Numerical simulation of cavitation flow characteristic on Pelton turbine bucket surface

C J Zeng, Y X Xiao, W Zhu, Y Y Yao 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 in the rotating bucket of Pelton turbine is free water sheet flow with moving boundary. The runner operates under atmospheric pressure and the cavitation in the bucket is still a controversial problem. While more and more field practice proved that there exists cavitation in the Pelton turbine bucket and the cavitation erosion may occur at the worst which will damage the bucket. So a well prediction about the cavitation flow on the bucket surface of Pelton turbine and the followed cavitation erosion characteristic can effectively guide the optimization of Pelton runner bucket and the stable operation of unit. This paper will investigate the appropriate numerical model and method for the unsteady 3D water-air-vapour multiphase cavitation flow which may occur on the Pelton bucket surface. The computational domain will include the nozzle pipe flow, semi-free surface jet and runner domain. Via comparing the numerical results of different turbulence, cavity and multiphase models, this paper will determine the suitable numerical model and method for the simulation of cavitation on the Pelton bucket surface. In order to investigate the conditions corresponding to the cavitation phenomena on the bucket surface, this paper will adopt the suitable model to simulate the various operational conditions of different water head and needle travel. Then, the characteristics of cavitation flow the development process of cavitation will be analysed in great detail.

1. Introduction
The Pelton turbine can be applied to a wide range water head from 30 to 3000 m. The unit working under great water head has a high flow velocity and the cavitation may occur on the bucket surface. The Pelton runner is the most important component of the unit for energy transformation. During the operation process the bucket cut into the high velocity jet, thus the buckets suffer from a pulsed stress. The operation condition is so bad that the occurrence of cavitation will damage the bucket surface and even destroy the runner structure.

The internal flow is quite complicated in the Pelton turbine and there exists controversy in the academic circle that whether the cavitation occurs in the Pelton turbine. By applying the Bernoulli equation, Zhou et al hold the opinion that the Pelton runner operates under atmospheric pressure and there is no low pressure area on the working surface of bucket. The damage of bucket rear surface and side surface is a kind of fatigue failure caused by the repeated impact [1]. However according to the practice runner operation condition, Li pointed out that for a certain runner used in high water head, the cavitation easily occurs near the splitter tip due to the poor manufacture process [2]. Rossetti et al

---

1 Corresponding author.
simulated the cavitating flow phenomenon in the Pelton turbine. The occurrence place and the development process of water vaporization and condensation is analyzed during the bucket cut-in process to the jet flow. The qualitative analysis shown that the pitting cavitation occurred at the rear surface of splitter tip and the position is coincidence with the damage part of the test case runner [3].

With the development of the computational fluid dynamics, the unsteady multiphase internal flow of the Pelton turbine can be simulated. This paper adopted Eulerian-Eulerian multiphase flow model and analyzed the water-air flow characteristic in the Pelton turbine without cavitation at first stage. Then with the identification of low pressure area on the bucket surface, the numerical simulation method for cavitation flow in the Pelton turbine was investigated and the cavitation flow characteristic was analyzed.

2. Simulation model

2.1. Governing equations

This simulation adopted the Eulerian-Eulerian multiphase flow model. As the flow in the Pelton turbine is a free surface flow where the interface is well defined, the homogeneous multiphase model is employed to simplify the simulation. Thus the common flow field is regarded as shared by all fluids. Before the numerical analysis of water-air-vapour multiphase flow, the unsteady water-air flow simulation is conducted to get a reasonable initial condition for the cavitation flow analysis.

The homogeneous model assumes all fluids share the common flow field. And the flow field can be obtained by the continuity equation and momentum equations, which is dependent on the volume fractions of all phases through the properties $\rho$ and $\mu$ as shown below.

$$\rho = \sum_{\alpha=1}^{N_p} \rho_{\alpha} \tag{1}$$

$$\mu = \sum_{\alpha=1}^{N_p} \mu_{\alpha} \tag{2}$$

$$U_{\alpha} = U_{1}, 1 \leq \alpha \leq N_p \tag{3}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j) = 0 \tag{4}$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \tau_{ij} - \rho u_i u_j \right) + S_M \tag{5}$$

The Reynolds stresses arise from the nonlinear convective term in the un-averaged equations. They reflect the fact that the convective transport due to turbulent velocity fluctuations will act to enhance mixing over and above that caused by thermal fluctuations at the molecular level. To close the equation, the eddy viscosity turbulence models are introduced to model the additional terms of Reynolds stresses. And the Shear Stress Transport model is chosen for achieving highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. The equation for the turbulence viscosity, $\mu_t$, turbulence kinetic energy, $k$, and turbulent frequency, $\omega$, are as shown below.

$$\mu_t = \frac{\rho a_i k}{\max(a_i \omega, SF_i)} \tag{6}$$

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} \left[ \frac{\mu + \mu_t}{\sigma_k \rho} \frac{\partial k}{\partial x_j} \right] + P_k - \beta \rho k \omega + P_{ib} \tag{7}$$

\[\text{International Symposium of Cavitation and Multiphase Flow (ISCM 2014)} \quad \text{IOP Publishing} \]
\[\text{IOP Conf. Series: Materials Science and Engineering} \quad 72 \ (2015) \ 042043 \quad \text{doi:10.1088/1757-899X/72/4/042043}\]
\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu}{\sigma_{\infty}} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{2 \rho (1 - F_1)}{\sigma_{\infty} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \frac{a_s P_i \omega}{k} - \beta_i \rho \omega^3 + P_{ob} \tag{8}
\]

The Rayleigh Plesset model is adopted to simulate the interphase mass transfer between the water and water vapor. The Rayleigh-Plesset equation provides the basis for the rate equation controlling vapor generation and condensation. And the equation describing the growth of a gas bubble in a liquid is given by:

\[
R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left( \frac{dR_B}{dt} \right)^2 + \frac{2 \sigma}{\rho_f R_b} = \frac{p_v - p}{\rho_f} \tag{9}
\]

Neglecting the second order terms (which is appropriate for low oscillation frequencies) and the surface tension, and assuming there are \(N_B\) bubbles per unit volume, the total interphase mass transfer rate per unit volume of vaporization and condensation is:

\[
\dot{m}_{fg} = F \frac{3 \rho_s R_B}{R_B} \sqrt{\frac{2}{3}} \frac{|p_v - p|}{\rho_f} \text{sgn}(p_v - p) \tag{10}
\]

### 2.2. Physical model

A vertical axis 19-bucket Pelton turbine is investigated. The ratio of the bucket inner breadth \(B\) and the reference runner diameter \(D_{ref}\) is \(B/D_{ref}=0.25\). The turbine optimum unit speed \(n_{11}=41\) rpm with one injector, a rated speed of 500rpm with the design head of 456 m. In the case of high water head, gravity has little influence on the turbine performance. Thereby, as the incoming jet is assumed to be ideal, the computational domain of Pelton turbine can be simplified to the half because of the structure symmetrical characteristic as shown in Figure 1.

![Figure 1. Computational domain](image1)

![Figure 2. Grid distribution](image2)
the angle between two adjacent buckets. For obtaining a higher calculation precision to satisfy the practical requests, the convergence residual was set as $10^{-4}$.

3. Results and discussion

3.1. Numerical results of no cavitation flow
Firstly, this article investigated the water-air two phase flow in the Pelton turbine. The internal unsteady flow results of Pelton turbine have been obtained without the consideration of cavitation. As the runner buckets cut through the free jet periodically, the working process of a single bucket is periodically. Figure 3 shows the torque variation curves of the cutting procedure. The detailed analysis of torque variation process can be seen in the reference [4].

![Figure 3. Torque variation curves of a single bucket](image)

The pressure distribution on the bucket front surface during its working process is analysed in Figure 4. As the bucket cutting through the free jet, the free jet began to impact on the front surface and became into free water sheet. The light blue isosurface of water volume fraction indicates the free jet boundary and the white lines indicates the boundary of the free water sheet flow region on the bucket surface. As can be seen from the Figure 4(a), the low pressure area began to appears near the cut-out among the water sheet flow region. With the development of flow, the low pressure expanded to the most at 3.52e-3 s and disappeared at 5.17e-3 s as shown in Figure 4(b) and Figure 4(c). The low pressure area among the water sheet flow region indicated the cavitation may occur on the bucket front surface.

3.2. Numerical results of cavitation flow
Based on the water-air two phase numerical results, this article conducted water-vapour-air three phase simulation on the internal flow of Pelton turbine to study the cavitation flow. The mass transfer model between water and water vapour adopted the Rayleigh Plesset model and the saturation pressure was set as 3574 Pa of 25 °C water. The numerical result is shown in Figure 5. As the water sheet flow spreads on the bucket front surface, cavity began to grow near the splitter. In Figure 5(a), the red vapour surface shows the initial cavity on the bucket front surface. The cavitation area shows good accordance with the low pressure area in Figure 4(a). Also the inflated cavity in Figure 5(b) and the shrunk cavity in Figure 5(c) are consistent with the pressure distributions in Figure 4.

3.3. Effect of cavitation on turbine performance
The torque variation curves of single bucket are compared in Figure 6. It can be seen from the difference between two cases that the maximum torque diminished and the working process is shorten because of the cavitation. It can be distinguished that the average torque of runner decrease because of
the cavitation. Because the efficiency of the injection mechanism is of the runner efficiency, the cavitation would decrease the output capacity of whole unit.

Figure 4. Low pressure area on the Bucket front surface

Figure 5. Cavity on the Bucket front surface

Figure 6. Bucket Torque curves of two cases
4. Conclusions
This article simulated the three dimensional unsteady flow in Pelton turbine. With the consideration of cavitition model, the water-vapour-air three phases numerical method is investigated. The main conclusion can be draw as follows:
(1) The water-vapor-air three phase model can be applied to simulation of cavitition flow in Pelton turbine. The predicted cavitition area is in good accordance with the low pressure area in the two phase simulation.
(2) Cavity may occur on the front surface of Pelton bucket. The three phases numerical results indicated cavity would initial near the cut-out on the front surface at the beginning phase of bucket cutting through the free jet.
(3) Cavitition can influence the unit performance. The numerical results indicated cavitition would decrease the output of Pelton runner.

Acknowledgments
Special thanks are due to the National Natural Science Foundation of China under Contract (No. 51479093), State Key Program of National Science of China (Grant No. 51439002), State Key Laboratory of Hydroscience and Engineering (Grant No. 2014-KY-05) for supporting the present work.

References
[1] Zhou W T 2008 Large Electric machine and Hydraulic Turbine 2 44-53.
[2] Li L 2014 Design and Application of Pelton Runner Transform and Upgrade Traditional Industries to Develop Strategic Emerging Industries Conference Proc.
[3] Rossetti A, Pavesi G, Ardizzon G and Santolin A 2014 J. Fluid Eng. 136(8) 081304
[4] Xiao Y X, Wang Z W, Zhang J, Zeng C and Yan Z 2014 Proc. of the Institution of Mech. Engineers, Part A: J. of Power and Energy 228(1) 46-55
[5] ANSYS ANSYS CFX User’s Guide (ANSYS Inc.).
[6] Xiao Y X, Cui T, Wang Z W and Yan Z G 2012 26th IAHR Symposium on Hydraulic Machinery and Systems( Beijing, China, 19-23, August, 2012)
[7] Xiao Y X, Han F Q, Zhou J L and Kubota T 2007 J. of Hydrodynamics. Ser. B 19(3) 356-64
[8] Xiao Y X 2013 IOP Conference Series: Materials Sci. and Eng. 52(5) 052032