Numerical investigation for one bad-behaved flow in a Pelton turbine

X Z Wei1, 2, 3, K Yang1, 2, 3, 4, H J Wang3, R Z Gong3 and D Y Li3
1 Harbin electric power equipment co., Ltd, Harbin 150040, China
2 Harbin Institute of Large Electrical Machinery, Harbin 150040, China
3 Harbin Institute of Technology, School of Energy Science and Engineering, Harbin 150001, China
E-mail: woshiyangkang@126.com

Abstract. The gas-liquid two-phase flow in Pelton turbines is very complicated, there are many kinds of bad-behaved flow in Pelton turbines. In this paper, CFD numerical simulation for the Pelton turbine was conducted using VOF two-phase model. One kind of bad-behaved flow caused by the two jets was captured, and the bad-behaved flow was analysed by torque on buckets. It can be concluded that the angle between the two jets and the value of ratio of runner diameter and jet diameter are important parameters for the bad-behaved flow. Furthermore, the reason why the efficiency of some multi-jet type turbines is very low can be well explained by the analysis of bad-behaved flow. Finally, some suggestions for improvement were also provided in present paper.

1. Introduction
The gas-liquid two-phase flow in Pelton turbines is turbulent and unsteady. Different from the reaction turbines, the hydraulic performance of a Pelton turbine is dynamic due to the unsteady flow in the rotating buckets in time and space [1]. Besides, the free surface flow should be modelled by a multiphase model [2]. So numerical simulation of the free surface flow in Pelton turbines is more complex and time consuming.

In recent years, the complex free surface flow is allowed to be simulated and developed by the CFD technology. A great deal of research of the free jet from the injector was done by Zhang [3-4]. One of the first attempts to simulate the free surface flow in a Pelton turbine was done by Kvinsky [5]. Numerical prediction on the efficiency of a Pelton turbine was done by Jost [2]. Numerical investigation of the bad-behaved flow caused by the interaction between jet and bucket in a Pelton turbine was done by Santolin [6]. On the basis of results mentioned above, the accuracy of CFD simulation on free surface flow in a Pelton turbine was proved, and it can be concluded that the research on bad-behaved flow in a Pelton turbine is very significant.

The bad-behaved flow mentioned in this paper can be explained as follows. There are two kinds of flow in one bucket at the same moment, one belongs to the present jet, and the other belongs to the previous jet. The fluid left in the bucket from the previous jet has not flowed out completely when the fluid from the present jet has already flowed in this bucket. So the two flows interfere with each other.
and the interference lead to this bad-behaved flow. This bad-behaved flow exists in the multi-nozzle pelton turbine. It can reduce the power output and efficiency of the turbine.

2. Numerical simulation

In this paper, the RNG $k$-$\epsilon$ model was chosen to close the momentum equation, which is used to describe the flow in a pelton turbine. The VOF multiphase model was chosen to capture the free surface of the two phase flow. There are two domains in the simulation, one is the rotating domain, the other is the static domain, and the data is transferred through the sliding interface between the domains. Besides, the independence verification of the grid number and the boundary conditions are described as follows.

2.1. The grid independence verification

The unstructured mesh was used for the simulation. In order to get a more accurate shape of the flow, the grid at the inlet and the buckets were refined separately. The grid of the rotating domain and the static domain are shown in Figure 1. In order to verify the independence of the grid number, calculations with different grid number were done, and the calculated efficiency of different grid is shown in Figure 2. Results show that the calculation efficiency tends to be stable as the grid number increases. Finally, the grid with 15 million mesh number is selected in consideration of the calculation efficiency and calculation time.

![Figure 1. Computational domain and grids for numerical analysis: a) rotating domain; b) static domain](image)

![Figure 2. Independence verification on numbers of grid](image)

2.2. Boundary conditions

As for inlet boundary conditions, the values of jet velocity and water volume fraction were prescribed. The outlet boundary condition was prescribed at the outlet of the rack with a constant pressure.
Transient calculations were performed with a time step of 0.5 degrees of runner rotation. 30 iterations were performed per each time step.

In this article, $\alpha$ is defined as the angle between jet 1 and jet 2, $V$ is the jet velocity, $\psi$ is the ratio of runner linear velocity and jet velocity, $M$ value is the ratio of runner diameter and jet diameter. Numerical analysis was performed at 8 operating points with maximum opening in this paper as shown in Table 1.

| Points | $\alpha$ (°) | $\psi$ | $M$ value |
|--------|-------------|--------|-----------|
| 1      | 75          | 0.48   | 9.2       |
| 2      | 180         | 0.48   | 9.2       |
| 3      | 110         | 0.48   | 9.2       |
| 4      | 90          | 0.48   | 9.2       |
| 5      | 75          | 0.45   | 9.2       |
| 6      | 75          | 0.42   | 9.2       |
| 7      | 75          | 0.35   | 9.2       |
| 8      | 75          | 0.48   | 11        |

3. Phenomenon analysis of the design condition

Operating point 1 is the design condition of this pelton turbine. Figure 3 shows the water flow within domains of point 1, it can be seen that the fluid from jet 2 is flowing into the bucket 5 while the fluid left in bucket 5 coming from jet 1 is still flowing out at this moment. So it can be concluded that the bad-behaved flow exists in bucket 5.

Figure 4 shows the value of the torque on the bucket 5 during the calculation at operating point 1, there are two torque peaks generated by the two jets. It is obvious that the torque peak generated by jet 2 is much smaller than that generated by jet 1. Then the average value of the torque generated by the two jets were calculated, the average value of the torque generated by jet 1 and 2 is 9170N·m and 7960N·m, respectively. It’s the bad-behaved flow leads to the difference of the value torque generated by the two jets, which makes the torque generated by jet 2 much smaller than that generated by jet 1. In other words, the bad-behaved flow reduces the working capability of jet 2.

In order to verify this kind of bad-behaved flow, the parameter $k$ is defined as the ratio of the average value of the torque generated by jet 1 and the average value of the torque generated by jet 2. Therefor when $k=1$, it can be claimed that the bad-behaved flow does not appear, and when $k>1$, it can be claimed that the bad-behaved flow appear. And the larger $k$ is the more serious this kind of bad-behaved flow is. $k=1.152$ was calculated in operating point 1.
This bad-behaved flow is determined by two time, one time called $T_1$ is the time that a bucket spends rotating from jet 1 to jet 2, the other time called $T_2$ is the time that the fluid from jet 1 costs flowing out from the bucket completely. It can be obtained that this kind of bad-behaved flow does not appear when $T_1 > T_2$, and it appears when $T_1 < T_2$.

In this article, $\omega$ is defined as the angular velocity of the runner, therefore $T_1$ is the ratio of $\alpha$ and $\omega$. Besides, as we know $T_2$ is inversely proportional to $V$. This kind of bad-behaved flow does not appear when $T_1 > T_2$, in order to avoid the bad-behaved flow, $T_1$ need to be longer while $T_2$ need to be shorter, in other words, $\alpha$ and $V$ should be increased.

### 4. Schemes to avoid the bad-behaved flow

Considering the analysis above, three schemes were designed to avoid the bad-behaved flow, scheme 1 is to increase $\alpha$ which means to increase $T_1$, scheme 2 is to increase $V$ which means to reduce $T_2$, and scheme 3 is to reselect the runner which also means to reduce $T_2$.

#### 4.1. Scheme 1

In this scheme, the angle between jet 1 and jet 2 is increased to avoid the bad-behaved flow. Operating points 2-4 are analysed in this scheme.

The water flow within the domains of the operating point 2-4 is shown in Figure 5. It can be obtained that the $\alpha$ is big enough to avoid this bad-behaved flow in operating point 2 and operating point 3, fluid from jet 1 has already flowed out completely from the bucket A when bucket A rotates from jet 1 to jet 2. However it can be obtained that the bad-behaved flow appears slightly in operating point 4. In the present moment, fluid from jet 2 has already flowed into bucket A while there is still some fluid left from jet 1 flowing out from bucket A.

![Figure 5. Water flow in the domains of operating point 2-4](image)

Then the parameter $k$ of the operating point 2-4 is calculated, as shown in Table 2. The bad-behaved flow doesn’t appear in operating point 2-3 because of the value of $k$ is approximately equal to 1. And the value of $k$ is 1.0132 in operating point 4, therefore the bad-behaved flow appears slightly in point 4, and 1.0132 is much smaller than the value 1.152 in design condition point, so it can be concluded that the bad-behaved flow have been greatly improved in operating point 4.
4.2. Scheme 2
In this scheme, the jet velocity is increased to avoid the bad-behaved flow. The purpose of increasing $V$ is to reduce $T_2$. The linear velocity of runner is determined by the angular velocity of runner and can be seen as constant, so $\psi$ can be used to measure the jet velocity. In order to investigate the influence the jet velocity on the bad-behaved flow, operating points 5-7 are analysed in this scheme.

The water flow in the domains of operating points 5-7 are shown in Figure 6. It can be obtained that the fluid from jet 1 has not flowed out completely when bucket A rotates from jet 1 to jet 2 at operating point 5 and operating point 6, so the bad-behaved flow still appears in operating point 5 and operating point 6. However it seems that the jet velocity is big enough to avoid the bad-behaved flow in operating point 7, the fluid from jet 1 has already flowed out completely at the moment bucket A rotates to jet 2 in this operating point, and the bad-behaved flow does not appear in operating point 7.

Then the parameter $k$ of the operating point 5-7 is calculated. As shown in Table 3, the value of $k$ in operating point 5 and operating point 6 is still greater than 1. It means that the bad-behaved flow still appears in these operating points, and the value of $k$ is much smaller than that of operating point 1. It means that the bad-behaved flow has already been improved. However $k=1.0008 \approx 1$ at operating 7. It means that the bad-behaved flow is avoided in operating point 7.

It is known that the efficiency of pelton turbine is determined by $\psi$. In this scheme, the bad-behaved flow was avoided by increasing jet velocity. But $\psi$ was changed at the same time and the efficiency reduced, so this scheme can tell us how the jet velocity influences the bad-behaved flow, but it can’t be a reasonable scheme to settle the problem of the hydropower station.

### Table 2. The parameter $k$ calculated in Scheme 1

| Operating points | $\alpha(\degree)$ | $k$      |
|------------------|-------------------|----------|
| Operating point 2 | 180               | 1.0004   |
| Operating point 3 | 110               | 1.0005   |
| Operating point 4 | 90                | 1.0100   |

**Figure 6.** Water flow in the domains of operating point 5-7
Table 3. The parameter $k$ calculated in Scheme 2

| Operating points | $\psi$ | $k$  |
|------------------|--------|------|
| Operating point 5| 0.45   | 1.0900 |
| Operating point 6| 0.42   | 1.0400 |
| Operating point 7| 0.35   | 1.0008 |

4.3. Scheme 3

In this scheme, in order to increase the jet velocity while keeping $\psi$ constant, the runner is reselected. The $M$ value is the ratio of the runner diameter and the jet diameter. It has a significant influence on pelton turbine. The $M$ value of the original runner is 9.2, and the number of buckets is 17. Then the $M$ value is reselected as 11 while the number of buckets is 19. Reselection of $M$ value can increase the jet velocity while keeping $\psi$ constant.

The water flow within the domains of operating point 8 is shown in Figure 7. It can be obtained that the fluid coming from jet 1 has flowed out completely when bucket A rotates from jet 1 to jet 2, and the bad-behaved flow does not appear in this operating point.

Figure 7. Water flow in the domains of operating point 8

Then the parameter $k$ of operating point 8 is calculated. As shown in Table 4, for this new runner, $k=1.0008\approx1$, it means that the bad-behaved flow is avoided after reselecting the runner.

Table 4. The parameter $k$ calculated in Scheme 3

|                | $M$ value | Number of bucket | $\psi$ | $k$  |
|----------------|-----------|------------------|--------|------|
| Point 1(Original runner) | 9.2       | 17               | 0.48   | 1.1520 |
| Point 8(New runner)        | 11        | 19               | 0.48   | 1.0004 |

5. Conclusions

The numerical analysis of flow in a Pelton turbine was performed for 8 operating points. Some conclusions can be obtained on the basis of the results:

1) There is a bad-behaved flow exists in the multi-nozzle pelton turbine, and it can decreased the working capability of some jets.

2) The bad-behaved flow is influenced by the angle of two jets, it can be avoid when the angle is big enough.

3) The bad-behaved flow is influenced by the jet velocity, it can be avoid when the jet velocity is big enough and it can also be avoid by increase the M value.
References
[1] Perrig A, Avellan F, Kueny J, Farhat M and Parkinson E 2006 J. Fluids Eng., Trans. ASME 128 350–8
[2] Jošt D and Mešnar P 2010 Earth and Environmental Sc. 12 012080
[3] Zhang Z and Casey M 2007 Power and Energy 221 1181-92
[4] Zhang Z 2009 Proc. of the Institution of Mech. Eng. 233 609-23
[5] Kvicinsky S, Kueny J L, Avellan, F and Parkinson E 2002 Experimental and numerical analysis of free surface flows in a rotating bucket 27th IAHR Symposium on Hydraulic Machinery and Systems (Montréal, Canada, 22-26 Sep 2014)
[6] Santolin A, Cavazzini G, Ardizzon G and Pavesi G 2009 Proc. of the Institution of Mech. Eng., Part A: J. of Power and Energy 233(6) 721-8