Application of particle-based numerical analysis to the practical design of Pelton turbine

Takashi Kumashiro1*, Siamak Alimirzazadeh2, François Avellan2, and Kiyohito Tani1

1Hydraulic R&D Laboratory, Hitachi Mitsubishi Hydro Corporation, Hitachi, Japan
2Laboratory for Hydraulic Machines (LMH), Ecole polytechnique fédérale de Lausanne (EPFL), Lausanne, Switzerland

kumashiro.takashi.ue@hm-hydro.com

Abstract. For the design optimization of the Pelton turbine, it is highly demanded to investigate the flow inside the turbine casing, which includes the water jet from the nozzles, the interaction between the jet and rotating runner, and the flow ejected from the bucket outlet, in various operating ranges. The behavior of the flow, however, is very complicated due to its unsteadiness and free surface. Further, the experimental observation in the model test is challenging because of the high time resolution requirement and the obstruction by the splashing water inside the turbine housing. In this regard, the numerical analysis is considered as a powerful method to approach the flow behavior in the Pelton turbine. Conventionally, the grid-based numerical analysis is applied to the calculation of the flow for its practical design. However, with the grid-based methods, a huge amount of fine elements is required to capture the unsteady behavior of the water free surface and the tiny splashing water particles. Increasing the number of elements directly results in higher computational costs, which makes it difficult to consider the splashing water in the process of practical design. From this point of view, the GPU-accelerated finite volume particle method is applied to the investigation of the flow in this research. It is firstly confirmed that the particle-based numerical result has a good agreement with the experimental result by the comparison of the turbine characteristics. Furthermore, several evaluations of the flow based on the analysis for the practical design are introduced in the paper.

1. Introduction

The Pelton runner is one of the most complicated components of hydraulic turbines to optimize its design due to the complex unsteady behavior of the free surface flows resulting from the interaction between the impinging water jet and the rotating buckets. One needs to evaluate the energy loss of the flow around the runner and understand its causes for design optimization. For this purpose, it is a key issue to investigate the flow pattern around the bucket, accurately. The flow observation in the model test is a challenging task due to both the high time resolution requirement and the splashing water inside the turbine casing. In this regard, the numerical analysis is considered as a powerful method to analyze the flow behavior in the Pelton turbine because it provides direct detailed information of the full flow field.

Conventionally, the grid-based Finite Volume Method (FVM) numerical analysis is applied to the calculation of the flow for practical designs [1, 2]. However, the grid-based method requires a large...
number of elements to be able to capture the unsteady behavior of the water free surface and the tiny splashing water drops ejected from the buckets. Increasing the number of the elements is directly linked to the computational cost, which can lead to costly computational analysis for this two-phase transient flow problem. Therefore, the interaction between the jet and the rotating bucket is mainly focused on the practical design process, which needs a lot of trial-and-error on the runner design, and this causes a difficulty to consider small water splash.

Unlike the grid-based methods, the particle-based methods are appropriate and robust in handling free surface problems with moving boundaries such as Pelton runner flow. Different particle-based methods, such as Smoothed Particle Hydrodynamics (SPH) have been suggested during the last half-century and are continuously researched for its improvement nowadays. In this research, the Finite Volume Particle Method (FVPM) is used to simulate the Pelton turbine flow, which is basically an Arbitrary Lagrangian-Eulerian (ALE) method but also includes many of the attractive features of conventional grid-based methods. The solver is called SPHEROS and is able to simulate the interaction between fluids, solid and silt [3]. SPHEROS runs on the Central Processor Unit (CPU), while an accelerated version has also been developed running on Graphics Processing Units (GPUs) [4].

In this paper, both the particle-based and conventional grid-based methods are used to calculate the flow around a Pelton runner, and the computed torque and turbine efficiency are compared to the experimental model test results. The numerical analysis is performed on several different operating points to confirm the turbine characteristics. The bucket design is then improved based on the detailed flow information obtained by particle-based simulations. Thanks to the power of GPUs for parallel computation, the computational speed is significantly accelerated. In addition, the flow inside turbine housing, which is hard to capture by the grid-based method, is simulated with FVPM and the effect of splashing water by the housing wall is investigated.

2. Specifications
The targeted hydraulic turbine in this study is a six-nozzle vertical Pelton runner with the specific speed of \( N_{QE} = 0.0167 \) running at the maximum turbine output. The specific speed is defined at the maximum output as \( N_{QE} = n Q^0.5 / E^{0.75} \) in IEC 60193-2019 [5] in which \( n \) is the runner rotational frequency, \( Q \) is the rated discharge of one nozzle and \( E \) is the specific energy. The runner reference diameter \( D_1 \) is 0.45 m which is in model scale and the number of buckets is 22. All the numerical simulations are performed in the model scale to allow direct comparison with the model test results. The runner model installed in a six-nozzle vertical Pelton turbine housing and the injected water jet from the nozzle are shown in Figures 1 and 2, respectively.

![Figure 1. Model of 6-nozzles vertical Pelton turbine.](image1)

![Figure 2. Injected water jet in model test.](image2)
3. Calculation setup

3.1. Calculation points
Both the rotational speed and the discharge of all the operating points are exactly the same as the measured values in the model test. The jet velocity is measured by a pitot-tube [6], and the jet diameter is then determined from the velocity and the discharge. The simulations are performed on four different discharge points, while the rotational speed remains constant to evaluate the trend in the turbine characteristics for the different turbine discharge. The operating points are shown in Table 1 in which the discharge factor $Q_{ED}$ is for one nozzle and the speed factor is for the best efficiency point (BEP).

| Calculation point No. | 1  | 2  | 3  | 4  |
|-----------------------|----|----|----|----|
| Needle stroke (mm)    | 21 | 17.5 | 14 | 10.5 |
| Speed factor $n_{ED}$ (-) | 0.2043 |
| Discharge factor per 1 nozzle $Q_{ED}$ (-) | 0.006957 | 0.006249 | 0.005362 | 0.004287 |
| Jet diameter $D_2$ (mm) | 35.8 | 33.9 | 31.4 | 28.1 |

3.2. Calculation condition
For the particle-based numerical analysis (with SPHEROS), an inlet boundary and three buckets are included in the calculation domain. Both the water and wall in the domain are represented by the uniform-size particles. The simulation setup is simplified with only three buckets to take the advantage of runner periodicity while covering the interaction between the water jet and the leading (1st) and following (3rd) bucket which affects the flow behavior around the intermediate (2nd) bucket. The transient flow behavior of the 2nd bucket is investigated and used for the torque computation. The water jet is injected from the inlet boundary, and the jet velocity and diameter are extracted from the model test results. The buckets rotate on a given rotational speed and go across the water jet once. The calculation setup and the sample of the initial particle distribution on the inlet boundary are shown in Figures 3 and 4, respectively.

![Figure 3. Calculation setup of SPHEROS](image)

![Figure 4. Particle distribution on inlet boundary](image)
The grid-based simulation is performed using ANSYS CFX 18.0 commercial solver to compare with the particle-based method. The two-phase VOF model has been used to simulate the air-water interaction. Based on our previous research, the calculation has been verified with the experimental results and it is considered to be accurate enough to discuss the relative differences in the transient flow patterns and turbine characteristics [2]. The total number of the elements is about 4 million and this is adopted to keep balance between the calculation load and the accuracy for the practical design. Both particle-based and grid-based analyses are performed on the same operating point and same bucket geometry. Regarding the inlet boundary, it was known that there is a deviation in the jet velocity based on the model test measurement, in which its impact on the torque was also studied in past research [7, 8]. However, since the jet deviation is different for each nozzle and needle stroke, for the present study, a uniform velocity is applied to the inlet boundary for the equivalent comparison.

3.3. Spatial resolution
The accuracy of the particle-based method is depending on spatial resolution, i.e. the particle size [9]. However, since the computational cost is significantly increased with refining the particle size, the spatial resolution for the current case is chosen in a way to make a reasonable balance between the accuracy and computation time. The survey is performed for four different resolutions $D_2/X_{ref} = 10, 20, 30$ and $40$ on the operating point No. 2 which is the closest point to the BEP. In the aforementioned expression, $D_2$ is the jet diameter and $X_{ref}$ is the distance between the adjacent particles. The computed torque time-histories and the comparison with the measured torque in the model test are shown in Figures 5 and 6, respectively.

![Figure 5. Torque time-histories for different spatial resolutions.](image1)

![Figure 6. Comparison with model test result.](image2)

As shown in Figure 5, the torque time-history is changed with the spatial resolution, especially when the jet first interacts with the bucket (rotation angle $\theta$ around 55-65 deg) as well as torque peak (rotation angle $\theta$ around 75-90 deg). Since, the torque time-history is almost converged from $D_2/X_{ref} = 30$ to $40$. Therefore, the simulations are performed with $D_2/X_{ref} = 30$ for the practical design of a bucket with a reasonable computational load.

4. Calculation results
4.1. Turbine torque and efficiency
The torque time-histories of the concerned 2nd bucket on the calculation point No. 2 is shown in Figure 7 as a representative calculation result and compared to the result of the grid-based method. The trends
are highly similar for both methods although there is a slight difference around the rotation angles $\theta = 60$, 70 and 100 deg. A small torque peak is shown around the rotation angle $\theta = 60$ deg in the particle-based result which is not seen in the grid-based computed torque.

Since in the real Pelton runner, all the buckets have the same periodic torque behavior, the calculated concerned 2nd bucket torque time-history has been periodically shifted and copied in time direction to compute the overall torque. The same procedure is applied for the grid-based method and the bucket torque time-history, as well as overall torque, are shown in Figure 8. The time-averaged torque computed by both methods is obtained based on the overall torque and compared to the model test result.

![Figure 7](image1.png)

**Figure 7.** The bucket torque time-history. Particle-based vs. grid-based method.

![Figure 8](image2.png)

**Figure 8.** The overall numerical torque time-history vs. the model test results.

![Figure 9](image3.png)

**Figure 9.** Turbine efficiency curve calculated from the time-averaged overall torque.

The turbine characteristic curve calculated based on the time-averaged overall torque of four operating points No. 1-4 is shown in Figure 9. Although both methods underestimate the turbine efficiency compared to the experimental measurement, both curves are in good agreement with the experimental result on their trend and while the particle-based method provides more accurate results than the grid-based method.
4.2. Flow behavior around buckets

The flow around buckets is shown in Figure 10 for experimental observation as well as grid-based and particle-based simulations. Due to splashing water, it is difficult to clearly observe the flow behavior around the buckets in the model test. Unlike the experiment, the flow detailed data can be directly accessed with the numerical analysis that is an important advantage in the bucket design process. However, the VOF method which is used to simulate the interphase in the grid-based solver is diffusive and requires an ultra-fine grid to capture the water splash. On the other hand, in the particle-based method, the small water drops and thin water sheets are well-captured thanks to its Lagrangian nature. The transient flow behavior around the buckets is shown in Figure 11.

![Figure 10. Comparison of flow around buckets. (a) The photo was taken from the outside of the turbine housing. (b) Iso-surface of water volume fraction (0.85) computed with VOF and grid-based method. (c) And smoothed free surface water sheet reconstructed based on the particle-based simulation.](image)

4.3. Applying the particle-based method to bucket design

Supported by useful information obtained from the particle-based numerical analysis, the bucket design can be further improved. As an example of evaluation, the torque time-history of the improved bucket and the pressure distribution on the outer surface of the bucket are shown in Figures 12 and 13, respectively.

In the torque time-history of improved design, there is no negative torque peak when the water jet impacts the bucket (rotation angle $\theta$ around 55 deg). In addition, since the torque peak occurs around $\theta = 90$ deg, the bucket can generate the torque effectively. These design modifications definitely contribute to improving turbine efficiency.

Furthermore, since the particle-based method provides detailed flow information ejected from the edge of the bucket, it can be confirmed whether the water particles hit to the outer surface of the next bucket or not, to design a suitable outlet angle of the bucket. It is found that the pressure on the outer surface for improved design is less than the pressure of the original one because of less hitting water which reduces the negative torque which also contributes to higher turbine efficiency.
Figure 11. Transient flow behavior around the buckets. The water jet impacts the 2\textsuperscript{nd} bucket in rotation angle $\theta = 50$ deg. The water enters and spreads along the bucket inner surface and reach to the edge of bucket in $\theta = 70$ deg and is completely interrupted by the 3\textsuperscript{rd} bucket in $\theta = 80$ deg.
4.4. Calculation speed check

SPHEORS code was ported to GPU from CPU by Alimirzazadeh et al. in 2017 to speed up the computations [10]. Thanks to GPUs parallel computing power, the calculation speed of GPU-SPHEROS is accelerated approximately six times compared to the CPU version for the other cases [10]. The acceleration of the calculation for the current simulations is also evaluated and the comparison of the computing time on calculation point No. 2 is shown in Table 2. The calculations have been carried out on NVIDIA® Tesla P100-SXM2-16 GB GPU with GP100 Pascal architecture and a dual CPU node equipped with two Broadwell Intel® Xeon® E5-2690 v4 CPUs with 32 total physical cores.

The GPU version is almost 5 times speedup can be achieved in this case GPU-SPHEROS calculation cost is approximately the same as CFX, which makes it possible to apply GPU-SPHEROS to the practical design of bucket now.

Table 2. Calculation time.

|                        | CPU-SPHEROS | GPU-SPHEROS |
|------------------------|-------------|-------------|
| Physical time in simulation (s) | 0.04        | 0.04        |
| Calculation time (hours)   | 125.4       | 25.4        |
| Speed up factor (-)        | 1           | 4.9         |

4.5. Flow inside the turbine housing

The GPU-accelerated version of SPHEROS provides the capability to perform heavy-loaded computations such as the calculation of the flow inside the turbine housing and not only the flow around buckets. As an introduction, the interaction between the flow ejected from buckets and the housing wall is shown in Figure 14. This calculation includes a part of the housing wall in the calculation domain to evaluate the influence of the wall on the flow. The water particles ejected upward from the bucket edge is hitting to the housing wall and going along with the wall. The simulation is helpful to design an optimized housing wall shape which induces the flow going outside smoothly and effectively and also down-sizing the design of the turbine housing to reduce the construction cost of the hydraulic power plant.
5. Conclusion
In this research, the flow around Pelton runner using SPHEROS as a particle-based solver, and the computed torque and turbine efficiency were compared to both model test and grid-based ANSYS CFX results. The analysis was performed for several different operating points to investigate the turbine characteristics. As a result, the turbine efficiency curve calculated with the particle-based method is in a good agreement in its trend with the experimental result, and its accuracy was at least at the same level as the grid-based method. The particle-based method is more robust in capturing the unsteady water flow behavior compared to the grid-based methods it is less diffusive for free surface simulations. Thanks to the obtained detailed flow information around the buckets, the bucket design was improved. The GPU-SPHEROS computational performance for this case is almost the same as CFX and it is fast enough to apply it for the practical bucket design.

Acknowledgment
This research has been done in the context of a collaboration between the EPFL Laboratory for Hydraulic Machines (LMH) and Hitachi Mitsubishi Hydro Corporation.
References

[1] 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.

[2] 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.

[3] Jahanbakhsh E 2014 Simulation of silt erosion using particle-based methods EPFL Thesis.

[4] Alimirzazadeh S 2019 GPU-Accelerated Finite Volume Particle Simulation of Free Jet Deviation by Multi-jet Rotating Pelton Runner EPFL Thesis.

[5] International Electrotechnical Commission (IEC) 60193 2019 Hydraulic turbines, storage pumps and pump-turbines – Model acceptance tests

[6] 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.

[7] Parkinson E, Garcin H, Vullioud G, Zhang Z, Muggli F and Casartelli E 2002 Experimental and numerical 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.

[8] Kumashiro T, Alimirzazadeh S, Maertens A, Jahanbakhsh E, Leguizamón E, Avellan F and Tani K 2018 Numerical investigation of the jet velocity profile and its influence on the Pelton turbine performance Proc. 29th IAHR Symp. Hydraulic Machinery and Systems (Kyoto, Japan) Paper No. 107.

[9] Vessaz C, Jahanbakhsh E and Avellan F 2014 Flow simulation of a Pelton bucket using finite volume particle method Proc. 27th IAHR Symp. Hydraulic Machinery & Systems (Montreal, Canada) Paper No. 1-1-3.

[10] Alimirzazadeh S, Jahanbakhsh E, Maertens A, Leguizamón S and Avellan F GPU-accelerated 3-D Finite Volume Particle Method Computers and Fluids 171 (2018) 79–93