Numerical investigation of the jet velocity profile and its influence on the Pelton turbine performance

Takashi Kumashiro1*, Siamak Alimirzazadeh2, Audrey Maertens2, Ebrahim Jahanbakhsh2,3, Sebastián Leguizamón2, François Avellan2 and Kiyohito Tani1

1 Hydraulic R&D Laboratory, Hitachi Mitsubishi Hydro Corporation, Hitachi, Japan
2 Laboratory for Hydraulic Machines, École polytechnique fédérale de Lausanne, Lausanne, Switzerland
3 Institute of Computational Science, Università della Svizzera italiana, Lugano, Switzerland

kumashiro.takashi.ue@hm-hydro.com

Abstract. Numerical investigation of the unsteady flow patterns around the bucket can be helpful to improve the Pelton turbine efficiency. In fact, by studying the loss mechanisms in the flow, one can optimize the bucket design. For this purpose, an accurate investigation of the water jet is also necessary as an inlet condition for the bucket flow. The water jet ejected from the nozzle is actually non-uniform, due to the upstream turbulence and secondary flow in the distributor, bending pipes, needle and supports. This non-uniformity causes a deviation of the water jet from the ideal jet center, which is tangent to the jet circle, and it directly affects the flow patterns around the bucket. Whereas water splashing makes experimental observations very challenging, a numerical approach is effective and convenient to study the unsteady flow pattern in a Pelton turbine. In the present research, numerical analysis of the two-phase flow through the distributor, bending pipe and nozzle, has been performed. The jet velocity profile has been verified with experimental results, which are measured in the model test equipment, and good agreement is obtained for both velocity profile and jet deviation. In addition, the analysis of the bucket unsteady flow is performed using the aforementioned non-uniform jet velocity profile and the calculated flow pattern around the bucket is compared to the previous calculation using a uniform velocity profile at the inlet boundary. The difference between the two cases is discussed based on the quantitative and qualitative evaluation from the numerical analysis.

1. Introduction
The Pelton turbine bucket is one of the most complicated components of hydraulic turbines to optimize due to the complex unsteady behavior of the free surface flows resulting from the interaction between the impinging water jet and the rotating buckets. From the view point of fluid dynamics, the investigation of flow pattern around the bucket is a key issue for the bucket design optimization. As a first step, we need to investigate the mechanisms responsible for energy loss by performing accurate numerical flow simulations. This is very important to evaluate the amount of the losses. Then, it makes possible to discuss how to reduce the losses by modifying the design.

It is known that a non-uniform water jet is formed due to the upstream secondary flow and the distributed velocity from the bend, distributor, needle rods, supports and nozzles. The non-uniformity causes a deviation of the water jet from the ideal jet center, which is tangent to the jet circle, and it...
directly affects the flow patterns around the buckets. This sometimes causes a non-negligible drop in the turbine performance. As the performance of the machine is directly affected by this phenomenon, the non-uniform water jet in a Pelton turbine has been experimentally and numerically investigated for a long time [1, 2]. An attempt to reduce this bias in velocity distribution by changing the design of nozzle has also been reported [3]. Nowadays, for the design optimization, the use of computational fluid dynamics (CFD) simulations is an effective and suitable method. Since the optimization process requires a lot of trial-and-error, CFD simulations can significantly reduce the development time and cost compared to experiments. Moreover, in the case of a bucket design, CFD simulations are much more helpful than observations in the model tests because they give direct access to the full flow field, whereas, in the model test, the splashing water ejected from the bucket outlet edges makes the observation of the transient flow around a bucket very challenging. Thanks to recent improvement in access to computational resources, it is getting easier to perform numerical analysis of the two-phase unsteady flow and to utilize the results for the practical bucket design process [4]. In this paper, the non-uniformity of the water jet is investigated using both numerical and experimental approaches. As a numerical approach, CFD simulations have been performed for the stationary parts, which include the distributor, the bend, the nozzle, the needle and the water jet domain. The jet velocity profile extracted from the CFD results has then been verified with the model test results which have been measured by a traversing pitot tube [5]. The non-uniform jet velocity profile has been used as inlet condition for the water jet feeding the rotating buckets. The influence of the inlet profile on the unsteady flow pattern around a bucket and the turbine characteristic is discussed based on the comparison with the case of a uniform velocity inlet profile.

2. Specification
The case study is a vertical Pelton turbine featuring 6 nozzles. The components of the Pelton turbine are indicated in figure 1. Its specific speed is \( N_{Qh} = 0.0167 \) at maximum turbine output on the rated operation condition. Specific speed is defined at the maximum output according to the definition of \( N_{Qh} = n \frac{Q^{0.5}}{H^{0.75}} \) in IEC 60193-1999 [6]. Here, \( Q \) is the discharge of 1 nozzle. The reference diameter of the model runner is 0.45m and the number of bucket is 22. As a specific feature of the turbine, there are rods for needle operation and guides lying across the distributor. This study focuses on the 6th nozzle, which is the last one in the distributor, because the large angle of the bend at that nozzle makes it the most affected by secondary flow and non-uniformity of the water jet. All the CFD simulations have been performed using the model size to allow comparison with the model tests. The completely same geometry of the turbine has been used for both CFD simulations and the model test.

![Figure 1. Components of the Pelton turbine.](image-url)
3. Calculation setup

3.1. Calculation condition

For the numerical investigation of the flow, commonly used conventional grid-based analyses are performed with the commercial CFD software ANSYS CFX 18.0. The two-phase unsteady flow model, in which the water and the air are solved inhomogeneously, has been used to simulate the water jet and its interaction with the rotating buckets. Based on our previous research, the calculation has been verified with the experimental data and it is considered to be accurate enough to discuss the relative differences in the unsteady flow patterns and turbine characteristics [4].

On the other hand, particle-based methods (e.g. SPH, FVPM, etc.) have grown remarkably in the last few years. These methods are robust for studying of Pelton turbine flows with a complicated free surface and small water drops around the buckets. In a series of the study, in consideration of future possibilities to adapt these particle-based methods on a practical design, the grid-based calculation results introduced in this paper will be compared with one of the above mentioned particle-based methods, finite volume particle method (FVPM) which was originally developed by Jahanbakhsh in 2014 [7, 8, 9] and recently ported to GPU by Alimirzazadeh et al [10], in near future.

The calculation is performed at the best efficiency point measured in the model test under the 3-nozzle operation (2nd, 4th and 6th nozzle are in operation). In this study, the numerical calculations are divided into three different stages in order to allow the use of the mesh size required for accuracy while keeping the calculation time reasonable. One is the calculation for the full-distributor, the second one is for the last bend, the 6th nozzle and the water jet, and the last one is for the water jet and the rotating buckets. It is necessary to use a very fine grid around the outlet of the nozzle in order to capture the boundary layer with a good accuracy. If all the flow stages were calculated in one analysis, it takes long calculation time.

3.2. Calculation for the full-distributor

First of all, the flow inside the full-distributor is calculated. The aim of this calculation is to extract the velocity distribution, including upstream effect, for the inlet condition of the aforementioned second analysis. The calculation domain and boundary conditions is shown in figure 2. The 1st, 3rd and 5th nozzles are omitted to simplify the calculation and walls are set instead of nozzles. The discharge measured in the model test at this operation point is used at the inlet boundary of the distributor. The opening boundary condition is set to the outlet of all 3 nozzles.

![Figure 2. Calculation domain and boundary conditions of full-distributor.](image-url)
3.3. Calculation for the last bend, 6th nozzle and water jet
As a second stage, the flow inside the last bend and the 6th nozzle, and the water jet are calculated. In this calculation domain, there are the needle rods for the 5th and 6th nozzles, more than 90° bend and 4 needle supports. These structures generally affect the flow inside the nozzle and cause the jet to deviate from the ideal jet center. The calculation domain, its mesh and the boundary conditions are shown in figure 3. The velocity distribution extracted from the full-distributor analysis is used at the inlet boundary and the cylindrical opening boundary is set as the outlet condition. In order to achieve small enough y+ values to accurately capture the boundary layer, there are thin mesh layers around the outlet of the nozzle.

Figure 3. Calculation domain and boundary conditions of the last bend, 6th nozzle and jet.

3.4. Calculation for water jet and rotating buckets
Due to unsteady rotating Pelton turbine physics, a transient two-phase flow analysis is required to solve the flow around the buckets, which is computationally expensive. In addition, it is not possible to use the circumferential average boundary condition between the rotor and the stator domain in this case. It means that all the water jets and all the buckets should be contained in the analysis domain. This makes it challenging to apply the CFD to the practical bucket optimization process which needs a lot of trails. Here, a simplified analysis domain has been created to reduce the calculation time significantly.

The calculation domain of the simplified analysis and boundary conditions is shown in figure 4. The simplified analysis domain includes only 1 water jet out of 6 and 4 buckets out of 22. Periodic boundary conditions are applied to both rotor and stator domains. This setting does not represent exactly the real setting of the model test because the pitch angle between the nozzles is different. However, it can be applied to the normal operation point in which there is no interference with the next jet. In this way, the number of mesh elements is significantly reduced compared to the full-domain analysis and calculation time is reasonably reduced. The velocity profile extracted from the second stage is used at the inlet boundary of the water jet and the opening boundary is set around the
buckets. General grid interface (GGI) is applied to the rotor-stator interface. The symmetric boundary condition has been also applied to divide the calculation load by two.

4. Calculation result

4.1. Flow inside the distributor, bend and nozzle

The pressure and velocity contours in the center plane of the turbine are shown in figure 5. The extracted velocity contour and vector to be used as inlet for the second calculation is shown in figure 6. Flow separation behind the needle rods and at the branching points of the distributor can be observed. This excites the secondary flow formation which is clearly seen in figure 6. Furthermore, it is found that the difference of the discharge between all the three nozzles is less than 0.3%. Specifically, the ratio of the discharge is 0.3348 for the 2nd nozzle, 0.3324 for the 4th nozzle and 0.3328 for the 6th nozzle.

The velocity contour in the center plane of the turbine from the calculation result of the last bend, the 6th nozzle and the jet are shown in figure 7. Additionally, velocity contours in the sections A, B, C and D, whose locations are indicated in figure 7, are shown in figure 8. The flow is separated at the
branch of the 5th nozzle and the main stream is deviated to the outer side of the distributor. At the same time, the secondary flow is increased when the flow goes around the needle rod of the 5th nozzle. Through the last bend, the secondary flow which is circulating is obviously confirmed in the section A, B and C, and the velocity deviation between the inner and outer side is gradually getting larger toward the downstream. On the section D, there is a clear low velocity region at the inner side and high velocity region at the outer side and the flow does not mix due to the supports which partition the channel. Since the flow accelerates toward the outlet of the nozzle, there is almost no chance to smooth the deviation before the ejection.

![Image](image1.png)

**Figure 7.** Velocity contour on the center section.

![Image](image2.png)

**Figure 8.** Velocity contour on the plane A, B, C and D. (All the sections are observed from the downstream side.)

4.2. **Comparison with the model test measurement**

The equipment for measuring the jet velocity in the model test is shown in figure 9. A pitot tube is set up to go across the water jet which comes from the 6th nozzle with 1 mm pitch in both horizontal and perpendicular directions. The jet velocity profiles are also extracted from the CFD results in order to compare with the experimental data. These are normalized by the maximum velocity in the experiment and are shown in figure 10. Here, the CFD velocity of the cells with a water volume fraction above a threshold is specifically extracted. The threshold is chosen such that the discharge corresponds to one third of the measured discharge in the model test. There is almost no deviation of the jet in perpendicular direction for both the CFD and the experiment. On the other hand, the jet is deviated in the horizontal direction toward the inner side of the turbine. Both the CFD and the experiment have this same trend of deviation, and the actual jet centers are quantitatively evaluated by weighted average of the velocity distribution. The jet deviations are defined based on the difference between the actual jet center and the ideal center which is tangent to the jet circle. As a result, the amount of the jet deviation on the velocity measurement section is about 1.2mm for the CFD and about 0.5mm for the model test. Therefore, the jet deviation is slightly overestimated in the CFD but it is in a good agreement with the model test. The evaluated jet centers are also shown in figure 10. However the calculated velocity is approximately 4% smaller than the experimentally measured velocity. This difference is attributed to difference in head between the calculation and the model test. It directly causes the velocity difference as there is a relation $c_{jet} = c_v \sqrt{2gH}$. Here, $c_{jet}$ is jet velocity, $H$ is head and $c_v$ is coefficient.

The sectional streamlines colored by their velocity, for the cells with volume fraction above the threshold, is shown in figure 11. The non-uniformity of the jet due to the secondary flow and the deviation of the velocity, which are originally caused in the upstream on the distributor, the bend and the nozzle, are confirmed in the horizontal direction. It causes the water jet to deviate from the ideal jet center. The deviation can be also qualitatively confirmed from the comparison of the jet between the CFD and the experiment shown in figure 12.
4.3. Unsteady flow pattern around the buckets

In order to investigate the influence of the non-uniform water jet, the computed jet velocity profile is given as the inlet boundary condition for the rotating buckets simulation. As a reference, the computation with the same discharge and a uniform velocity profile, using measured maximum velocity and adjusted jet diameter, has been also performed. The torque time history of a bucket is shown in figure 13. The pressure coefficient distribution of the bucket inner surface at the rotation angle 60°, 68°, 87° and 104° for both conditions are shown in figure 14. It is found that there are a small peak at around 60° and a drop at around 68° in the torque time history in the case of the non-uniform velocity, and the pressure distributions are clearly different from the other cases at those timings. In addition, the maximum torque in the case with the non-uniform velocity is smaller than the other one. However, at the last stage of the receiving the jet, that relation is reversed. These features are actually reasonable because the jet is deviated to inner side from the ideal center as described above. This can cause the flow disturbance when a bucket goes across the jet, and the torque increase after 90° because the jet will hit on a more inner, which means more ideal on this rotation angle, position of the bucket. The turbine efficiency is calculated from the time-averaged summation torque of several buckets. The difference in the calculated efficiency between the two cases is about 2.8%.
5. Conclusion
In this research, the unsteady flow pattern around the buckets has been investigated in more detail, by feeding the buckets with a non-uniform water jet. The CFD simulation is performed by dividing the simulations process into three different stages, the calculation for i) The full-distributor, ii) The last bend, the 6th nozzle and the water jet and, iii) The water jet feeding and rotating buckets. The calculated jet velocity is compared with experimental measurement results on a model turbine whose geometry is exactly the same as the geometry of the calculation. The jet deviation toward the inner side has been evaluated quantitatively in both CFD and model test and the results show a good agreement. Furthermore, the influence of the non-uniformity of the water jet on the unsteady flow pattern and the turbine characteristic has been investigated based on the CFD giving the non-uniform velocity profile as inlet condition of the water jet. Consequently, the reduction of the turbine efficiency and the difference in the flow pattern are obviously confirmed compared to the case with uniform inlet velocity.
Acknowledgement
This work was done in the context of a collaboration between the EPFL Laboratory for Hydraulic
Machines (LMH) and Hitachi Mitsubishi Hydro Corporation.

References
[1] Parkinson E, Garcin H, Vullioud G, Zhang Z, Muggli F and Casartelli E 2002 Experimental and
umerical investigations of the free jet flow at a model nozzle of a Pelton turbine Proc. 21st
IAHR Symp. Hydraulic Machinery and Systems (Lausanne, Switzerland) Paper No. 43.
[2] Patel K, Patel B, Yadav M and Foggia T 2010 Development of Pelton turbine using numerical
simulation Proc. 25th IAHR Symp. on Hydraulic Machinery and Systems (Timisoara,
Romania) Earth and Environmental Science 12 (2010) 012048.
[3] Mack R, Gola B, Smertnig M, Wittwer B and Meusburger P 2014 Modernization of vertical
Pelton turbines with the help of CFD and model testing Proc. 27th IAHR Symp. Hydraulic
Machinery and Systems (Montreal, Canada) Paper No. 1-1-2.
[4] Kumashiro T, Fukuhara H and Tani K 2016 Unsteady CFD simulation for bucket design
optimization of Pelton turbine runner Proc. 28th IAHR Symp. Hydraulic Machinery and
Systems (Grenoble, France) Paper No. 143.
[5] Fukuhara H, Kumashiro T and Tani K 2017 Numerical Prediction of Water Jet Deviation
Contributed by Bending Distributor in Pelton Turbines Proc. 14th AICFM Asian
International Conference on Fluid Machinery (Zhenjiang, China) Paper No. 239.
[6] International Electrotechnical Commision (IEC) 60193 1999 Hydraulic turbines, storage pumps
and pump-turbines – Model acceptance tests
[7] Jahanbakhsh E 2014 Simulation of silt erosion using particle-based methods EPFL Thesis.
[8] Jahanbakhsh E, Vessaz C, Maertens A and Avellan F 2016 Development of a Finite Volume
Particle Method for 3-D fluid flow simulations Computer Methods in Applied Mechanics and
Engineering, vol. 298, pp. 80–107.
[9] Jahanbakhsh E, Maertens A, Quinlan N J, Vessaz C and Avellan F 2017 Exact finite volume
particle method with spherical-support kernels Computer Methods in Applied Mechanics and
Engineering, vol. 317, pp. 102–127.
[10] Alimirzazadeh S, Jahanbakhsh E, Maertens A, Leguizamon S, Avellan F 2017 A GPU-
accelerated versatile solver based on the finite volume particle method 12th International
SPHERIC Workshop (Ourense, Spain)