averaged values as shown in figure 3. Experimental efficiencies from model test are also presented in the figure 3 along with the scaled efficiencies of prototype turbine according to the IEC 60193 code. The predicted efficiency curves have similar variation with the model test results. While, the maximum efficiency difference between the simulation and experimental results is about 2.5% for the model and 0.85% for the prototype. The scale-up formula are as follows

$$\Delta \eta_h = \eta_h - \eta_{hm} = \Delta \eta_{Fr} + \Delta \eta_{We} + \Delta \eta_{Re} = 5.7 \cdot \Phi_B^2 \left(1 - C_{Fr}^0\right) + 1.95 \cdot 10^{-5} \frac{C_{We} - 1}{\Phi_B^2} + 10^{-8} \left(\frac{C_{Re} - 1}{\Phi_B^2}\right)$$

(8)

$$\Phi_B = \frac{4Q}{\pi v B^2}$$

(9)

$$C_{Fr} = \frac{F_{Fr}}{F_{FrM}}$$

(10)

$$C_{We} = \frac{We_{Fr}}{We_{FrM}}$$

(11)

$$C_{Re} = \frac{Re_{Fr}}{Re_{FrM}}$$

(12)
Table 2 shows the step-up efficiency values calculated according to the IEC 60193 code. As shown in the figure 3 and table 2, the scaled-up efficiency values (Δηₜₜ) decrease as the $Q_{11}$ increase. It is positive under the small $Q_{11}$ condition, while negative under the other two larger $Q_{11}$ conditions. Unlike the usual positive scale-up effect for reaction turbines from model to prototype, this indicates that the hydraulic efficiency scale-up effect for the Pelton turbine may be negative. The step-up values in table 2 denote that Fr has an increasing negative influence on the scale-up effect as the $Q_{11}$ increase. However, the We and Re has a decreasing positive influence on the scale-up effect.

While, the predicted hydraulic efficiencies for model and prototype Pelton turbine in figure 3 show positive scale-up effect under the three operating conditions. The predicted efficiencies for prototype are 1.26~1.56% higher than those for model. In this paper, the gravity and surface tension are not included in the simulations. Thus it could assume that the efficiency difference between the numerical results for model and prototype turbine is mainly caused by different Re.

| $Q_{11}$ (L/s) | $\Delta \eta_{Fr}$ (%) | $\Delta \eta_{We}$ (%) | $\Delta \eta_{Re}$ (%) | $\Delta \eta_{h}$ (%) |
|----------------|------------------------|------------------------|------------------------|------------------------|
| 20.9           | -0.08                  | 0.32                   | 0.04                   | 0.270                  |
| 30.6           | -0.25                  | 0.10                   | 0.01                   | -0.138                 |
| 47.3           | -0.42                  | 0.06                   | 0.01                   | -0.353                 |

3.2. Water sheet flow patterns analysis

Figure 4 shows the water sheet flow at a same time-step in the rotating runner for the model and prototype turbines under three operating conditions. Flow pattern results are similar in the model and prototype turbines under the similar condition. While, due to the difference amount of water jet inflow, the water sheet flow may hit the outside surface of the next bucket after evacuated from the bucket. Thus, the outflow angle need to be adjust to avoid the unhealthy outflow.
To evaluate the energy transfer process in the rotating bucket, total pressure coefficient ($C_{Tp}$) is introduced to make the energy data dimensionless and defined as

$$C_{Tp} = \frac{T_p - p_{ref}}{\rho g H}$$  \hspace{1cm} (13)

where $T_p$ is the total pressure, $p_{ref}$ is a reference pressure (1 atm for this paper).

Figures 5, 6 and 7 show the $C_{Tp}$ distribution on the inside surface of the rotating bucket at three different time-steps. The black line on the inside surface of the bucket is the projection of ideal jet boundary, which could represent the jet entrance position at that time-step. Figure 5, 6 and 7 depict the $C_{Tp}$ distribution at the torque rising stage, high torque stage and torque descending stage, respectively. Difference between the distribution contours between the model and prototype turbine under the same operating condition is very small. The water sheet flow spread a little fast on inside surface for the prototype turbine than the model one. While, it can clearly indicate that the water sheet flow spread faster on the inside surface as the $Q_{11}$ increase. As shown in figure 6, the water sheet is evacuating from the bucket and there are a portion of water sheet flows out from the cut-out area under the two high $Q_{11}$ conditions. Comparison of the $C_{Tp}$ distribution along the cut-out in figure 6 and 7, it can be concluded that the portion of water sheet flows out from the cut-out area will be larger as the Q11 increase. Thus, the cut-out area need to be adjust to avoid this kind of unhealthy outflow.
Figure 6. Total pressure coefficient distribution on inside surface of the bucket at 230\textsuperscript{th} time-step

Figure 7. Total pressure coefficient distribution on inside surface of the bucket at 350\textsuperscript{th} time-step

4. Conclusion
Unsteady flow simulations for a pair of homologous model and prototype Pelton turbine are carried out under three different flow rate conditions. Hydraulic performance is compared with the model test results and scaled prototype results according to IEC code. Unsteady water sheet flow patterns in the rotating bucket are discussed. Main conclusions are as follow,

- The predicted efficiency curves have similar variation with the model test results. Simulations would underestimate the hydraulic efficiency for model turbine. As the gravity and surface
tension are not included, simulations could not reflect the real scale-up relationship between the model and prototype turbines.

- The outflow angle need to be adjust to avoid the contact between the water sheet outflow and the outside surface of the next bucket.
- The cut-out area need to be adjust to avoid outflow of water sheet from this area.

Acknowledgments
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] Xiao Y, Han F, Zhou J and Kubota T 2007 Numerical Prediction of Dynamic Performance of Pelton Turbine. *Journal of Hydrodynamics, Ser.B.*
[2] Liu X, Luo Y, Karney B and Wang Z 2015 A selected literature review of efficiency improvements in hydraulic turbines. *Renewable and Sustainable Energy Review.*
[3] International Electrotechnical Commission 60193 1999 Hydraulic turbines, storage pumps and pump-turbines–Model acceptance tests. *International Electrotechnical Commission.*
[4] Risberg S and Haugen J O 2004 Upgrading of Pelton turbines - observations of how turbine design influence on model to prototype efficiency scaling. In *Hydraulic Machinery and Systems. Proceedings of 22nd IAHR Symposium.*
[5] Grein H, Meier J and Klicov D 1986 Efficiency scale effects in Pelton turbines. *Proceedings of 13th IAHR Symposium.*
[6] Brekke H 1996 Analysis of Losses in Hydraulic Turbines. *Proceedings of 18th IAHR Symposium.*
[7] Han F, Kubota T and Wei C 2000 A New Formula for the Performance Conversion of Pelton Turbine (I). *Journal of Huazhong University of Science and Technology.*
[8] Han F, Liu J, Wei C and Kubota T 2000 A New Formula for the Performance Conversion of Pelton Turbine (II). *Journal of Huazhong University of Science and Technology.*
[9] Xiao Y, Wang Z, Zhang J, Zeng C and Yan Z 2014 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.*
[10] Xiao Y, Wang Z, Zeng, J, Zhang J, Lin J and Zhang L. 2015 Prototype and Numerical Studies of Interference Characteristics of Two Ski-Jump Jets from Opening Spillway Gates. *Engineering Computations.*
[11] Zeng C, Xiao Y, Wang Z, Zhang J and Luo Y 2017 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.*
[12] Zhao X, Xiao Y, Wang Z, Luo H, Ahn S, Yao Y and Fan H 2018 Numerical analysis of non-axisymmetric flow characteristic for a pump-turbine impeller at pump off-design condition. *Renewable Energy.*
[13] Ahn S, Xiao Y, Wang Z and Luo Y 2017 Numerical prediction on the effect of free surface vortex on intake flow characteristics for tidal power station. *Renewable Energy.*