Understanding casing flow in Pelton turbines by numerical simulation

M. Rentschler, M. Neuhauser, J.C. Marongiu and E. Parkinson
ANDRITZ Hydro, Rue des Deux-Gares 6, 1800 Vevey, Switzerland
martin.rentschler@andritz.com

Abstract. For rehabilitation projects of Pelton turbines, the flow in the casing may have an important influence on the overall performance of the machine. Water sheets returning on the jets or on the runner significantly reduce efficiency, and run-away speed depends on the flow in the casing. CFD simulations can provide a detailed insight into this type of flow, but these simulations are computationally intensive. As in general the volume of water in a Pelton turbine is small compared to the complete volume of the turbine housing, a single phase simulation greatly reduces the complexity of the simulation. In the present work a numerical tool based on the SPH-ALE meshless method is used to simulate the casing flow in a Pelton turbine. Using improved order schemes reduces the numerical viscosity. This is necessary to resolve the flow in the jet and on the casing wall, where the velocity differs by two orders of magnitude. The results are compared to flow visualizations and measurement in a hydraulic laboratory. Several rehabilitation projects proved the added value of understanding the flow in the Pelton casing. The flow simulation helps designing casing insert, not only to see their influence on the flow, but also to calculate the stress in the inserts. In some projects, the casing simulation leads to the understanding of unexpected behavior of the flow. One such example is presented where the backsplash of a deflector hit the runner, creating a reversed rotation of the runner.

1. Introduction

Although Hydropower is amongst the greenest technologies that exist, it is subject to strong energy price pressure for the power plant owner and consecutively on the investments dedicated to rehabilitation projects. Numerical tools have hence gained importance to support those projects to screen rehabilitation options, even though they cannot fully replace a model test.

The hydraulic performances of all parts of a Pelton turbine, from Penstock to Tailrace, are today analyzed by numerical simulation. Each component has its own physical modelling as different physical quantities govern the flow in each of those parts. Each part influences the overall performance of the machine (for example: secondary flows in the penstock might produce cavitation on the runner). A detailed analysis of all components is therefore a key for an optimal rehabilitation.

To numerically simulate Pelton components, those can be divided into two groups: the internal flow parts and the free surface flow parts.

The internal flow parts of a Pelton turbine, which consist of the flow inside the distributor, injector and penstock, can be simulated with state of the art commercial solvers, giving a good prediction of their potentials in rehabilitation projects [9][10].
The standard CFD approach for the representation of the free surface boundary condition is usually done either implicitly by a two phase simulation treating the free surface as an interface between the phases [13][2], or by doing single phase simulations and defining the boundary conditions explicitly as for example when using deformable meshes [1][7].

It is technically feasible to simulate free surface flows inside a Pelton turbine with classical mesh-based CFD tools. However a proper tracking of the water inside the casing is very demanding, as mesh-based techniques naturally diffuse the free surface. Moreover, the meshing of a Pelton casing requires time and expertise, while mesh quality can strongly influence the numerical flow. These intrinsic characteristics push classical CFD tools to the limit of their cost-effectiveness.

Single phase simulations, where only volume occupied by the fluid is simulated, require only the discretization of this one phase. Pelton casing application this is attractive as a lot of space in the machine is filled with air which is not simulated and which influences only marginally the flow in the casing. One challenge of those single phase simulations are changes in topology of the volume occupied by water. This is difficult to handle, as neighbors have to be searched as soon as two water volumes unify [3].

An algorithm that does not use the notion of connectivity to neighbors and that uses a single phase approach, is the smoothed particle hydrodynamics (SPH) algorithm. This algorithm is described in section 2. An in-house tool based on this algorithm (SPH) has been further developed inside ANDRITZ Hydro for the last ten years and opens now new simulation possibilities. The engineering time and computational effort for those free surface simulations is drastically reduced thanks to this approach. As a consequence it becomes possible and affordable to evaluate potential for improvements of hydraulic components related to free surface flows from the offer phase.

An overview of the application to casing flows is shown in Section 3.

2. Numerical assessment

2.1. SPH

The meshless Lagrangian method SPH is not only used for a variety of academic problems but also for industrial applications ranging from hydrodynamics to solid mechanics. One of the reasons for its growing popularity lies in the meshless character of the method. In contrast to most of the other traditional CFD methods, SPH does not require the generation of a mesh because calculation points, called particles, are not connected and move according to the flow. In this way, they follow the position of the free surface which makes the method especially well adapted for the simulation of complex free surface flows where the position of the free surface is not known beforehand.

The SPH operators are derived by two discretization steps. The first is the convolution of a scalar or vector field $f(x)$ with a smooth, positive, symmetric and normalized kernel function $W$ that has a compact support $D(x)$ depending on the smoothing length $h$. The second step is the discretization of the convolution integral that yields

$$f(x_i) \approx \sum_{j \in D_i} \omega_j f(x_j) W(x_i - x_j, h),$$

with $W_{ij} := W(x_i - x_j, h)$. The particle with index $i$, position $x_i$ and volume $\omega_i$ is called the particle of interest and the particles that are situated in its kernel support $D_i$ are called neighbour particles. The SPH approximation of the gradient $\nabla f(x_i)$ is derived using integration by parts and yields

$$\nabla f(x_i) \approx \sum_{j \in D_i} \omega_j f(x_j) \nabla_i W_{ij} + \sum_{k \in \partial D_i} \omega_k^\partial f(x_i) n_k W_{ij},$$

where the second term is only non-zero if the kernel support intersects the boundary of the domain. The normals of the discretized boundary are denoted by $n_k$, and $\omega_k^\partial$ is the area of element $k$ of the surface discretization. Since the kernel function is a smooth analytical function, its gradient can be computed analytically [15].
In this paper we use Smoothed Particle Hydrodynamics-Arbitrary Lagrange Euler (SPH-ALE), a variant of SPH that was published by Vila [14] in 1999 in order to increase the accuracy and stability of SPH. The weakly compressible Euler equations written in an arbitrarily moving frame of reference are discretized by the SPH operators. Fluxes of mass and momentum are exchanged between neighboring particles that are computed by solving moving Riemann problems. The initial states for the Riemann problems are computed by the MUSCL scheme. Density and velocity are reconstructed at the midpoint between two particles by linear interpolation. The gradient that has to be computed for the reconstruction is obtained by a SPH approximation [6].

Hence, we obtain the following system of semi-discrete equations [6][4]

\[
\begin{align*}
\frac{d}{dt} (\omega_i) &= \omega_i \sum_{j \in D_i} \omega_j (v_0(x_j, t) - v_0(x_i, t)) \nabla_i W_{ij}, \\
\frac{d}{dt} (\omega_i \rho_i) &= \omega_i \sum_{j \in D_i} \omega_j 2 \rho_j \left( v_j^E - v_0(x_{ij}, t) \right) \cdot \nabla_i W_{ij} = 0, \\
\frac{d}{dt} (\omega_i \rho_i \mathbf{v}_i) &= \omega_i \sum_{j \in D_i} \omega_j 2 \left( \rho_j v_j^E \otimes (v_j^E - v_0(x_{ij}, t)) + p_j^E \right) \cdot \nabla_i W_{ij} = \omega_i \rho_i \mathbf{g},
\end{align*}
\]

with \( \rho_{ij}, p_{ij}^E \) and \( v_{ij}^E \) denoting the solutions of the Riemann problems for density, pressure and velocity respectively. The vector \( \mathbf{g} \) is the gravity vector. The last two equations are the SPH-ALE discretization of continuity and momentum equation, while the first two equations describe the evolution of particle positions and the particle volumes. The particle velocity \( \mathbf{v}_0 \) is an additional parameter that is introduced by the ALE formalism. It is independent of the flow velocity and can be chosen freely. In general for free surface flows it is set \( \mathbf{v} = \mathbf{v}_0 \).

An explicit time integration scheme, the 3rd order Runge Kutta scheme, is used and the time step size is determined by a CFL condition.

The system is closed by a barotropic equation of state, often referred to as Tait’s equation [15],

\[
p_i = \frac{\rho_0 c_s^2}{\gamma} \left[ \left( \frac{\rho_i}{\rho_0} \right)^{\gamma} - 1 \right], \quad \gamma = 7
\]

that links the pressure \( p \) to the density \( \rho \). The reference density is \( \rho_0 = 1000 \, \text{kg/m}^3 \) and \( c_0 \) is the reference speed of sound. If the physical speed of sound is chosen, Tait’s equation models the compressibility behavior of water accurately. However, the speed of sound has an influence on the time step size and numerical errors, in particular the numerical viscosity. Therefore, it is desirable from a numerical point of view to reduce the reference speed of sound. In practice it is chosen ten times the maximum flow velocity which ensures Mach numbers where the effects of compressibility are negligible, i.e. \( Ma \leq 0.1 \), on the one hand, and yields time step sizes that make the simulations feasible in the time frame of an industrial project, on the other hand.

Note that no energy equation is required to close the system. This is justified by the relatively small temperature variations that are observed in the considered hydraulic applications.

All simulations that are presented below are inviscid simulations. Nevertheless, phenomena are often observed in inviscid simulations that resemble the behavior of viscous flows and that are due to numerical errors. There are several strategies to reduce the so-called numerical viscosity that can be summarized in two groups. Either the number of calculation points is increased or a more accurate solver is applied in regions where the numerical viscosity is too large.

SPH is an intrinsically isotropic method where particle sizes are supposed to vary only slightly. It is very difficult to use local refinement and it is even more difficult to refine particles anisotropically.

As a consequence simulations with low numerical viscosity require small computational particles in the whole domain. This is computationally expensive and often not possible.

For this reason some work has been done to improve the accuracy of the solver. It has been shown that the numerical viscosity of the SPH-ALE method is considerably reduced if the initial states for the
Riemann solver are reconstructed by higher order polynomial functions instead of constant or linear functions. For the polynomial reconstructions of density and velocity the required gradient and the Hessian matrix are obtained by the Moving Least Squares (MLS) approach based on a weighted least squares fitting over the neighbor particles. For more information on this method the reader is referred to [12].

The importance of applying a more accurate method to industrial applications like the simulations of Pelton casing flows is demonstrated in section 3.3.

Figure 1: Casing flow: Water sheets leaving the runner impact on the casing and are guided into the tailrace.

2.2. Visualization

By the lack of local particle refinement, dataset up to some 10 million particles needs to be visualized. Efficient storage concepts and enhanced visualization tools are required to cope with this amount of data. Transient phenomena can often only be understood by creating movies under a certain perspective.

Relevant flow patterns can only be highlighted through thorough investigations of the 3D dataset under various view perspectives at a fixed time. The creation of a movie under selected view perspective(s), including a time range of the dataset adds the dynamic aspect to this perspective, which sometimes changes the interpretation. The computation time for the generation of the movie depends on the size of the dataset and might take some hours.

The parallel handling of the visualization enhances the possibilities of analyzing the data. But to obtain a “real time” visualization, in the sense that the dynamic behavior of the flow can instantaneously be observed on a pre calculated dataset, some more development is required.

While the computation time is getting cheaper, post-processing is the most demanding share of resources within the complete simulation.

3. Casing simulations

3.1. Casing flow

The flow in a Pelton casing is a transient multiphase turbulent flow that is governed by the water sheets exiting the runner and impacting the solid walls of the casing. The geometry of the runner
determines the velocity and thickness of the water sheets and therefore has a major influence on the casing flow (see Figure 1).

The flow in a Pelton casing is generally driven by inertia; the water is guided to the tailrace by the casing walls and inserts. As soon as there is an impact of the water sheets on a solid obstacle or when there is an intersection of one water sheet with another one, kinetic energy is dissipated and gravitation might start to drive the flow. Inserts are used to regularize the flow and to prevent jamming of the water. Water jams might produce water returning on the runner, which will reduce the efficiency of the runner in combination with a specific casing.

Figure 2: Flow in a round casing. The casing is large, so that one would not expect a negative influence of the casing on the runner. The casing simulation however indicated some potential risks.

Gravitational effects are difficult to capture, not due to numerical issues but due to simulation time. The typical simulation time for one rotation of a runner is in the order of one day. To capture gravitational effects some 10 rotations of the runner have to be simulated, which quickly results in computation times around a month.

The main challenge for casing simulation is that CFD in general does not map the full physics. Numerical errors, like the numerical viscosity, can modify flow features as well as the reduction of the complexity of the model, for example by ignoring the influence of the air/bubbles. The sum of those numerical and modelling errors results in the fact, that a quantitative prediction of the efficiency of a casing/runner system is not yet possible.

However the qualitative prediction of the flow in the casing helps a lot analyzing and understanding the flow in casings.

3.2. Flow analysis

Figure 2 shows an example where a casing simulation helped understanding the behavior of a machine. Normally the dimensions of the casing would lead to the estimation that only a small negative influence of the casing would be expected. At first sight the simulation (as on Figure 2) shows no specific behavior that would indicate a loss of efficiency. But several time later and under another perspective (see Figure 3), a strange behavior is observed. The water enters the injector bay that is in front of the originating jet. This water will flow along the injector return on the runner. The
theory could be confirmed by index measurements showing that the second injector, that has no injector bay in front, had a slightly better efficiency than the two other jets.

This example illustrates the advantage of CFD simulation compared to model tests, which is the direct accessibility of all physical quantities in a short time and at low costs. Velocity and pressure distributions are available everywhere in the fluid and on the wetted solid surfaces. To obtain the same information out of model tests or prototype measurements would require a very expensive and time intensive instrumentation of the machine.

![Figure 3: Flow in a round casing from an unusual perspective. Today a qualitative assessment of the flow based on numerical simulations is possible.](image)

The observation of the free surface position of the flow is limited on the prototype by the presence of foam, bubbles and droplets that circulate in the casing and that prevent positioning a camera anywhere in the casing. Only very specific places, like below an injector are possible for a camera. To estimate the influence of obstacles on the flow, the velocity and pressure distribution on the solids are observed. The position of the water impacts on obstacles is well documented by comparison with paint removal on prototypes and direct observations on model tests. From comparison with pressure distribution in the runner, the here used SPH code proved that pressure distributions can be accurately reproduced [14][11]. Thereby the influence of obstacles on the flow can be qualitatively judged based on those simulation results (see support of injector on Figure 3).

For the impacts on walls, the main danger is that there is water flowing back on the runner. As seen on Figure 3, the reflected water on the conus of the casing is still travelling to the top of the casing and will not return on the runner.

Intersecting water sheets suffer the most from numerical viscosity effects. In general the intersecting water sheets are relatively slow and thin. This means that those domains not only have a very low Mach number, but that they are also poorly resolved in terms of number of discretization point in one diameter.

3.3. Influence of numerical viscosity

The velocity in casing flows varies of 2 orders of magnitude, a jet speed of 100 m/s, which is equivalent to a head of 500 m, at the one extreme of the velocity range and a flow of around 1 m/s for water on the casing walls, which defines the lower extremum of the velocity scale.

For weakly compressible flow simulations, the numerical viscosity increases while increasing the speed of sound (cf 2.1.). So on the one side, the speed of sound must be high to keep the Mach number in a range where compressible effect are negligible, whereas on the other side the numerical viscosity can have an important influence on the flow in low velocity regions or in thin water layers.
Reducing the size of discretization is one possible way out of this dilemma but only at high computational costs, as the computational effort scales with \((1/\Delta x)^4\), meaning that a division by 2 of the particle size leads to an increase of the computation time by a factor 16.

Another possibility to reduce numerical viscosity is the implementation of higher order discretization schemes. These schemes allow for more precise results with similar discretization size, at the price slightly increased memory requirements and computation load. But to obtain the same result with pure particle refinement would be much more expensive.[12]

Figure 4 shows a comparison between two numerical simulations at the same physical time of the simulation with two different discretization schemes for the convection terms of the Euler equations. Subfigure (a) shows the classical SPH approach for the spatial reconstruction to calculate the fluxes of the Riemann solver, whereas on (b) an increased order algorithm is used. The increased order of the solver reduces the numerical viscosity. Therefore the water dissipates less kinetic energy while travelling thru the bucket and leaves the bucket with higher residual energy. This slightly changes the angles of the leaving water sheets. This explains why the impact of the water sheet (indicated by the arrows on Figure 4) on the casing wall is slightly moved towards the injectors when the increased order algorithm is used. Comparing to model tests confirms that the reduced viscosity results in a better position of the impact of water sheets on the casing walls.
This use of increased order schemes opens new possibilities in the field of casing simulations, as it increases the confidence on the impact positions on the casing walls and inserts and thereby helps designing inserts to reduce the risk of efficiency losses due to unexpected flow patterns for example in the case of a change of runner geometry without model test.

![Figure 5: Stress assessment on inserts using casing simulations. On the left the start-up of a Pelton runner, on the right the resulting pressure distribution on a deflection plate.](image)

However the higher order schemes cannot help for under discretization. Especially for very thin layers of water or for thin water sheets, refinement of the simulation is the only way how to capture those phenomena correctly, simply by the fact that a thickness of several particles is required to resolve the equations of motion adequately.

The simulation of droplets for example is far from being possible in the length scale of a casing. The influence of these flow patterns on the overall efficiency of the machine is expected to be in the range of the other numerical and physical errors that are introduced into the simulation.

3.4. FEM on inserts / transient Operating point

While the pressure and stresses on casing and inserts are relatively small during nominal operation, the picture might completely change for off design operating points. Especially because filigree structures will have more potential to increase efficiency than fat or robust ones. Therefore the calculation of the pressure distribution on casing insert is standard when structural questions arise.

Non-standard operating points are another domain of application for the casing simulation; those non-standard operating points can be for example the use of deflectors, start-up or runaway. For nominal operating point, the water exiting the runner has minimal residual speed, but during start up the energy in the water sheets leaving the runner will be an order of magnitude higher and their positions change (see Figure 5). This leads to increased noise and to higher stresses in the concerned area.

An accurate prediction of the pressure distribution on solid obstacles is the starting point for a structural analysis. The transient pressure fields of the CFD simulation are used to calculate the deformation of the solid faces and their fatigue behavior. Especially when adding inserts in rehabilitation projects, where the casing is often made of casted steel and where the inserts needs to be fixed by screws, the calculation of the forces on inserts is important for the lifetime assessment of the inserts.

Due to the free surface character of the flow, the displacement of the solids does not influence the overall behavior of the flow. Therefore a one way coupling between the CFD and the structural analysis is sufficient.

However an estimation of the excitation/damping character is important to determine an order of magnitude of the displacement in case of a resonance effects. Therefore the same approach as for Francis turbines [8] will be of interest for the estimation of hydraulic damping on casing inserts.
The use of deflectors is classically not part of the casing simulation, but it is neither a classical runner application, as it is not clear, which part are interfering with the deflected water sheet. It might be only an interplay between runner and deflector (like in Figure 6), but it might also include a roof of an injector or even a sidewall of a horizontal casing. The power curve in Figure 6 is using a sliding average. The torque peaks are the bucket passages. The variation of the bucket passages indicates the stochastic behavior in the flow.

The estimation of such a braking torque on the runner is important not only from a stress perspective, as the bucket is loaded from behind, but also from a generator perspective, as the runner might start turning in the opposite direction as usual, which might cause damage to the machine.

4. Conclusion

The application of the smoothed particle hydrodynamics simulation technique on the casing flow of Pelton turbines has been presented. The use of this meshless technology enables the simulation of a casing flow as a single phase flow by neglecting the influence of the surrounding air. This enables the simulation of the flow in the complex geometry of a casing within project timescales. However these simulations could not replace model testing because of their modest fidelity. Better accuracy can be obtained with improved and precise numerical schemes. But the varieties of length scales that must be properly captured in Pelton casing flows require particle refinement and heavier computation loads. Accordingly expertise is necessary in manipulating this simulation approach, with knowledge of simplifications made and intrinsic limitations.

Comparison with model tests and return of experience from prototype increase the confidence into the numerical results, such that these numerical tools can be used today to a variety of applications ranging from design of additional inserts over fatigue assessment to the analysis of strange behavior of a machine.

Especially for rehabilitation projects, were unusual geometries might create unexpected behavior, these tools help finding robust solutions at reasonable costs.
References

[1] Hirt C.W, Amsden A., Cook J.L 1974 An arbitrary Lagrangian-Eulerian computing method for all flow speeds Journal of computational physics vol 14, (Elsevier) pp 227-253

[2] Hirt C.W, Nichols B.D 1981 Volume of fluid (VOF) method for the dynamics of free boundaries, Journal of computational physics vol 39, (Elsevier) pp 201-225

[3] Kucharik M., Shashkov M. 2007 Extension of Efficient, Swept-Integration-Based Conservative Method for Meshes with Changing Connectivity, International Journal for Numerical Methods in Fluids vol 56 (Wiley) pp 1359–1365

[4] Leduc J., Leboeuf F., Lance M., Parkinson E., and Marongiu J.-C. 2010 Improvement of multiphase model using preconditioned Riemann solvers. 5th Int. SPHERIC Workshop, Manchester (England)

[5] Marongiu J.-C. Leboeuf F., Caro J., Parkinson E. 2010 Free surface flows simulations in Pelton turbines using an hybrid SPH-ALE method Journal of Hydraulic Research vol 48 (Taylor & Fracis), pp 40-49

[6] Marongiu J.-C., Leboeuf F. and Parkinson E. 2008 Riemann solvers and efficient boundary treatments: an hybrid SPH-finite volume numerical method, Proc. 3rd Int. SPHERIC Workshop, Lausanne (Switzerland).

[7] Münch C., Ausoni P., Braun O., Farhat M., Avellan F. 2010 Fluid–structure coupling for an oscillating hydrofoil, Journal of Fluid and Structures vol 26, (Elsevier) pp 1018-1033

[8] Monette C, Nennemann B, Seeley C, Coutu A and Marmont H 2014 Hydro-dynamic damping theory in flowing water 26th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, Canada

[9] Parkinson E., Bissel C., Popescu E., Tirsi C. 2011 Upgrading Pelton turbines of LOTRU-CIUNGET HPP, Romania, HYDRO Conference, Prague, Czech Republic,

[10] Parkinson E., Rentschler M., Lais S., Karakolcu A., Weiss Th. 2014 , Life cycle of a pelton runner, HYDRO ASIA Conference, Colombo, Sri Lanka

[11] Perrig A. 2007 Hydrodynamics of the free surface flow in Pelton turbine buckets, Thesis, EPFL Lausanne (Switzerland)

[12] Renaut G.-A., Marongiu J.-C. and Aubert S. 2015 High-order and adaptive procedures for SPH-ALE simulations based on Moving Least Squares method. In 10th Int. SPHERIC Workshop, Parma (Italy)

[13] Sussman M., Smereka P., Osher S. 1994 A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow, Journal of computational physics vol 114 (Elsevier), pp 146-159

[14] Vila J.P., 1999 On particle weighted methods and Smoothed Particle Hydrodynamics, Mathematical Models and Methods in Applied Sciences, (Wiley) vol 9: pp 161-209

[15] Violeau D. 2012 Fluid Mechanics and the SPH Method. Theory and Applications. (Oxford, Oxford University Press)