Physics of bad-behaved flow in 6-Nozzle Pelton turbine through dynamic simulation

Han Lei, Zhang Gaofu, Gong Ruzhi, Wei Hongda, Li Wei

School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China
E-mail: hit.hanlei@hit.edu.cn

Abstract. The bad-behaved flow in Pelton turbine, particularly in the runner region influences the efficiency performance of the system, sometimes brings out problems such as cavitation and efficiency loss. This paper will focus the physics of negative flow which are interaction of the cutting edge, two jets flow into the same bucket and flow towards the back side of bucket. Dynamic rotating process is numerically investigated in the runner region using CFX commercial code under one operating condition. These unsteady numerical simulations were conducted by RNG k-ε turbulence model combined with two-phase homogeneous model. Through the whole working process of the water jet on the bucket, the torque of the bucket and the whole runner is analyzed in detail. Four parts are divided as bucket head, pressure surface, suction surface and cutting edge region. This study deeply recognized the influences of two bad-behaved flows and proves that they are the main reasons of the torque decrease characteristic.

1. Introduction
From more than hundred years, Pelton turbine develops mostly on test and trial experimental developments, expertised by project engineers[1]. The working principle is the momentum transfer occurring between two parts when the jet from an injector impacts buckets[2]. This type of turbine is mostly used in the mountainous areas which has high water head at low flow rates relatively. For instance, some plant is built up to 1800m head. A head of 1000m for instance represents a potential energy of about $9.8\times10^6 J/m^3$, which has a jet velocity up to $140m/s$[3]. Furthermore, in the PRO (pressure-retarded osmosis) pilot plant, Pelton turbine is used to recover the energy[4][5].

Besides the importance and long history of Pelton turbine, the flow characteristics of the flow inside have been widely investigated by researchers. A small sized Pelton turbine is tested by [2]. Test results show that when the flow rate is above critical value, the efficiency of the turbine was nearly constant. However, the efficiency decreases rapidly when the flow rate is below critical value. The head loss coefficient of the injector is about 0.06 above a critical flow rate but it is doubled to 0.13 below critical value. One reason is that the jet diameter changes unpredictably depending on flow rates and the velocity profile across the water jet is not constant due to the spear wake in the injector. Hence, the flow in the injector is important to be studied.

In the experimental side, [3] found that the non-uniform and rotational flow conditions the injector introduces secondary flows in the bifurcations which seriously affect the development of the jet. As Laser Doppler anemometry (LDA) was used in the measurement of the flow field of Pelton turbine, it has been approved to be effective to investigate the jet flow behind
the nozzle[6][7]. [8] compared the experimental pressure data and numerical data which is measured on a fixed bucket with a uniform jet cutting. They verified the usability of VOF and homogeneous model in Pelton turbine. [9] performed the experiments to measure the thrust and torque of Pelton turbine with validation of flow visualization. The thrust and torque varies with the jet incidence and reach the maximum between 90° to 100°. [10] carried out a design of experiments (DOEs) study to find the impact of nozzle and spear on the injector losses. Results show that the improved style with steeper injector reduces the losses up to 0.6%.

As CFD is developed rapidly recently [11], so many platforms such as ANSYS CFX, ANSYS Fluent, Star-CCM+ or OpenFOAM software packages are well developed and used widely by the researchers. More and more simulations show good agreement with experimental data[12][13].

[14] investigates numerical model and method for the unsteady 3D water-air-vapor multiphase cavitation flow. The predicted cavitation area is confirmed by the low pressure area calculated by the classical two phase model. Furthermore, with cavitation the output of Pelton runner will decreased. [15] simulated a single-jet Pelton turbine with two different jet configurations. One is ideal jet configuration, the other one is equipped with all real working parts of spear. Through the comparison between the two configurations, it concludes that the real jet flow and the jet shape modified the jet-bucket interaction. For cavitation research, [16] presented three different vapor volumes in different zone in the bucket and highlighted different evaporation mechanics. Furthermore, [17] clarified three types of negative flow in Pelton turbine which includes flow interaction between two jets. At the same time, they indicated that the ratio of runner diameter and jet diameter and also the angle between the two jets are the main parts of the bad-behaved flow.

Meanwhile, the bad-behaved flow should be investigated in detail. The rotating bucket optimization requires further study relating to the unsteady feeding, centrifugal and Coriolis forces and interference of the sheets of water[9].

Hence, this paper will focus on the complex two phase flow in the Pelton turbine. Firstly, numerical cases in different spear opening and total head condition are calculated in order to analyze the flow characteristics in the distributor tube and nozzle system. Accordingly, energy loss will be discussed in detail. Furthermore, the free jet flow behind the nozzle system is analyzed in detail which includes the jet-bucket interaction process in order to find out the bad-behaved flow mechanism and point out how to improve the bucket design.

2. Simulation strategy and mesh information

2.1. Mesh information

In the last two decades, Computational Fluid Dynamics (CFD) has become a potential tool for designing and optimizing the Pelton turbines. The Pelton model is chosen as the parameters shown in the Tab.1. A numerical Pelton turbine is built by UG software with the distributor and 6 nozzles structure as shown in Figure.1. The calculating mesh is made by ICEM. This

| Parameter                  | Symbol | Value  |
|---------------------------|--------|--------|
| Number of bucket          | Z      | 21     |
| Nozzle diameter/mm        | d      | 41.49  |
| Width of bucket/mm        | B      | 128.53 |
| Base diameter of runner/mm| D      | 497.58 |
| Effective Head/m          | H      | 80     |
mesh calculates the flow in the distributer and also the free jet flow passing through the nozzle in order to analyze the performance of the distributer and nozzle system with help of the loss theory. Four different node quantities which includes 2.16 million, 3.18 million, 4.14 million and 6 million were used to varify the mesh independency as shown in Figure 2. It is obviously that when the node quantity reaches above 4.14 million, head changes less than 0.2%. Finally, the totality of the mesh with 4.14 million was chosen in this paper. The second mesh includes the runner(bucket) region and also the distributor as shown in Figure 3. This mesh is designed for detecting the complex flow in the runner region and then for find out the bad-behaved flow mechanism. For improving the calculating efficiency, the mesh of the distributor is coarser than the nozzle part, in the same time, the region of the jet and the bucket is fulfilled with the dense grid which can protect the mesh quality and make the $y^+$ more adaptable to the turbulent model. The totality of this type of mesh reaches 3.59 million.

![Structured mesh of the distributer and the nozzle.](image)

**Figure 1.** Structured mesh of the distributer and the nozzle.

![Mesh independency verification through four different mesh.](image)

**Figure 2.** Mesh independency verification through four different mesh.

2.2. **Controlling equations and turbulence model**

[18] claimed that the basic assumption of the fluid is incompressible which is predicted available to turbine calculation. The numerical simulation is performed by CFX code. The continuity and momentum equations are discretized using a second-order upwind scheme. Renormalization Group(RNG) $k – \varepsilon$ turbulent model is chosen in this paper to close the N-S equations. This

![Unstructured mesh of the distributer, nozzle and bucket.](image)

**Figure 3.** Unstructured mesh of the distributer, nozzle and bucket.
model is obtained via a statistical mechanics approach, in which the small scale motions are
systematically removed from the governing equations by expressing their effects in terms of
larger scale motions and a modified viscosity[19]. For the transport term, they are interpreted
as below:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \mu_i k) = \frac{\partial}{\partial x_j} (\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}) + G_k - \rho \varepsilon
\]

(1)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \mu_i \varepsilon) = \frac{\partial}{\partial x_j} (\alpha_\varepsilon \mu_{eff} \frac{\partial k}{\partial x_j}) + \frac{C_1^* \varepsilon}{k} G_k - C_2 \rho \varepsilon^2
\]

(2)

Through modifying the turbulent model, RNG \(k-\varepsilon\) considers the swirling and vorticity flow
in average flow status. Furthermore, in \(\varepsilon\) equation, it adds a term representing the time-average
stress \(E_{ij}\). Therefore, RNG \(k-\varepsilon\) model can precede well the high shear stress. Flow in the
pelton turbine is with great curvature which is similar to the classical bend pipe flow, so this
RNG model is adaptable in this case.

2.3. Two-phase calculation model: Volume of Fluid (VOF)

In the Pelton turbine, the surface structure between water phase and air phase is obtained by
volume of fluid (VOF) which is a classical methodology[20]. This model will classify the phase in
each mash by calculating the fraction between water and air. For example, \(\phi = 1\), it means that
water fills up the mesh zone. Oppositely, for \(\phi = 0\), there is only air inside. When \(0 < \phi < 1\),
the mesh contains the interface of air and water[21].

3. Energy loss analysis in distributor and nozzle

As the design theory of Pelton turbine, there is no air flow in the distributor and nozzle. The
flow with large velocity flow passing through the distributor and then become 6 jet flow outside
6 nozzles. The efficiency of the Pelton turbine is influenced by many parameters in which the
energy loss of nozzle is the most important one that can present nearly half of the loss. In this
section, unsteady simulation is performed from the inlet of distributor to the outlet of nozzle to
find out the loss distribution and mechanism aiming to understand the characteristics of Pelton
turbine. Two major key parameters are chosen to design the numerical experience. One is the
head which are chosen as 40m, 60m, 80m, 100m, 120m. The other is chosen as dimensionless nozzle
opening \(S\) which is defined by the fraction of spear opening distance \(l\) and spear diameter \(d\). \(S\) is chosen as 0.25, 0.4, 0.55, 0.7.

Firstly, head \(H\) is fixed at 80m with \(S\) changing. One part of the energy loss is from
the distributorand bifurcation tube. The loss is defined as the Eq.3 where energy loss is
dimensionless by the total head of 80m. Note that: \(P_{bifur_{in}}\) is the inlet total pressure and
\(P_{bifur_{out}}\) presents the total pressure of the bifurcation outlet.

\[
L = \frac{\Delta P}{\rho g H} = \frac{P_{bifur_{in}} - P_{bifur_{out}}}{\rho g H}
\]

(3)

The results of dimensionless loss of 6 bifurcation tube are shown in Figure.4. Globally, the
percentage of the loss is near to 0.1%. This is cause to the well design of the distributor which
make the flow inside more smooth. Although the loss percentage is not so high, the variance of
the each bifurcation tube is evident by comparing the loss of 6 bifurcation tubes.

As a consequence, the major energy loss should locate in the nozzle part which is shown in
Figure.5. The loss is defined as the Eq.4 where energy loss is dimensionless also by the total
head of 80m. Note that: \(P_{nozzle_{in}}\) is the nozzle inlet total pressure and \(P_{nozzle_{out}}\) presents the
total pressure of the nozzle outlet where the free jet occurs.
\[ L = \frac{\Delta P}{\rho g H} = \frac{P_{\text{nozzle in}} - P_{\text{nozzle out}}}{\rho g H} \quad (4) \]

The energy loss reaches 42% when \( S \) equals to 0.25. It is because that the nozzle is nearly close so the local resistance which can evidently induces the energy loss. When the \( S \) increases, the loss decreases because that the flow passing the nozzle become more smooth. The energy loss decreases to about to 22% when the \( S \) equals to 0.7. Secondly, comparing to the 6 different nozzles, the energy loss in same \( S \) is extremely approximate which indicates that the needle distance does not influence the energy loss.

**Figure 4.** Energy loss of each bifurcation. \( H \) is 80m.

**Figure 5.** Energy loss of each nozzle. \( H \) is 80m.

Besides the influence of \( S \), different head is another major parameter which can influence the energy loss. Five different effective heads are chosen as the variance as shown in Figure 6. For each case of different nozzle, the energy loss locate between 26.4 \% and 26.6\%. The difference is 0.2\% which could be the bifurcation tube structure is not designed perfectly uniform. Hence, the loss difference induced by the different head is nearly miniscule.

**Figure 6.** Energy loss of each nozzle. \( S \) is 0.55. Effective head varies from 40m to 120m.
4. Physic of bad-behaved flow in the runner region

For improving the efficiency of the Pelton turbine, torque of the bucket is the key point besides the energy loss of the distributer. Numerical simulation from distributer to the bucket herein performed an important role to analyze the torque characteristics of the bucket and so on the bad-behaved flow. As given in the previous research, there are three typical types of bad-behaved flow which can induce the complex flow phenomena and decrease the efficiency of the Pelton turbine. This section will analyze in detail the two negative flows and indicate the suggestion for the runner designing.

As the symmetrical structure of the bucket and nozzle system, every bucket has the same flow pattern. Torque is obtained as shown in Figure 7. The flow patterns of the 8 status are shown in Figure 8.

Figure 7. Torque of No.1 bucket through one complete circle of running.

Working process contains 6 periods which present the bucket passing through 6 jets of nozzle. In each period, working process is described in 8 status. Status I is the start-up process of the turbine, there is not yet the jet injection to the runner. Status II indicates that the jet starts to touch the No.1 bucket. Status III presents the normal working process where the precedent jet leaves the bucket. In Status IV, the water jet begins to flow into the edge of the bucket, torque increases until the jet fulfill the bucket which is the Status V. Status VI is the middle working process of the bucket that the jet completely covers the bucket meanwhile the torque increases to the maximum at Status VII. From this status, the jet starts to inject the next bucket, the outflow of next bucket will touch the back side of the No.1 bucket, then the torque will decrease from Status VII to Status VIII. Furthermore, how the three bad-behaved flow influence the torque will be explained in the next three parts.

4.1. Jet acting on the back of the bucket

As one of the bad-behaved flow in Pelton turbine, jet which acts the back of bucket introduces the negative torque to the turbine which can decrease the efficiency. As shown in Figure 9, No.1 bucket marked red is influenced by the jet from the previous bucket. The pressure contour of the bucket back side is shown in Figure 10. There are obviously some high pressure parts which caused by the jet flow which can induce the negative phenomenon to the efficiency.
Figure 8. Flow patterns of 8 different status which marked in Figure.7. Note that the green surface is the interface of water and air.

Figure 9. Flow characteristics of the bucket where red one is No.1 bucket. Note that the green body is the water calculated by VOF. T=0.1104s.

Figure 10. Pressure contour of the bucket backside. T=0.1104s.
4.2. Flow interaction with the cutting edge

The jet which is from the previous bucket cutting edge when act on the next bucket will induce the negative flow. Figure.11 shows that the torque is always below zero which decreases the total efficiency of the turbine.

![Torque of NO 1 edge](image)

**Figure 11.** Torque of cutting edge of the bucket induced by the flow.

The flow characteristics of T=0.112s is shown in Figure.12. It is clear that the outflow of the previous bucket from the cutting edge strongly influences the next bucket’s back side. This can be proved by the Figure.13. On the cutting edge area, the low pressure presents the high velocity region which induces the outflow and then induces the negative torque of the turbine. This phenomenon should be improved by adjusting the cutting edge pattern will be studied in the future research.

![Flow interaction by the outflow from the cutting edge of the previous bucket.](image)

**Figure 12.** Flow interaction by the outflow from the cutting edge of the previous bucket. Note that the green body is the water calculated by VOF. T=0.112s.

![Pressure contour of the bucket working side.](image)

**Figure 13.** Pressure contour of the bucket working side. T=0.112s.
5. Conclusion
In this paper, bad-behaved flow is well investigated by two types of simulations through the CFX code. First part calculates the distributor and the jet of nozzle which investigate the energy loss. The second part calculates the whole Pelton turbine including the bucket running. The two part corresponds to each other and obtain 3 conclusions.

1) There are two parameters which influence the energy loss and furthermore the efficiency. They are the head $H$, dimensionless nozzle opening $S$. Finally, the $S$ is approved as the sensitive parameter to the head loss up to 42%.

2) The negative flow is obtained by simulation. Two types of negative flow are found that can decreases the efficiency. One is the jet acting on the back side of the bucket. The other is the flow interaction with the cutting edge.

3) For avoiding the bad-behaved flow, cutting edge pattern would be improved and better designed.

6. Acknowledgement
This work was supported by “the Fundamental Research Funds for the Central Universities” (Grant No.HIT.NSRIF2017044) and “China Postdoctoral Science Foundation” (Grant No.2017M621270).

References
[1] Parkinson E, Vullioud G, Geppert L and Keck H 2002 International Journal on Hydropower and Dams 9 100–103
[2] Jeon H, Park J H, Shin Y and Choi M 2018 Renewable Energy 123 273–280
[3] Zhang Z and Casey M 2007 Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 221 1181–1192
[4] Zeng C, Xiao Y, Luo Y, Zhang J, Wang Z, Fan H and Ahn S H 2018 Renewable Energy 125 270–282
[5] Jo I C, Park J H, Kim J W, Shin Y and Chung J T 2016 Desalination and Water Treatment 57 24626–24635
[6] Zhang Z and Parkinson E 2002 Lda application and the dual-measurement-method in experimental investigations of the free surface jet at a model nozzle of a pelton turbine International Symposium on Applications of Laser Anemometry To Fluid Mechanics
[7] Zhang Z 2004 Experiments in Fluids 37 29–39
[8] Kvicinsky S 2008 Experimental and numerical analysis of surface flows in a rotating bucket Proceedings of the 21st IAHR Symposium on Hydraulic Machinery and Systems
[9] Pellone C, Maitre T and Leroy P 2006 Journal of Turbomachinery 128
[10] Benzon D, Zidonis A, Panagiotopoulos A, Aggidis G A, Anagnostopoulos J S and Papantonis D E 2015 Journal of Fluids Engineering 137 041106
[11] Fiereder R, Riemann S and Schilling R 2010 Numerical and experimental investigation of the 3d free surface flow in a model pelton turbine pp 012072
[12] Kvicinsky S, Kueny J L and Avellan F 2002 Numerical and experimental analysis of free surface flow in a 3d non rotating pelton bucket pp 32–45
[13] Perrig A 2007 Hydrodynamics of the free surface flow in pelton turbine buckets
[14] Zeng C J, Xiao Y X, Zhu W, Yao Y Y and Wang Z W 2015 Numerical simulation of cavitation flow characteristic on pelton turbine bucket surface p 042043
[15] Santolin A, Cavazzini G, Ardizzon G and Pavesi G 2009 Numerical investigation of the interaction between jet and bucket in a pelton turbine vol 223 pp 721–728
[16] Rossetti A, Pavesi G, Ardizzon G and Santolin A 2014 Journal of Fluids Engineering 136 081304
[17] Wei X Z, Yang K, Wang H J, Gong R Z and Li D Y 2015 Numerical investigation for one bad-behaved flow in a pelton turbine p 042033
[18] Vessaz C, Jahanbakhsh E and Avellan F 2015 Journal of Fluids Engineering 137
[19] Hoang M L, Verboven P, Baerdemaeker J D and Nicola B M 2000 International Journal of Refrigeration 23 127–140
[20] Hirt C W and Nichols B D 1981 Journal of Computational Physics 39 201–225
[21] Xiao Y X, Zeng C J, Zhang J, Yan Z G and Wang Z W 2013 Numerical analysis of the bucket surface roughness effects in pelton turbine pp 668–672