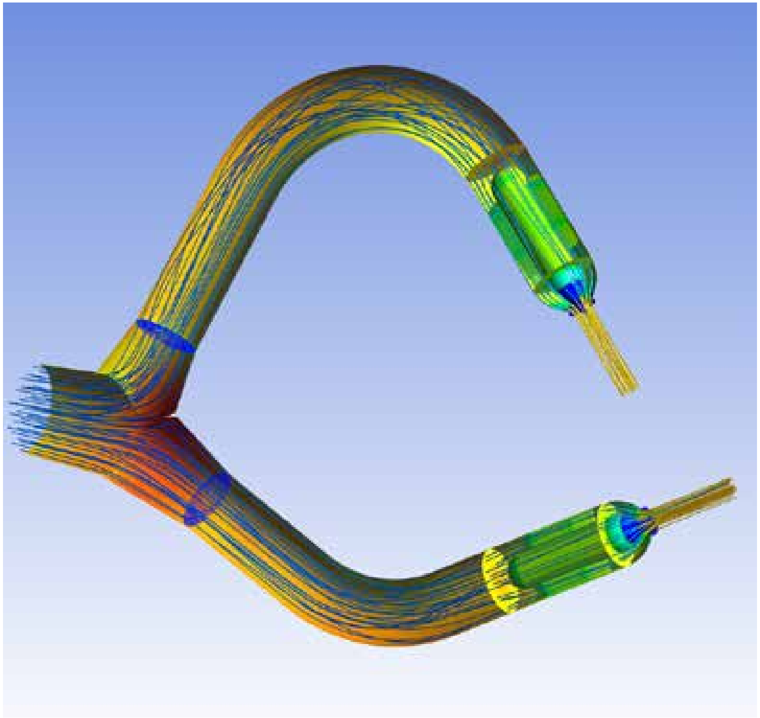
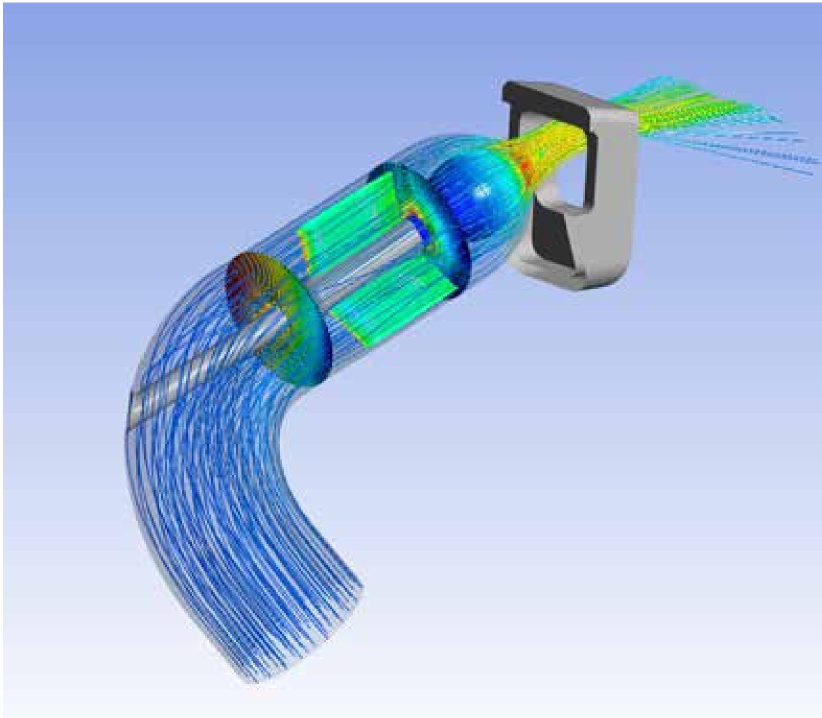


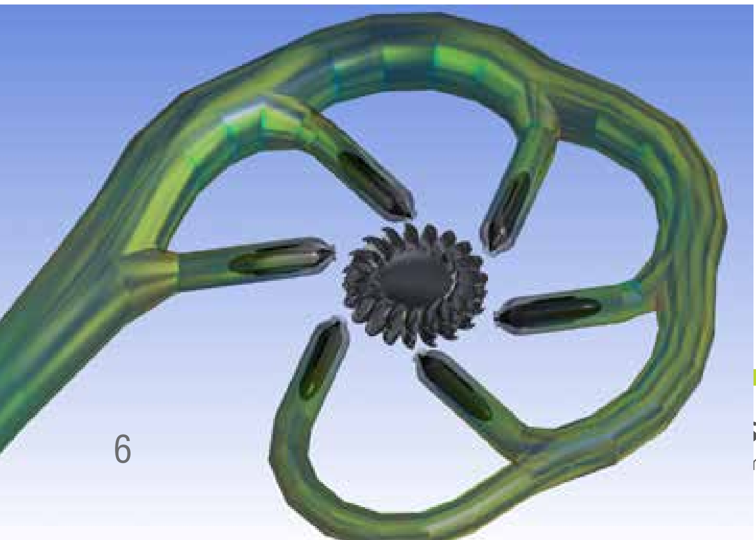


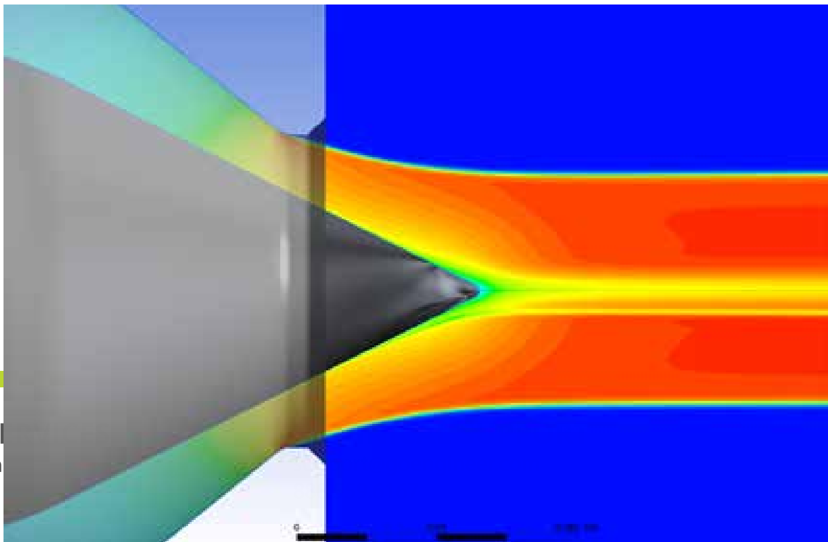
# CFD study of a Pelton turbine runner

## Comparison between traditional Eulerian and novel Lagrangian approaches

**By M. Minozzo[1], R. Bergamin[1], M. Merelli[2], M. Galbiati[2]**
1. ZECO - 2. EnginSoft

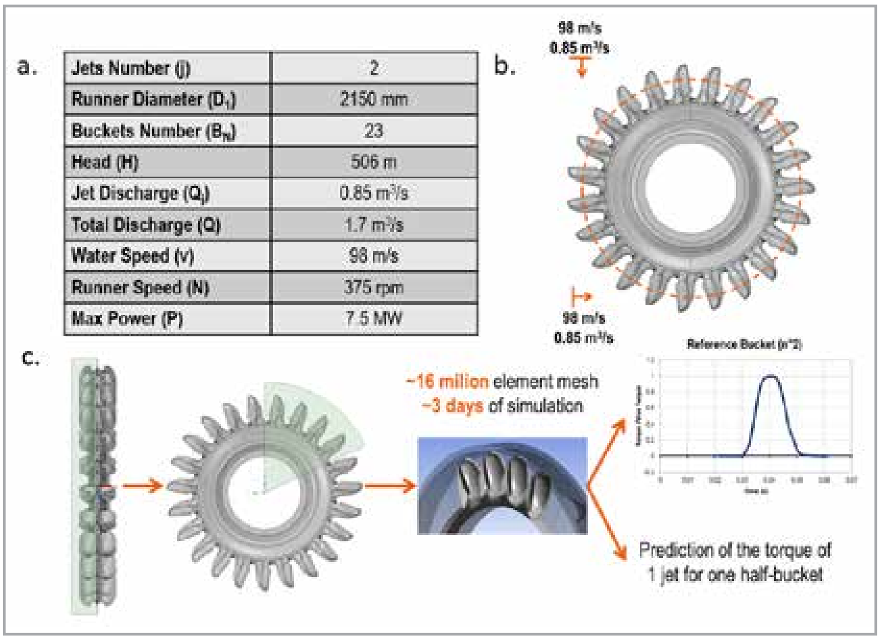
*Hydroelectric power generation is currently the predominant source for low-carbon power generation and to support grid stability in the face of the growing use of other unpredictable, renewable energy sources. This means that water turbines are becoming the focus of increased study and optimization. This technical article, a collaboration between EnginSoft and ZECO, compares two different methodologies for the study and optimization of impulse turbines, specifically Pelton turbines, in order to evaluate which is the quicker and more reliable method. Pelton impulse turbines are more difficult and challenging to analyze than reaction turbines due to the complexity of their fluid dynamics and the resulting computational resources required for the necessary transient multiphase simulation. The unsustainably high time and computing requirements mean that there are some technical deficiencies in sector knowledge about specific elements of these turbines and their functioning, such as the inside of the water jet. Two methods were evaluated: the traditional Eulerian approach and a novel Lagrangian approach using Moving Particle Simulation (MPS). The novel MPS approach proved to save considerable time and revealed information not discovered before, opening up new possibilities for optimizing these turbines.*





Hydroelectric power generation is a crucial source of electricity, accounting for 44% (IEA, 2020) of global low-carbon power generation. Its leading role is expected to be consolidated, as it becomes reinforced by developing countries and by the growing awareness of climate change. In addition, the renovation or repowering of old power plants is crucial for greener power production and to support grid stability, considering the growing use of unpredictable renewable energy sources, such as wind and solar.

The combination of these factors will increase the need to study and optimize water turbines under different conditions, not only at nominal design points. The standard Eulerian computational fluid dynamics (CFD) approach has been extensively tested and validated for reaction turbines such as Kaplan and Francis turbines.



|  |  |
|---|---|
| Jets Number (j) | 2 |
| Runner Diameter ($D_1$) | 2150 mm |
| Buckets Number ($B_N$) | 23 |
| Head (H) | 506 m |
| Jet Discharge ($Q_j$) | 0.85 m$^3$/s |
| Total Discharge (Q) | 1.7 m$^3$/s |
| Water Speed (v) | 98 m/s |
| Runner Speed (N) | 375 rpm |
| Max Power (P) | 7.5 MW |

*Fig. 1 - a) Summary table with details on the presented benchmark; b) Boundary conditions and geometry included in Particleworks (Lagrangian model), where no geometry modification was performed; c) Schematics of the geometry simplification necessary in CFX (Eulerian model) – the turbine is reduced using symmetry (sectors in green), the simulation is run, and the half-bucket profile is extracted.*

It is already standard practice to optimize their hydraulic design due to the limited computing resources required. Impulse turbines, such as Pelton turbines, have also been continuously studied using CFD [1], [2], [3], [4]. These studies usually focus on predicting the efficiency of the buckets and on the fluid behavior of the water entering and leaving the individual buckets, in order to understand how the bucket geometry influences the performance of the machine.

However, compared to reaction turbines, Pelton analysis is much more complex and demanding, both because of the fluid-dynamic complexity of the jet diffusion, and the computational resources required for transient multiphase simulation.

The lack of knowledge about the inside of the water jet – as a result of the unsustainable time and computing resources required – is a technical deficiency that needs to be addressed. For these reasons, ZECO partnered with EnginSoft to investigate a new methodology to quickly and reliably conduct CFD simulations for Pelton turbines.

This article discusses the differences between a turbine runner simulation using a classic CFD (Eulerian) approach and a Moving Particle Simulation (MPS) (Lagrangian) approach. The test case presented is the analysis of a two jets horizontal shaft Pelton turbine. The project data is shown in Fig. 1b.

## Conventional CFX (Eulerian) approach

From a hydraulic point of view, the Pelton turbine consists of a water inlet pipe or penstock, from 1 to 6 nozzles, and a runner. The manifold is a pipe, branched into up to six deviations, that leads water to the injector nozzle. The nozzle consists of a needle, which acts as an opening valve, and a water flow regulator that releases the flow in a free jet that impinges on the runner.

From a fluid-dynamic point of view, manifolds and nozzles are quite simple to study as they are either channeled flows or two-phase flows with a jet in the air in a limited and static portion of the volume [5], [3]. Instead, studies of runners involve greater challenges, due to the complex nature of the free-surface flow to be modeled. A Eulerian multiphase analysis of a complete turbine is time consuming and limited by the computational power requirements due to the complexity of the geometry and the simulation. To conduct a feasible Eulerian CFD analysis, the following assumptions establish the best practice for a traditional CFD simulation (see Fig. 1a):

- Reduction of the geometry using symmetry
- Reduction in the number of buckets analyzed, down to a minimum of three
- Creation of a domain (a statoric-rotoric for the inlet boundary condition and the rotating runner).

This enables the torque of a half bucket to be simulated and calculated for the full duration of the action of a single jet.

From there, it is necessary to work backwards to reconstruct the torque for the entire turbine. In other words, starting from the torque produced by a single jet in a half bucket, the torque must be doubled to calculate the torque of the whole bucket. The complete time history of the turbine runner's action is reconstructed manually to yield the total torque and its average value (see Fig. 3).

Using the planes of symmetry, it is possible to visually reconstruct the interaction of the jet with the bucket to better visualize the interaction between the two. This approach accurately estimates the power and therefore the performance of the machine and the hydraulic behavior of a bucket. However, it is obvious that some issues remain unresolved because some hypotheses do not always apply. In addition, jet-jet and jet-casing interactions are totally excluded from this CFD approach, as the simulations required to analyze these phenomena are unfeasible in an industrial R&D workflow.

## Advantages of the MPS methodology

Particleworks uses a Moving Particle Simulation (MPS), a CFD approach in which the fluid is discretized into particles (computational fluid volumes). The Navier-Stokes equations are solved on these particles using a Lagrangian approach which does not require the mesh-generation step, as the fluid has already been discretized. This allows for rapid model preparation and poses no additional problems when moving/rotating domains or wall boundaries are considered.

Typically, software based on this methodology is widely used in the automotive industry, where gearboxes, electronic axles and transmissions are simulated in whole-simulation systems. Other types of applications are soiling, mixing tanks and cleaning-jet analysis. In fact, thanks to its Lagrangian approach, Particleworks is ideal for the study of complex, free-surface flows. In this article, we present another interesting possible application: using MPS to improve product properties and design.

As mentioned, preparing and reducing the geometry slows the simulation time and limits the amount of information that can be extracted from the simulation. On the contrary, thanks to the characteristics of the MPS method, the preparation phases and times are considerably reduced. In fact, the geometry provided by ZECO only needed to be converted to a compatible format for Particleworks (Fig. 1c). It was possible to import the entire turbine without the splitting or meshing steps. After setting the numerical and boundary conditions, the simulation was ready to run. The simulation process was further accelerated by the possibility of parallel processing, enabled by the graphics processing unit (GPU) solver. In addition, it can be seen that the extraction of the torque prediction was easier and did not require the time-consuming profile reconstruction steps.

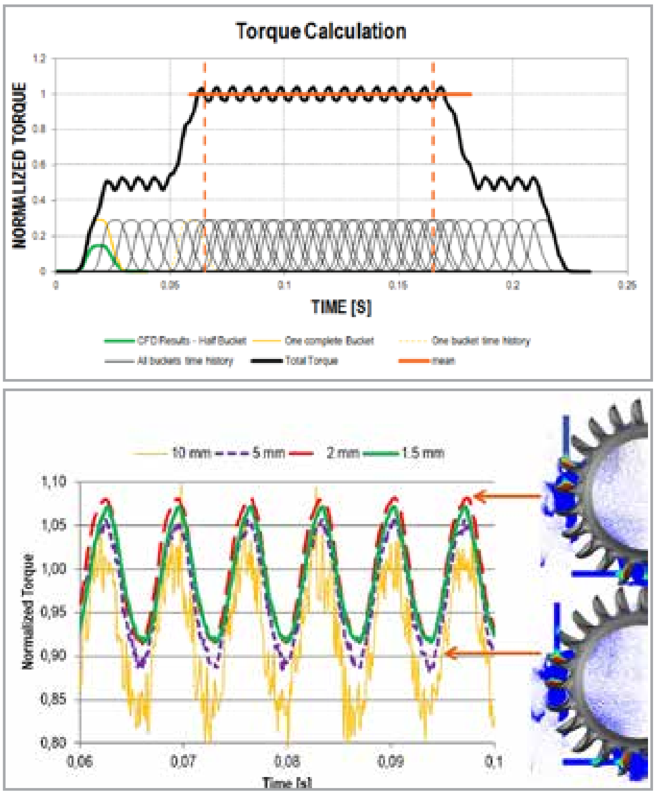


*Fig. 2 - Top: Normalized torque predicted by CFX – the average value is obtained after reconstruction of the turbine profile from an initial half-bucket profile; Bottom: Normalized torque based on the configuration of the entire turbine – the minima and maxima can be related to specific jet-turbine interactions (on the right).*

Just like in conventional CFD, computed results improve with smaller mesh features, at the cost of longer simulation times. In general, you can observe a convergence for better, theoretically expected results. In Particleworks, this type of analysis is performed by changing the particle size, i.e. the dimension of the computational volume. In this way, a solution can be found independent of the simulation settings and the discretization of the fluid volume. We performed several simulations with particle sizes of 10, 5, and 2mm. To quantitatively analyze the results, we extracted the torque on the turbine and plotted it over time. As can be seen, the torque prediction graph becomes smoother and converges into values closer to the theoretical value (Fig. 2).

To further validate the simulation results obtained using Particleworks, we compared them to the CFX simulation results. As can be seen from Fig. 3, both software packages overestimated the overall efficiency of the Pelton runner by the same percentage. The difference between the two approaches is negligible and simulations within a 1% error margin can be considered an excellent result considering the literature in this field ([2], [6], [7]).

MPS not only achieves qualitatively comparable results to traditional CFD, it does so in less time. Because it can simulate the entire turbine, it also provides design information about long-range runner-water interactions. This makes it possible to analyze the effect of residual water in otherwise active buckets, or other undesirable interactions between the water and the turbine. In addition, the optimization of the casing can be accomplished with the same simulation.

Another type of analysis that is usually performed in this sector is the evaluation of the static mechanical stresses on the turbine buckets. In CFX, due to the division of the simulated domain, remapping the pressure from the data of only the half bucket is time consuming. On the other hand, due to Particleworks' integration with Ansys Workbench, data transfer to an FEM solver is simple(see Fig. 4c).

To summarize the comparison between the Particleworks (Lagrangian) and the CFX (Eulerian) approaches, the simulation steps and their related time-costs are presented in Table 1. As

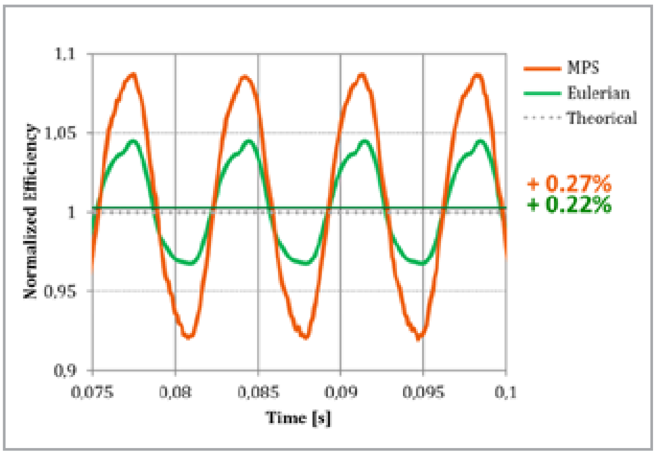


*Fig. 3 - Normalized efficiency prediction for Particleworks (in orange) and CFX (in green). The percentage of error is reported at the side. The theoretical mean values are also reported (dashed, black line).*

can be seen, Particleworks enables a significantly faster and easier simulation procedure. Since time is crucial in industrial applications, simulation times can be the bottle neck that block the development and investigation of new products. Various applications are not studied with CFD because of the complexity of the simulation steps. Particleworks can both accelerate the development of products that have already been studied, and pave the way for new studies and optimizations.

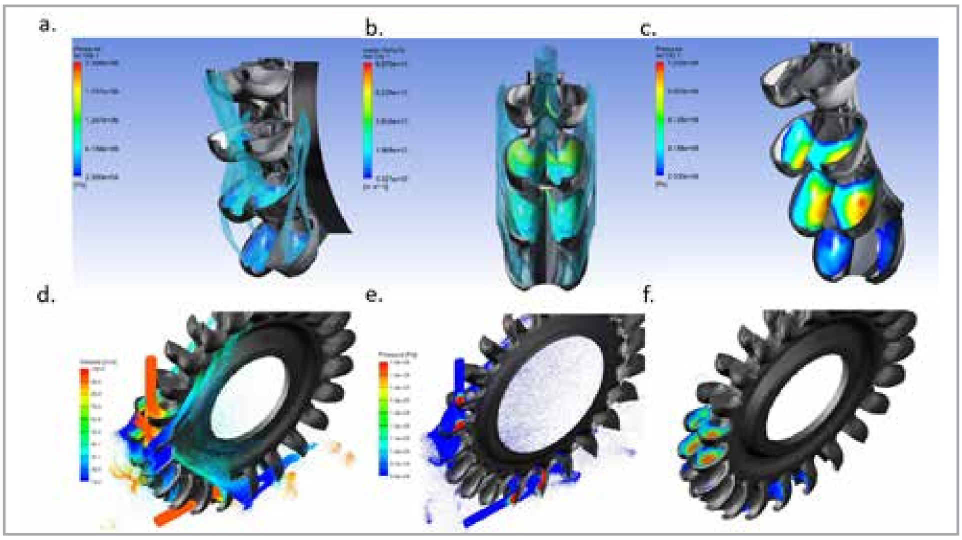


*Fig. 4 - a) and b) Images of the reconstructed surface (using the mirror plane) of the water jet for CFX (Eulerian method) – the velocity and pressure profiles are mapped on the Pelton bucket; c) Reconstruction of the pressure profile on the runner bucket (Eulerian method); d) and e) Images of the two water jets simulated with Particleworks (Lagrangian method) – the color map represents the predicted velocity and pressure; f) Mapping of the turbine pressure profile – Ansys Workbench allows direct data transfer of the profile to the finite element method (FEM) solver.*

| | CFX | PARTICLEWORKS |
|---|---|---|
| Pre / Post Processing | 3 working days / 4h | 2h / 1h |
| Simulation time | 70 h | 2h |
| Simulated rotation (°) | 138° | 225° |
| Geometry | 4 half buckets | Complete turbine |
| Complete runner simulation (multi jet, casing…) | Not feasible | Possible |
| Mesh elements/particles | 16 M | 4M |
| Hardware | 12 CPU<br>(Intel Xeon X5650 @2.67 GHz 96 GB RAM) | 1 CPU + 1 GPU<br>(NVIDIA V100) |
| Calculated vs model efficiency (absolute) | +0.22% | +0.27% |

*Table 1 - Summary comparison between the two approaches analyzed, highlighting working and simulation times, geometrical assumptions and hardware settings.*

## Conclusions

This article has analyzed the outstanding issues with and the possibilities of simulating a Pelton turbine runner using CFD. The traditional Eulerian, mesh-based approach was compared to the MPS method.

We found that the qualitative results obtained are comparable and in good agreement with the theoretical values. The Eulerian approach, however, obtained this result through a complex definition and simplification of the model, requiring a considerable amount of simulation and working time.

On the contrary, MPS can easily simulate the entire runner and the estimated workflow should only take 2-3 days. Moreover, the MPS method, from the same simulation, provides additional information never before investigated.

For instance, it provides insights into the jet-jet influence and the long-term jet-runner interactions. Those observables, together with the considerable acceleration in simulation time, open up new product optimization possibilities in the field of Pelton turbines.

## References

[1] K Patel, B Patel, M Yadav, and T Foggia. Development of Pelton turbine using numerical simulation. In IOP conference series: Earth and environmental science, volume 12, page 012048. IOP Publishing, 2010.

[2] Zh. Zhang. Pelton Turbines. 2016.

[3] D. Jo ˇst, P. Me ˇznar, and A. Lipej. Numerical prediction of Pelton turbine efficiency. 25th IAHR Symposium on Hydraulic Machinery and Systems, 2010.

[4] V. Gupta, V. Prasad, and R. Khare. Numerical simulation of six jet Pelton turbine model. Energy, 104:24–32, 2016.

[5] J. Zhang, Y. X. Xiao, J. Q. Wang, X. J. Zhou, M. Xia, C. J. Zeng, S. H. Wang, and Z. W. Wang. Optimal design of a Pelton turbine nozzle via 3D numerical simulation. Asian Working Group - IAHR's Symposium on Hydraulic Machinery and Systems, 2018.

[6] T. Kumashiro, H. Fukuhara, and K. Tani. Unsteady CFD simulation for bucket design optimization of Pelton turbine runner. 28th IAHR symposium on Hydraulic Machinery and Systems, 2016.