Flow simulation of a Pelton bucket using finite volume particle method

C Vessaz¹, E Jahanbakhsh¹ and F Avellan¹
¹ EPFL Laboratory for Hydraulic Machines, Av. de Cour 33bis, 1007 Lausanne, Switzerland
E-mail: christian.vessaz@epfl.ch

Abstract. The objective of the present paper is to perform an accurate numerical simulation of the high-speed water jet impinging on a Pelton bucket. To reach this goal, the Finite Volume Particle Method (FVPM) is used to discretize the governing equations. FVPM is an arbitrary Lagrangian-Eulerian method, which combines attractive features of Smoothed Particle Hydrodynamics and conventional mesh-based Finite Volume Method. This method is able to satisfy free surface and no-slip wall boundary conditions precisely. The fluid flow is assumed weakly compressible and the wall boundary is represented by one layer of particles located on the bucket surface. In the present study, the simulations of the flow in a stationary bucket are investigated for three different impinging angles: 72°, 90° and 108°. The particles resolution is first validated by a convergence study. Then, the FVPM results are validated with available experimental data and conventional grid-based Volume Of Fluid simulations. It is shown that the wall pressure field is in good agreement with the experimental and numerical data. Finally, the torque evolution and water sheet location are presented for a simulation of five rotating Pelton buckets.

1. Introduction
The deviation of a high-speed water jet by the Pelton buckets is a challenging fluid mechanics problem, which involves complex geometries, moving boundaries, free surface flows and high-pressure variations. The ability to simulate accurately the wall pressure field in the Pelton buckets is a key issue for the design of Pelton runners. Mack and Moser [1] and Perrig et al. [2] used grid-based Volume Of Fluid (VOF) computations to simulate the flow in a Pelton bucket. Later, Marongiu et al. [3] used Smooth Particle Hydrodynamics (SPH) with Riemann solver to benefit from its Lagrangian formulation and overcome the mesh difficulties of VOF. The drawback of particle-based simulation compared to grid-based simulation is the significant increase of computing time. Recently, Anagnostopoulos and Papantonis [4] proposed a fast Lagrangian computation to design Pelton buckets. However, this method is only based on the inlet and outlet velocity vector of the particles, which provides an estimation of the integrated pressure. Neither the whole pressure field nor the exact water sheet location can be accurately computed.

The Finite Volume Particle Method (FVPM) is a particle-based solver introduced by Hietel [5]. This method features an Arbitrary Lagrangian-Eulerian (ALE) formulation, which means that the computing nodes can either moves with the material velocity or a user-prescribed velocity. Therefore, FVPM combines attractive features of Smoothed Particle Hydrodynamics and conventional mesh-based Finite Volume Method (FVM). Like SPH, FVPM is based on a
kernel and a smoothing length to compute the interactions between the particles. Here, we use a method based on the work of Quinlan and Nestor [6] to compute the particle interaction vectors exactly. Like FVM, the interaction vectors are used to weight the fluxes exchanged between the particles. Moreover, FVPM is locally conservative, which enables to perform accurate simulations with variable smoothing length. A variable smoothing length simulation is required to allow an efficient simulation of complex multi-physics phenomenon such as elasto-plastic solid simulation [7] and interactions between fluid and silt particles [8].

The purpose of the present paper is to validate that FVPM simulations are able to capture accurately the deviation of a high-speed water jet by a Pelton bucket. The FVPM simulations are done with the software SPHEROS, developed at EPFL since 2010 [9, 10]. The FVPM simulations of the flow in a stationary bucket are validated by VOF numerical simulations and experimental data obtained by Kvicinsky et al. [11]. The wall pressure field in the stationary bucket inner surface is used to compare the FVPM results with pressure sensors and VOF results.

In the following sections, we first introduce the governing equations, FVPM discretization and top-hat kernel. Then, we present the case study including the numerical setup. Finally, we compare the numerical simulations to experimental data and grid-based simulations of the flow in a stationary bucket for three different impinging angles and present the FVPM result of the flow in five rotating buckets.

2. Governing equations
The water flow is assumed weakly compressible. The flow motion is governed by the mass and linear momentum conservation equations

\[ \frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{C} \quad \text{and} \quad \frac{d(\rho \mathbf{C})}{dt} = \nabla \cdot \mathbf{\sigma} + \rho \mathbf{g} \]  

(1)

where \( \rho \) is the density, \( \mathbf{C} \) is the velocity vector, \( \mathbf{g} \) is the gravity vector and \( \mathbf{\sigma} = \mathbf{s} - \rho \mathbf{I} \) is the stress tensor, which includes \( \mathbf{s} \) the deviatoric stress contribution and \( \rho \) the static pressure. The latter is computed from the state equation

\[ p = \rho_o a^2 \left( \left( \frac{\rho}{\rho_o} \right)^7 - 1 \right) \]  

(2)

where \( \rho_o \) is the reference density and \( a \) is the sound speed. According to the weakly compressible assumption, the sound speed is set to \( 10 \cdot C_{\text{max}} \), \( C_{\text{max}} \) being the discharge velocity of the water jet. The governing equations (1) can be written as the Partial Differential Equation (PDE)

\[ \frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot \mathbf{F} (\mathbf{U}) = 0 \]  

(3)

where \( \mathbf{U} = \{ \rho, \rho \mathbf{C} \} \) represents the conserved variables and \( \mathbf{F} = \mathbf{Q} + \mathbf{P} - \mathbf{G} \) is the flux function, which is decomposed in \( \mathbf{Q} = \{ \rho \mathbf{C}, \rho \mathbf{C} \mathbf{C} \} \), \( \mathbf{P} = \{ 0, \rho \mathbf{I} \} \) and \( \mathbf{G} = \{ 0, \mathbf{s} \} \).

In FVPM, the Sheppard interpolating or shape function \( \psi \) is used to discretized the conservative form of the PDE

\[ \int_{\Omega} \frac{\partial \mathbf{U}}{\partial t} \psi_i dV + \int_{\Omega} \nabla \cdot \mathbf{F} \psi_i dV = 0 \]  

(4)

where \( \Omega \) represents the whole computational domain and \( dV \) an element of volume. The Sheppard function is zero-order consistent and is defined as

\[ \psi_i (\mathbf{x}) = \frac{W_i (\mathbf{x})}{\sigma (\mathbf{x})} \]  

(5)
where \( W_i(x) = W_i(x - x_i, h) \) is the kernel function and \( \sigma(x) = \sum_j W_j(x) \) is the kernel summation. The spatial resolution of the interpolation is given by the smoothing length \( h \).

After some mathematical operations [12], the PDE is simplified as

\[
\frac{d}{dt} (U_i V_i) + \sum_j \left( Q_{ij} - U_{ij} \cdot \frac{1}{2} (\dot{x}_i + \dot{x}_j) + P_{ij} \right) \cdot \Delta_{ij} - \sum_j G_{ij} \cdot \Delta_{ij} = 0 \tag{6}
\]

where \( \dot{x} \) is the particle velocity. In order to damp the spurious numerical oscillations, \( Q_{ij}, U_{ij} \) and \( P_{ij} \) are computed using the AUSM\(^+\) scheme of Liou [13] and a correction term is applied to the mass flux as described in [12]. The expression of the deviatoric stress \( G_{ij} \) is given by

\[
G_{ij} = \mu \left( \nabla C_{ij} + \left( \nabla C_{ij} \right)^T \right) \tag{7}
\]

where \( \mu \) is the dynamic viscosity, \( C_{ij} \) the averaged velocity and \( \nabla \) the gradient operator obtained from weighted least square to avoid double summation of gradient operator [12]. In equation (6), \( \Delta_{ij} \) represents a weight vector

\[
\Delta_{ij} = \Gamma_{ij} - \Gamma_{ji} \tag{8}
\]

which depends on the interaction vector between particles \( i \) and \( j \)

\[
\Gamma_{ij} = \int_{\Omega} \psi_i \nabla W_j \sigma dV = \int_{\Omega} \frac{W_i \nabla W_j \sigma}{\sigma^2} dV \tag{9}
\]

In the present study, we use a rectangular top-hat kernel to compute the interaction vectors, which reads

\[
W_i(x) = \begin{cases} 1 & \|x - x_i\|_{\infty} \leq h \\ 0 & \|x - x_i\|_{\infty} > h \end{cases} \tag{10}
\]

A 2D example of particles interactions with rectangular support is given in figure 1(a). The top-hat kernel is less smooth than a bell-shaped kernel as shown by the contours of the Sheppard shape function given in figure 1(b). However, Quinlan and Nestor [6] demonstrated that top-hat kernel allows a fast and exact computation of the interaction vector in 2D with a circular support. In 3D, Jahanbakhsh et al. [7] showed that the use of top-hat kernel with a rectangular support reduces significantly the geometric computations, in order to compute the integral of eq. (9). The latter is simplified as

\[
\Gamma_{ij} = - \sum_m \left( \frac{\Delta S_i}{\sigma^- \sigma^+} \right) \tag{11}
\]

where \( m \) is the number of partitioned rectangles, \( \Delta S \) represents the surface vector of the partitions, \( \sigma^- \) and \( \sigma^+ \) are the summation kernel inside and outside the surfaces respectively. An outline of the 2D computation of eq. (11) is given in figure 1(c), where the rectangular partitions are simplified as lines segments.

FVPM is an ALE method, which means that \( \dot{x}_i \), the velocity of the particle \( i \), could be prescribed arbitrarily. In the present study, we set the particle velocity equal to the flow velocity plus a correction vector

\[
\dot{x}_i = C_i + 0.4 \left( \frac{h}{C_{max}} \right) \sum_j \Omega_{ij}^* \frac{\Gamma_{ij} - \Gamma_{ji}}{|\Gamma_{ij} - \Gamma_{ji}|^2} \tag{12}
\]
where $\Omega_{ij}^*$ represents the interaction of bisected volume between the kernels of particle $i$ and $j$. The correction vector is applied to ensure a uniform distribution of particles in the flow and avoid particles clustering. The no-slip wall boundary condition is imposed by setting one layer of fluid particles on the wall surface and fixing their velocities $C$ and $\dot{x}$ equal to the wall velocity. The free-surface boundary condition is given by the particles location. However, the velocity correction applied in eq. (12) is modified to avoid an artificial spreading of particles through the interface. The time integration is performed using a second-order explicit Runge-Kutta scheme and the time step is computed by

$$\Delta t = 0.6 \cdot \min \left( \frac{h}{a + \| C_i \|} \right)$$

(13)

3. Case study
In the present study, a high-speed water jet impinges on: first one stationary bucket and second five rotating buckets. An outline of the case study is given in figure 2.

![Figure 2](image-url)
The bucket geometry and the 32 pressure samples are taken from Kvicinsky et al. [11]. The bucket width is \( B_2 = 0.09 \) m and its reference diameter is \( D_1 = 0.315 \) m. The location of the pressure samples fits the position of lines \( X_1 \) to \( X_7 \) and \( Z_1 \) to \( Z_5 \). In the stationary analysis, the bucket is tilted of an angle \( \theta = 72^\circ, 90^\circ \) or \( 108^\circ \) around the \( Z \) axis. In the rotating analysis, five buckets rotate around the \( Z \) axis with a rotation speed \( N = 1280 \) rpm. The five buckets are spaced with an angle of \( 18^\circ \). The jet diameter is \( D_2 = 0.03 \) m and its axis is in the \( -X \) direction at a distance \( Y = -D_1/2 \). The discharge velocity of the water jet is \( C_{\text{max}} = 28.5 \) m s\(^{-1}\) and \( C_{\text{max}} = 38.056 \) m s\(^{-1}\) for the stationary and rotating analysis respectively.

The FVPM simulations are run on two Intel Xeon CPUs E5 2670 at 2.6 GHz with 32 cores (hyper-threading \( 2 \times 16 \)) and 32 Gb of memory. The domain decomposition is \( 4 \times 4 \times 4 \) and the domains size is adapted according to the particles load using the adaptive domain decomposition strategy from Vessaz et al. [14].

4. Results

4.1. stationary bucket

The influence of the reference particle spacing \( X_{\text{ref}} \) on the FVPM results is analyzed for an impinging angle of \( \theta = 90^\circ \). Figure 3 shows the time history of \( F/F^* \), where \( F \) is the magnitude of the force applied on the bucket and \( F^* = 2\rho\pi(D_2/2)^2C_{\text{max}}^2 \) is the maximum force of the water jet. A mean value of the converged force, the computing time as well as the number of particles at the end of the simulation are given in table 1.

| \( D_2/X_{\text{ref}} \) | \( F/F^* \) | Computing hours | \( N_{\text{final}} \) |
|------------------------|-------------|-----------------|-----------------|
| 10                     | 0.8226      | 6.17            | 9’075           |
| 20                     | 0.8761      | 5.55            | 51’174          |
| 30                     | 0.8956      | 9.55            | 146’495         |
| 40                     | 0.9097      | 20.92           | 317’688         |
| 50                     | 0.9188      | 44.33           | 584’336         |

Figure 3. Time history of the force in a \( \theta = 90^\circ \) stationary bucket: influence of the spatial discretization.

Figure 4. FVPM simulation in a stationary bucket at an impinging angle of \( \theta = 90^\circ \): free surface reconstruction.
According to these results, the convergence of the FVPM results with the refinement of the spatial discretization is highlighted. However, the computing cost increases significantly with the spatial discretization. Figure 4 shows a free surface reconstruction of the water sheet for a discretization of $D_2/X_{ref} = 30$. The free surface location is less influenced by the spatial discretization compared to the force or pressure measurements.

The pressure coefficient $C_p = (p - p_{ref})/(0.5 \rho C_{max}^2)$ is compared to the VOF computations and measurements from Kvicinsky et al. [11]. The averaged $C_p$ profile along the lines X1 to X7 and Z1 to Z5 are given in figures 5 and 6 respectively.

**Figure 5.** Averaged $C_p$ along the lines X1 to X7 of the bucket at an impinging angle of $\theta = 90^\circ$; influence of the spatial discretization.
Figure 6. Averaged $C_p$ along the lines $Z_1$ to $Z_5$ of the bucket at an impinging angle of $\theta = 90^\circ$: influence of the spatial discretization.

In these figures, the FVPM pressure profiles fit qualitatively well the VOF and measurements from Kvicinsky et al. [11] despite some quantitative differences. Moreover, the convergence of the FVPM results according to the spatial discretization is highlighted. Finally, we present in figure 7 the wall pressure field comparisons between the finest FVPM results of $D_2/X_{\text{ref}} = 50$ for the three different impinging angles $\theta = 72^\circ$, $\theta = 90^\circ$ and $\theta = 108^\circ$. Once again, the FVPM results for the three different impinging angles fit qualitatively well the VOF computations and measurements.

4.2. Rotating buckets

The operating point simulated corresponds to a discharge coefficient $\phi_{B_2} = 0.20$ and an energy coefficient $\psi_1 = 3.38$. The simulated time corresponds to the rotation of the buckets of $230^\circ$, which allows the passage of the five buckets through the high-speed water jet. The spatial discretization selected for the FVPM simulation is $D_2/X_{\text{ref}} = 30$ because it provides a good compromise between accuracy and computing time. This simulation of $224'150$ wall particles and up to $279'152$ fluid particles lasted 5 days on 64 cores. Figure 8 shows the evolution of torque in each bucket as well as the total torque applying on the five buckets in function of the angular position. Figure 9 shows the evolution of torque in one bucket. This averaged torque is obtained by averaging the torque in the buckets 2, 3 and 4. A free surface reconstruction of the water sheet is given in figure 10. These results are promising by comparing them to the experimental measurements of Perrig et al. [2]. However, further FVPM simulations have to be investigated in order to obtain a proper comparison.
Figure 7. Comparison of the wall pressure field between experimental (left), VOF (middle) and FVPM (right) for the impinging angles $\theta = 72^\circ$ (up), $\theta = 90^\circ$ (middle) and $\theta = 108^\circ$ (down).

Figure 8. Evolution of the torque for each bucket (colors) and total torque (black) in function of the angular position.

Figure 9. Averaged torque for one bucket in function of the angular position.

5. Conclusion
The FVPM method with rectangular top-hat kernel allowed us to compute exactly and efficiently the particle interaction vectors. The convergence of the method according to the spatial discretization was highlighted in the stationary bucket analysis. Moreover, the FVPM results were qualitatively validated with the VOF computations and measurements from Kvicinsky et al. [11]. The FVPM results of the five rotating buckets were satisfactory. However, further simulations will be investigated to compare the FVPM simulation of a rotating Pelton runner to experimental data.
Figure 10. FVPM simulation of five rotating buckets: free surface reconstruction.

Acknowledgments
The research leading to the results published in this paper has received funding from both SCCER SoE, the Swiss Energy Center for Energy Research Supply of Electricity, granted by the Swiss Commission for Technology and Innovation (CTI), and the Ark, the foundation for innovation of Valais Canton, through the Project HydroVS. The authors would like also to acknowledge the financial support and technical assistance of ALSTOM Power Hydro for the development of the SPHEROS software.

References
[1] Mack R and Moser W 2002 Numerical investigation of the flow in a Pelton turbine 21st IAHR Symposium on Hydraulic Machinery and Systems
[2] Perrig A, Avellan F, Kueny J L, Farhat M and Parkinson E 2006 Flow in a Pelton turbine bucket: Numerical and experimental investigations Journal of Fluids Engineering 128 pp 350-358
[3] Marongiu J C, Leboeuf F, Caro J and Parkinson E 2010 Free surface flows simulations in Pelton turbines using an hybrid SPH-ALE method J. Hydraulic Research 48 pp 40-49
[4] Anagnostopoulos J S and Papantonis D E 2012 A fast Lagrangian simulation method for flow analysis and runner design in Pelton turbines Journal of Hydrodynamics 24 pp 930-941
[5] Hietel D, Steiner K and Struckmeier J 2000 A finite-volume particle method for compressible flows Mathematical Models and Methods in Applied Sciences 10 pp 1363-1382
[6] Quinlan N J and Nestor R M 2011 Fast exact evaluation of particle interaction vectors in the finite volume particle method Meshfree Methods for Partial Differential Equations V pp 219-234
[7] Jahanbakhsh E, Vessaz C and Avellan F 2014 Finite volume particle method for 3-D elasto-plastic solid simulation 9th international SPHERIC workshop
[8] Jahanbakhsh E, Vessaz C and Avellan F 2014 Silt motion simulation using Finite Volume Particle Method 27th IAHR Symposium on Hydraulic Machinery and Systems
[9] Jahanbakhsh E, Pacot O and Avellan F 2012 Implementation of a parallel SPH-FPM solver for fluid flows Zetta Numerical Simulation for Science and Technology 1 pp 16-20
[10] Vessaz C, Jahanbakhsh E and Avellan F 2012 FPM simulations of a 3D impinging jet on a flat plate comparison with CFD and experimental results 7th international SPHERIC workshop pp 214-220
[11] Kvicinsky S, Kueny J L and Avellan F 2002 Numerical and experimental analysis of free surface flow in a 3D non rotating Pelton bucket 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery
[12] Jahanbakhsh E 2014 Simulation of silt erosion using particle-based methods EPFL Thesis
[13] Liou M S 1996 A sequel to AUSM: AUSM+ Journal of Computational Physics 129 pp 364-382
[14] Vessaz C, Jahanbakhsh E and Avellan F 2013 FPM flow simulations using an adaptive domain decomposition strategy 8th international SPHERIC workshop pp 227-232