

# Unsteady CFD simulation for bucket design optimization of Pelton turbine runner

**Takashi KUMASHIRO, Haruki FUKUHARA and Kiyohito TANI**

Basic Engineering / Hydraulic Laboratory, Hitachi Mitsubishi Hydro Corporation,  
3-2-1 Saiwai, Hitachi, Ibaraki, Japan

Email: kumashiro.takashi.ue@hm-hydro.com

**Abstract.** To investigate flow patterns on the bucket of Pelton turbine runners is one of the important issues to improve the turbine performance. By studying the mechanism of loss generation on the flow around the bucket, it becomes possible to optimize the design of inner and outer bucket shape. For making it into study, computational fluid dynamics (CFD) is quite an effective method. It is normally used to simulate the flow in turbines and to expect the turbine performances in the development for many kind of water turbine including Pelton type. Especially in the bucket development, the numerical investigations are more useful than observations and measurements obtained in the model test to understand the transient flow patterns. In this paper, a numerical study on two different design buckets is introduced. The simplified analysis domain with consideration for reduction of computational load is also introduced. Furthermore the model tests of two buckets are also performed by using the same test equipment. As the results of the model test, a difference of turbine efficiency is clearly confirmed. The trend of calculated efficiencies on both buckets agrees with the experiment. To investigate the causes of that, the difference of unsteady flow patterns between two buckets is discussed based on the results of numerical analysis.

## 1. Introduction

The bucket of Pelton turbine runner is one of the most difficult components to optimize its design due to complicated unsteady behavior of the free surface flow during the interaction between jet and rotating bucket. In addition, the energy losses are caused by several different processes and it is also hard to capture. For bucket design, both hydraulic efficiency and structural strength must be considered because the high water head causes repetitive high stresses due to impact of water jet. Furthermore, the prototype manufacturing method is also an important factor which must be considered because the shape of the bucket is definitely more complicated than that of the other types.

From the view point of fluid dynamics, the investigation of flow pattern around the bucket is a key issue in order to accomplish the improvement of turbine performance and the design optimization. The mechanism of energy loss generation can be discussed by understanding this unsteady flow pattern around the bucket. And it also becomes possible to reduce the flow-induced loss and optimize the bucket design by studying the relationship between the mechanism of energy loss generation and the design parameters. There are several types of energy loss generation processes during the interaction between jet and rotating bucket. The representative examples are the angular momentum loss, wall friction loss, turning loss and collision loss. The total loss on the bucket is defined by a superposition



of these several aforementioned types of energy losses and it should be minimized on the optimized bucket. The amount of each component of energy losses is closely related to each design parameters e.g. inlet angle, outlet angle, curvature of inner bucket surface, bucket width, length and depth, the shape of edge, splitter, cutout and so on. However, since these design parameters are mutually affected by each other, it is not so easy to reduce each loss independently by changing several design parameters simultaneously. This means that there are some kind of design trade-offs for each types of loss. Therefore, it is necessary to survey the best combination of design parameters which can reduce the total energy loss in the bucket for the case one aims to optimize the geometry. For this purpose, the design optimization method and definition method of bucket are actively researched recently [1].

On the other hand, computational fluid dynamics (CFD) simulation is quite effective, suitable and widely used method to optimize the design of water turbines including Pelton type. As it requires a lot of trial-and-error procedure in the design optimization process, CFD simulation can significantly save the development time and cost compared to the model tests. Furthermore, in the case of bucket design, CFD simulation is more useful and powerful than observations and measurements obtained in the model test to understand the unsteady flow patterns around the bucket. Because it is difficult to observe the flow around the rotating bucket in the model test due to the splashing water ejected from the bucket outlet edge and flow measurement is also hard due to the flow unsteadiness although some new measurement techniques are improving recently [2]. In addition, thanks to the recent improvement of computational resources, it is getting easier to perform two-phase unsteady flow numerical analysis and to take it into the practical bucket design process.

On the basis of the above considerations, a numerical investigation for two different design buckets is introduced as a basic study for the bucket design optimization in the present research. The model tests of the two buckets are also performed to verify the numerical results and evaluate the accuracy which is important for CFD-based optimization procedure.

## 2. Specification and CFD simulation setup in this study

### 2.1. Specification

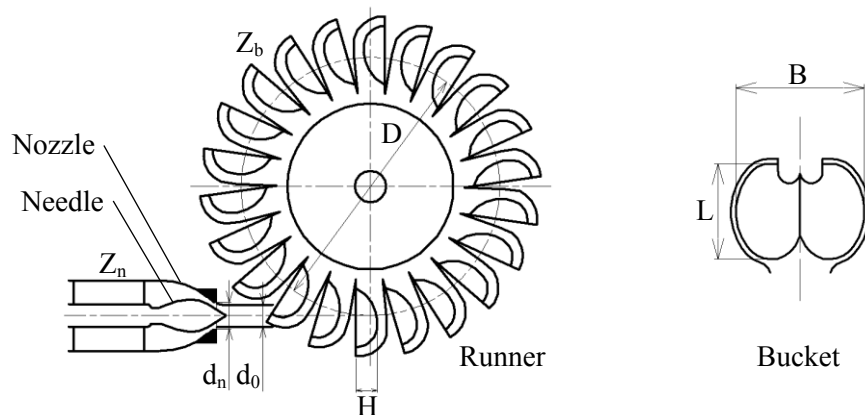
The investigation object in this study is a 6-nozzle vertical Pelton turbine with the specific speed of  $N_{QE}=0.0167$  at the maximum turbine output on the rated operation condition. The reference diameter of runner is 0.45m and the number of buckets is 22. CFD simulation is performed in the model size to compare with the model test. The model specifications of two buckets of A and B are shown in table 1. The locations of each dimension are also indicated in figure 1.

**Table 1.** Model specifications.

Bucket name			A	B
Reference diameter	D	(m)	0.45	0.45
Bucket width <sup>a</sup>	B	(-)	$3.6d_0$	$3.3d_0$
Bucket length <sup>a</sup>	L	(-)	$3.2d_0$	$2.8d_0$
Bucket depth <sup>a</sup>	H	(-)	$1.05d_0$	$1.05d_0$
Nozzle outlet diameter <sup>a</sup>	$d_n$	(-)	$1.3d_0$	$1.3d_0$
Number of bucket	$Z_b$	(-)	22	22
Number of nozzle	$Z_n$	(-)	6	6
Specific speed <sup>b</sup>	$N_{QE}$	(-)	0.0167	0.0167

<sup>a</sup>  $d_0$  is the expected jet diameter at the maximum discharge.

<sup>b</sup> Specific speed is defined at the maximum output according to the definition of IEC 60193-1999  $N_{QE}=nQ^{0.5}/E^{0.75}$ . Here, Q is the discharge of 1 nozzle.



**Figure 1.** Locations of each dimension.

## 2.2. CFD simulation setup

For the numerical investigation of water flow around the bucket, grid-based unsteady two-phase flow analyses are carried out using the commercial CFD software ANSYS CFX 16.1. As some particle-based simulation methods are grown remarkably recently [3, 4] and are often applied to a study of Pelton turbine, the comparison of analysis results on each method attracts attentions in these days.

The turbulence of flow is taken into account to use the standard  $k-\varepsilon$  turbulence model. The water and the air are solved inhomogeneously in these analyses. The calculation conditions are shown in table 2. CFD simulations are performed on 4 different needle strokes for both buckets A and B to investigate the characteristics for discharge change. These calculation points include the maximum discharge which corresponds to the maximum output. The rotation angle of runner per 1 time step is set as 1.0227degrees to take 16 time steps for 1 pitch angle of bucket  $360/22=16.3636$ degrees.

**Table 2.** Calculation conditions.

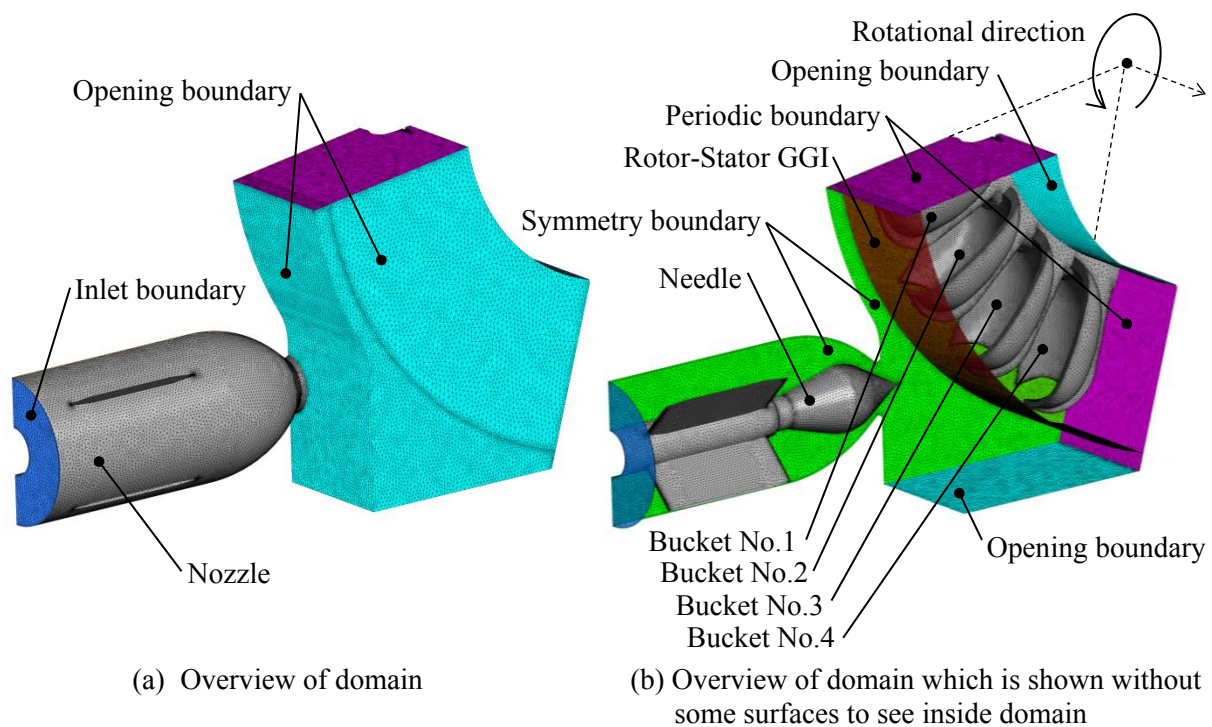
Needle stroke	(mm)	24.5	19	14	10	
Speed factor	$n_{ED}$	(-)	0.2068			
Discharge factor	$Q_{ED}$	(-)	0.0449	0.0390	0.0320	0.0249

The calculation load of CFD simulation for the water jet and the bucket is generally quite high because it takes much time to solve the time evolution of two-phase unsteady flow. Furthermore, due to its unsteadiness, the average boundary condition for the circumferential direction between rotor and stator domain, which is commonly used in numerical studies for turbo machinery, cannot be applied to this case. Therefore, all water jets and all buckets, or at least the lowest number of water jets and buckets which are divided by the largest common factor, should be contained in the analysis domain. Since the feature of this analysis definitely contributes to increase the number of mesh elements and calculation time, it has a difficulty to apply to the practical design optimization which needs a lot of trials. In this study, in order to solve this problem, a new and unique attempt is performed by using simplified analysis domain which can reduce the calculation load significantly.

The mesh of simplified analysis domain and its boundary conditions are shown in figure 2. The approximate number of mesh elements for the simplified analysis domain is shown in table 3. The unstructured mesh which consists of tetra and prism mesh is applied to this study. The prism mesh is put on the inner surface of bucket and the wall surface of nozzle. The simplified analysis domain includes only 1 nozzle out of 6 nozzles and only 4 buckets out of 22 buckets. The periodic boundary conditions are applied to the circumferential direction for the both rotor and stator domains. This setting cannot describe completely same situation to the real setting of the model test because the pitch

angle of nozzle is different. Since the difference causes that the timing of next water jet inflow is delayed, it should not be applied to the calculation points which are far from the normal operating range and have a jet interference phenomenon between the existing water and the next water jet inflow on the inner surface of bucket. However, this setting can be applied to the calculation point in the normal operation range which is not almost affected by the jet interference. And it is quite helpful to capture the flow patterns with a low calculation cost. As a result of adopting the simplified analysis domain, the number of mesh elements is simply reduced more than 80% for the both rotor and stator domains compared to the full analysis domain which includes all jets and all buckets.

As inlet boundary condition, a uniform water velocity is given to the inlet of nozzle. Non-uniformity of velocity which is caused by the upstream secondary flow in the distributor is not taken into account for this case although it is well known that the water jet is deformed due to the non-uniform velocity distribution [5] and it sometimes causes non-negligible turbine performance decrease in the hydraulic power stations. The attempt to reduce this bias of velocity distribution by changing the design of nozzle was also reported [6]. However, in this study, it is worth giving the water jet as an outflow of nozzle to compare the flow patterns around two different bucket geometries. An opening boundary condition which permits free inflow and outflow and maintains static pressure of 0Pa on a surface is adopted for the region around the bucket. A symmetry boundary condition is also applied to the center section of bucket and nozzle with consideration to reduce the calculation load. For the rotor-stator boundary, the general grid interface (GGI) is adopted to transfer physical quantity across the boundary. Smooth wall boundary condition is applied to the other wall surfaces.



**Figure 2.** Mesh of simplified analysis domain and its boundary conditions.

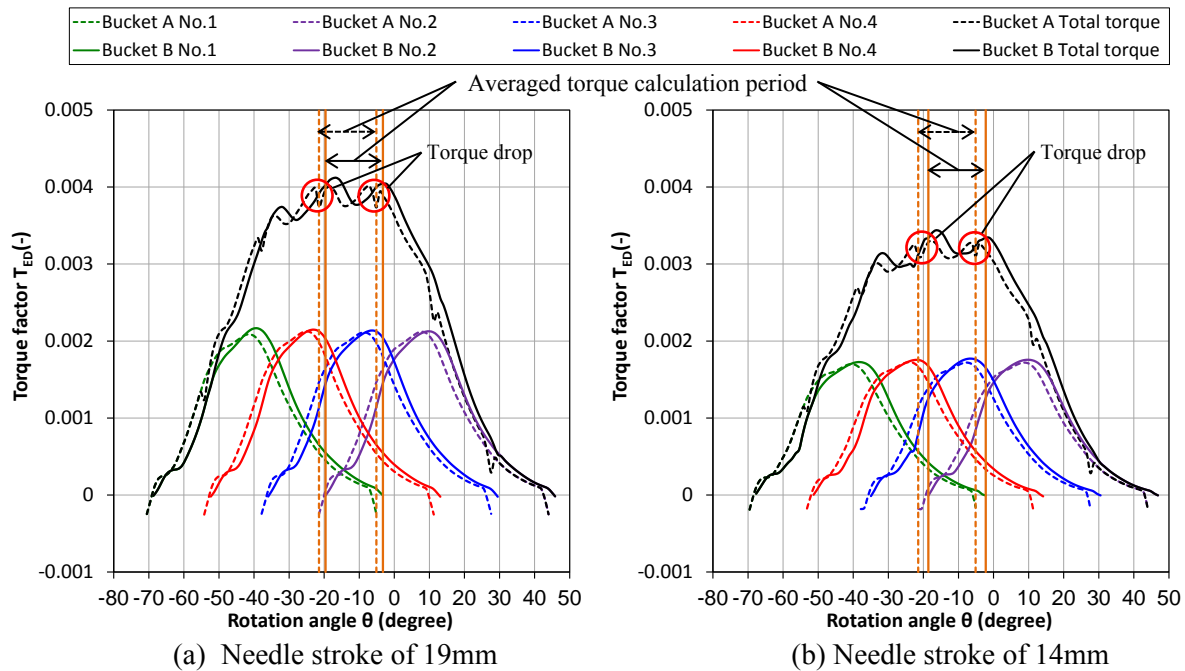
**Table 3.** Approximate number of mesh elements (Unit : million).

Bucket name	A	B
Rotor domain	2.05	1.98
Stator domain at needle stroke 19mm	1.73	1.73
Summation	3.78	3.71

### 3. CFD simulation results

#### 3.1. Time evolutions of torques

The time evolution data of torques on each 4 buckets are extracted after the simulation becomes stable through the proper iterations. As representative examples, the relationships between rotation angle and torque factor for both buckets A and B at the needle stroke of 19mm and 14mm are shown in figure 3. Some differences can be observed in the time evolution of torque between bucket A and B.



**Figure 3.** Relationship between rotation angle and torque factor on each bucket.

The broken lines show torque of bucket A and the solid lines show torque of bucket B. Here, the rotation angle 0 degrees is corresponding to a position that No.3 bucket receives the water jet vertically, and the positive angle means rotational direction and the negative angle means opposite to rotational direction. The averaged torque is calculated in the period which is indicated by orange lines.

At first, a maximum torque of bucket B generated on each bucket is higher than one of bucket A on the both needle stroke conditions. This contributes that the time averaged total torque of bucket B is also higher than one of bucket A. Consequently, it results that the calculated turbine efficiency of bucket B which will be mentioned later is higher than one of bucket A.

At second, a rotation angle of bucket B on which it generates a maximum torque is almost 2-3 degrees larger than one of bucket A. This is caused by the difference of the start timing of receiving water jet. As bucket A starts to receive water jet earlier than bucket B, the end timing of bucket A is also earlier than one of bucket B. Since the torque generation is theoretically the most efficient at the rotation angle that a bucket receives water jet vertically, the design of bucket B is better than one of bucket A on this view point. And these timings can be controlled by changing some design parameter and the number of buckets.

At third, the torque drops which are indicated by red circles in figure 3 are observed in the torque time evolution of bucket A on the both needle stroke conditions. Therefore, it decreases the total torque. This has never seen in the case of bucket B. As the drop occurs at the start timing of receiving

water jet, it is expected that the drop is caused by the flow disturbance when a bucket goes across into the water jet. This concludes that the design around the splitter and the cutout of bucket A can be optimized like bucket B not to cause this torque drop.

Furthermore, on the both needle stroke conditions, although the absolute value of torque is definitely different from each other, there are similar trends of torque time evolution in the both buckets A and B. This means that it is possible to design a bucket based on one calculation point for a rough investigation at the beginning phase of the bucket design optimization process.

In addition, all the features mentioned above are also observed on all other needle stroke conditions.

### 3.2. Comparison of turbine efficiency with model test

The comparison of the model turbine efficiency between CFD simulation and model test for both buckets A and B is shown in figure 4. And the model test equipment is shown in figure 5 as an introduction. The efficiency is normalized by the maximum model test efficiency of bucket B. The efficiency in the numerical analysis is calculated by using the averaged torque and head in the time period mentioned above. The calculation head is defined by the difference between total pressure of inlet boundary and opening boundaries.

The efficiency difference between bucket A and B is clearly observed in the model test. The efficiency of bucket B is almost 2-3% higher than one of bucket A. The same trend is observed in the calculation efficiency and there is also almost 2% difference. This means that it is possible to estimate the relative difference of efficiency in model test from CFD simulation, of course, although the absolute value of efficiency and the trend for turbine characteristics of discharge change between CFD simulation and model test needs much improvement. As the difference of absolute efficiency and the tendency of discharge change between CFD simulation and model test is mainly caused by the under or over estimation of calculation head and calculation jet velocity, it needs more appropriate settings to solve the relationship of these parameters accurately in the numerical analyses which have a water injection from the water domain to the water-air domain. For the purpose of verification, velocity distribution measurement in the model test will be performed as a future task.

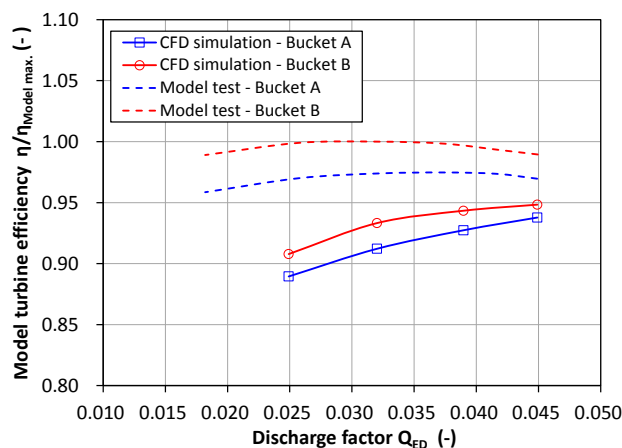


Figure 4. Comparison of model turbine efficiency.

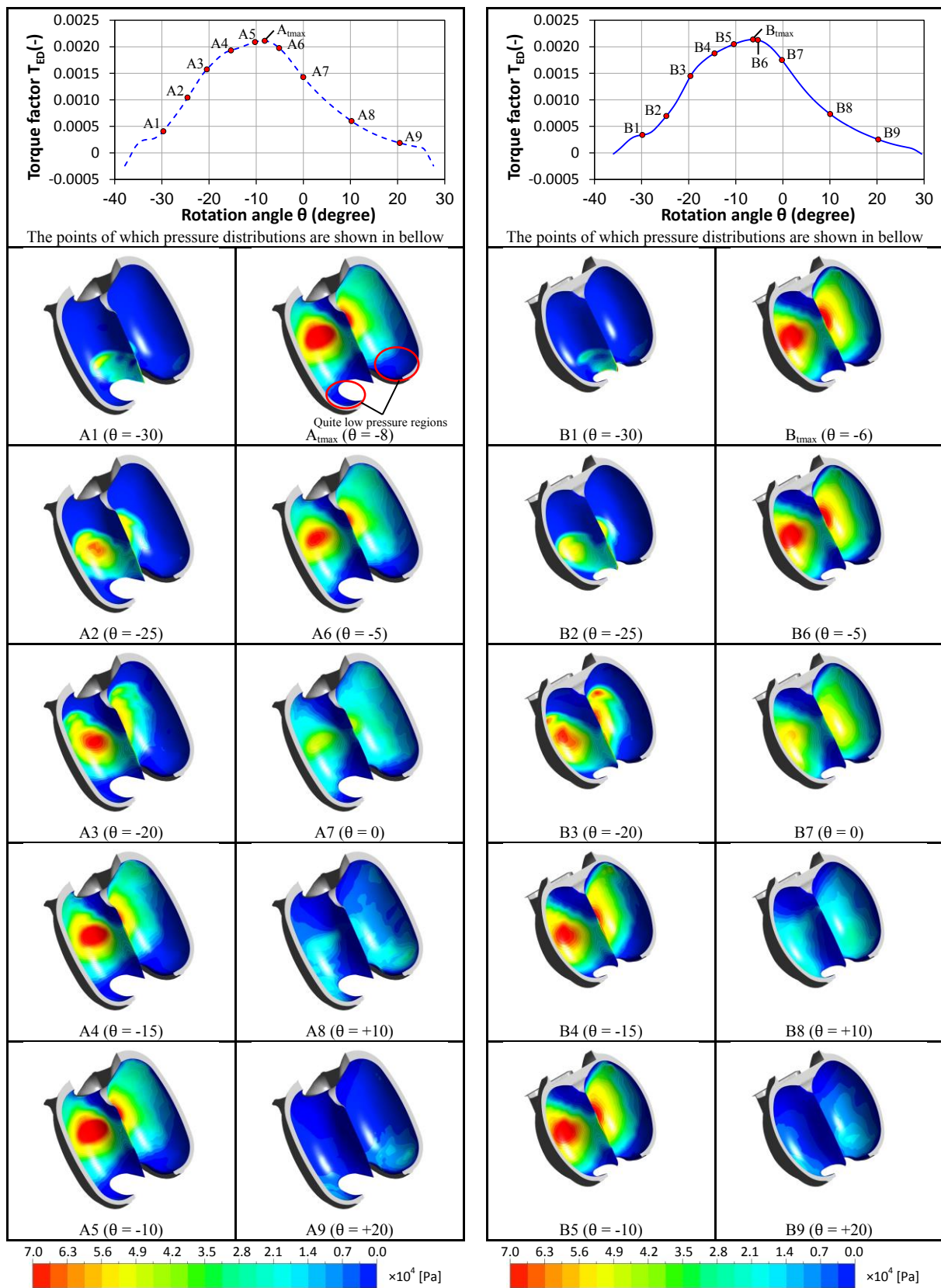


Figure 5. Model test equipment.

### 3.3. Time evolutions of flow patterns

As mentioned above, from the comparison between bucket A and B in CFD simulations which is verified by the model test, it can be found that there is more loss on bucket A than on bucket B. For the purpose of bucket design optimization, the difference of flow pattern between bucket A and B is investigated. On both buckets A and B, the time evolutions of pressure distributions on inner surface of bucket which are extracted from bucket No.3 at the needle stroke of 19mm are shown in figure 6.





(a) Bucket A (b) Bucket B  
**Figure 6.** Time evolutions of pressure distributions on inner surface of bucket.

The pressure distributions at the beginning stage of receiving water jet, which almost corresponds to A1-A3 for bucket A and B1-B3 for bucket B in figure 6, are almost similar to each other. This tendency of similarity can be also observed in the torque time evolution curves on the increasing phase which almost go parallel to each other in figure 3(a).

However, at the stage around maximum torque point, which almost corresponds to A4-A6 and B4-B6 in figure 6, the pressure distributions are clearly different from each other. Making a comparison on the maximum torque points of  $A_{tmax}$  and  $B_{tmax}$ , middle pressure region, which is indicated by around yellow color scale, of bucket B is relatively larger than one of bucket A. Meanwhile, the low pressure region, which is indicated by around blue color scale, of bucket A is widely distributed compared to one of bucket B. This difference of pressure distributions is explained by the difference of design parameters. Since the width and length of bucket A are larger than those of bucket B despite same depth of bucket as shown in table 1, the inner surface area of bucket A is definitely larger than the area of bucket B. Due to this, the water is easily spread wide inside bucket A and the low pressure region is also diverged widely. At the same time, in spite of this water spreading, it can be observed that the regions, which are indicated by red circles on the maximum torque point  $A_{tmax}$  in figure 6, almost do not have any pressure and never contribute to generate torque. From these observations on the around maximum torque point, it is found that the dimensions of bucket A are large and the design of bucket B is more suitable to receive this amount of water discharge.

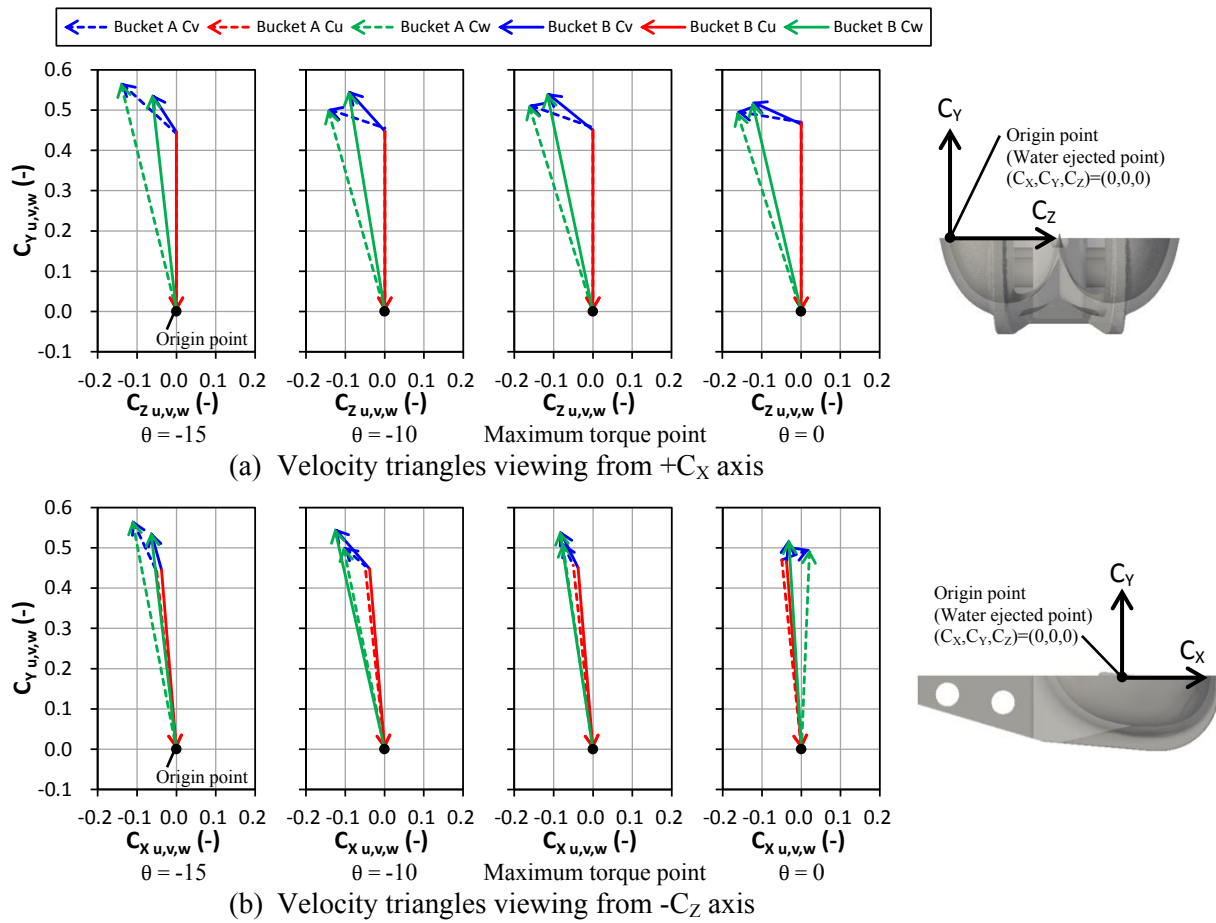
Moving on to the end stage of receiving water jet which almost corresponds to A7-A9 and B7-B9 in figure 6, there is also a difference on the pressure distributions. Making a comparison on the points A7 and B7 of which rotation angles are almost 0degrees, which is the most suitable point to generate torque, the pressure of bucket B is obviously higher than one of bucket A because bucket B finishes receiving water jet later than bucket A. This difference directly connects the difference of total torque.

Although a qualitative evaluation like the comparison of pressure distributions mentioned above is quite useful and helpful to capture and understand the features of flow pattern in bucket. On the other hand, in the process of design optimization, it is necessary to evaluate the characteristic of bucket from flow pattern calculated by CFD simulation in a quantitative way for finding out its faults which should be improved. As one of quantitative evaluation method, velocity triangle which is drawn by using weighted average of water volume fraction from numerical results is considered. The time evolutions of normalized and averaged velocity triangles at outlet in the static system on the points corresponding to the rotation angle of -15, -10, maximum torque point and 0degrees are shown in figure 7.

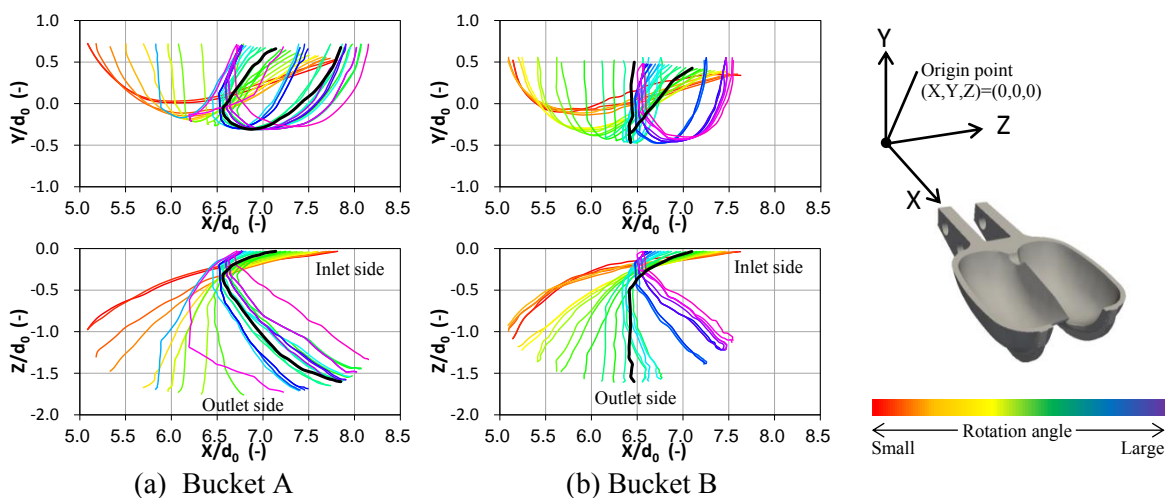
From a comparison of absolute velocity  $C_v$  on the maximum torque point, it is found that absolute velocity of bucket B is almost 9% smaller than one of bucket A. This difference of the velocity corresponds to the difference of almost 0.6% angular momentum loss in this case. If the absolute velocity at outlet is zero, theoretically the angular momentum loss ought to be zero. However, this also means that the water ejected from the outlet edge of bucket remains at the ejected point and it will be hit by next rotating bucket. Therefore, by designing the optimum outlet angle, the absolute velocity should be adjusted to minimum value with which the water can avoid to be hit by next bucket and not to generate collision loss. If bucket has a margin for strength, much smaller absolute velocity can be set by adopting thin thickness and suitable outer design of bucket. And it is also found that the absolute velocity has components of  $C_x$  and  $C_y$  directions for both buckets A and B on all the points shown in figure 7. The reason is that the speed factor of calculation point is smaller than that of best efficiency points obtained in the model test. The ratios of speed factors are  $n_{ED cal}/n_{ED BEP A}=0.99$  for bucket A and  $n_{ED cal}/n_{ED BEP B}=0.94$  for bucket B. The absolute velocity component of  $C_y$  direction should be adjusted to be zero on the best efficiency point that the circumferential velocity becomes equal to half of the absolute jet velocity at the inlet of the bucket  $C_{u inlet} = 0.5C_{v inlet}$ .

Moreover, an investigation of transient water passage based on CFD simulation results is performed to evaluate the design of inner surface. The representative passages of water which comes into bucket at every time steps corresponding to the rotation angle  $\theta$  from approximately -34 to -1degrees in figure 8. These water passages are expected from the velocity inside bucket and especially the initial point of water jet inflow is given by weighted average of water volume fraction.





**Figure 7.** Time evolutions of averaged velocity triangles at outlet edge of bucket in static system. Here,  $C_u$  is circumferential velocity,  $C_v$  is absolute velocity and  $C_w$  is relative velocity which are all normalized by expected jet velocity. Each velocity is calculated by using weighted average of water volume fraction around the outlet edge of bucket. For making a comparison between bucket A and B, the triangles are drawn from the same origin point  $(C_x, C_y, C_z) = (0, 0, 0)$ .



**Figure 8.** Time evolutions of representative water passages inside bucket. Black bold lines show water passages which come into bucket at rotation angle of 20 degrees.

As the water jet has a certain diameter, there are many water passages at the same time depending on the initial point of passage at inlet. Therefore, it simply shows a representative water flow passage of which initial point is located near the center of jet diameter. However, it is possible to calculate some effective parameters for the evaluation of inner surface design from these passages on each time step e.g. time required from inlet to outlet, length of passage and energy loss along the passage. For instance, from a comparison of passages which are shown as black bold lines in figure 8 between bucket A and B, it is found that the required time of bucket B is almost 10% less than one of bucket A and the passage length of bucket B is also almost 15% shorter than one of bucket A. And the energy loss integrated along the passage of bucket B is almost 0.2% less than one of bucket A. It is mainly caused by the difference of passage length and curvature change along the passage.

#### 4. Conclusion

As a basic study for bucket design optimization of Pelton turbine runners, unsteady two-phase flow CFD simulations based on the two different design buckets A and B were introduced. To reduce the computational load and to take it into practical design process, simplified analysis domain was applied to this study and its effectiveness was also confirmed. And the model test was also carried out to verify the accuracy of CFD simulation. There were same trends of relative efficiency difference in both model test and CFD simulation. To understand what causes the difference of efficiency, some investigations for time evolution of flow patterns on bucket were performed based on the numerical analysis. Furthermore, from the comparison of flow patterns between bucket A and B, some types of loss were evaluated and its generation mechanisms were also discussed. It is obvious that these evaluation and discussion about the relationship between mechanisms of loss generation and design parameters are quite effective and helpful to design optimized bucket. As the future tasks, to apply the appropriate optimization method to the practical bucket design and to improve the accuracy of analysis will be continued.

#### References

- [1] Solemslie B W and Dahlhaug O G 2014 A reference Pelton turbine - design and efficiency measurements *Proc. 27th IAHR Symp. Hydraulic Machinery & Systems (Montreal, Canada)* Paper No. 1-1-4.
- [2] Perrig A, Valle M, Farhat M, Parkinson E, Favre J and Avellan F 2006 Onboard flow visualization in a Pelton turbine bucket *Proc. 23rd IAHR Symp. Hydraulic Machinery & Systems (Yokohama, Japan)* Paper No. F236.
- [3] Nakanishi Y, Kubota T and Shin T 2002 Numerical simulation of flows on Pelton buckets by particle method (Flow on a stationary/rotating flat plate) *Proc. 21st IAHR Symp. Hydraulic Machinery & Systems (Lausanne, Switzerland)* Paper No.48
- [4] 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.
- [5] 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 & Systems (Lausanne, Switzerland)* Paper No.43
- [6] 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 & Systems (Montreal, Canada)* Paper No. 1-1-2.