A COMPARATIVE STUDY ON PELTON TURBINES WITH CFD BASED METHODS

A. Ferrarini[1], R. Bergamin[1], M. Merelli[2], M. Galbiati[2], J. M. Domínguez[3]

[1]ZECO di Zerbaro e Costa e C.
Fara Vicentino (VI), Italy
anna.ferrarini@zecohydropower.com
riccardo.bergamin@zecohydropower.com

[2]EnginSoft
Bergamo (BG), Italy
m.merelli@enginsoft.com
m.galbiati@enginsoft.com

[3]Environmental Physics Laboratory
Universidade de Vigo, Ourense,
Spain
jmdominguez@uvigo.es

I. Introduction

Among the various sustainable energy resources, hydroelectric power generation accounts for 4.185 TWh of electricity in the world, representing the main source of renewable energy, according to IEA, 2023. In the field of hydroelectric power generation, two kinds of hydraulic turbine can be distinguished: reaction turbines, such as Francis and Kaplan turbines, and impulse turbines, such as Pelton turbines. In the last 30 years, Computational Fluid Dynamics (CFD) simulations have been widely used in the design optimization process and in calculation of hydraulic turbine performance. Although CFD simulations have been shown to achieve good accuracy for reaction turbines, their application to Pelton turbines still poses a challenge. In simulating Pelton turbines the fluid solver must capture ventilation losses, spray formation, secondary flows, interaction between jet and runner, unsteadiness, and the mixing phases of water and air.

Depending on the method chosen to describe the fluid, two possible approaches can be used in CFD simulations: the traditional Eulerian approach and the Lagrangian approach. The Eulerian approach is the most widely and accurate approach, especially for reaction turbine simulations. However, when addressing Pelton turbines, their CFD simulations are very computationally expensive, given the requirement for complex geometrical features, which consequently entail fine meshing, to resolve an unsteady, multiphase flow. The most popular software distributions for Pelton simulations based on Eulerian approaches are ANSYS CFX, ANSYS FLUENT, and OpenFOAM.

The Lagrangian approach is a meshless method and, therefore, can significantly reduce the computational cost. The main methods using Lagrangian approach are: Smoothed Particle Hydrodynamics (SPH), Fast Lagrangian Solver (FLS), and Moving Particle Semi-implicit (MPS). There is sparse literature on the use of Lagrangian methods applied to solve Pelton turbines. Koukouvinis et al. [1] simulated a Turgo turbine and compared the results of an Smoothed Particle Hydrodynamics (SPH) method against Fluent's. Sun et al. [2] performed a simulation of a Pelton turbine and validated the results with experimental data. Moreover, they analyzed the differences between ANSYS CFX, ANSYS Fluent, and DualSPHysics. DualSPHysics [3] is an open-source software that uses the SPH method and is a reference in the field of coastal and ocean engineering. Both publications achieved good results in describing the jet flows and capturing free surface. However, the torque curves present unwanted oscillations and the accuracy gap is significant compared to the results obtained with ANSYS CFX and ANSYS FLUENT, also due to the inability to describe the negative pressure region at the back of the bucket.

Despite the rapid development over the last 20 years, Lagrangian based schemes have not consistently emerged as engineering tools for Pelton turbine simulation. Surely, Lagrangian methods have promising advantages by making feasible simulations without geometry reductions and symmetric boundary conditions at low computational cost. However, the accuracy of solution is still challenging. Based on the previous study [4] and the above-mentioned context, the aim of this study is to perform an in-depth analysis of the torque characteristic curve, the efficiency, and the jet flow pattern of a Pelton turbine. The results of the simulation performed with the software DualSPHysics were compared with results from ANSYS CFX and Particleworks (commercial software that uses MPS), and theoretical values. To our knowledge, a detailed comparison between two Lagrangian methods (i.e., SPH and MPS) and a validation with a theoretical value have never been performed.

II. Methodology

The same Pelton turbine was simulated using ANSYS CFX, Particleworks, and DualSPHysics software. For each of these software a different method was used for this particular application: a homogeneous multiphase flow model was set in ANSYS CFX, the MPS in Particleworks, and the SPH method in DualSPHysics.

A. *Homogeneous multiphase flow model*

The homogeneous multiphase flow model was set in ANSYS CFX, combined with the volume of fluid (VOF), used to predict the shape of the interface between air and water. In particular, in each control cell the volume fraction for the water phase, α_w, and the air phase, α_a, was solved. The total volume fraction in a control cell is 1, that is, $\alpha_w + \alpha_a = 1$. The Shear Stress Transport (SST) $k - \omega$ turbulence model, commonly used in literature [5], was chosen for the simulation in ANSYS CFX.

B. Moving Particle Semi-implicit

The MPS method discretizes the fluid into particles, in which the Navier–Stokes are solved. It includes a kernel function, not necessarily differentiable, and particle number density. It is widely used in automotive industry, and in soiling, mixing tanks and cleaning-jet analysis as well. The main difference compared to the SPH method is that the MPS method is semi-implicit and not fully explicit.

C. Smoothed Particle Hydrodynamics

The SPH method was first developed for astrophysics applications. The continuous field is discretized into a set of particles, in which the discretized Navier-Stokes equations are locally integrated. The integral equations are based on interpolation function, that is the smoothing kernel function. The neighboring particles are generated according to a distance based function with an associated characteristic length, called smoothing length, h.

III. Numerical Set-up

This section describes the numerical set-up set in ANSYS CFX, ParticleWorks, and DualSPHysics. The main turbine data are reported in Table I.

TABLE I: Test case parameters

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

A. ANSYS CFX

An ideal, perfectly symmetrical jet configuration was considered, neglecting the impact of the injector geometry on the jet shape. In accordance with the state of the art [5], only four buckets and a single jet were considered in the simulation, in order to reduce the computational cost. Furthermore, only half of the turbine was modeled, taking advantage of the geometric symmetry of the system. Figure 1 on the left shows the numerical model, which includes a stationary, colored in gray, and a rotating domain, colored in blue. The stationary domain represents a portion of jet and an inner portion of the casing, while the rotating domain corresponds to the considered four buckets and an added volume to complete the geometry of the runner. To allow the flow to pass from the static to the rotating domain (and vice versa) an interface, called Transient Rotor-Stator in CFX, has been added. The mesh was generated in ANSYS Meshing and contains a total of 16.0 million elements. An unsteady, two-phase simulation was performed using the version 2022 R2 of Ansys CFX. In particular, a fixed time-step was set to $1.2{\cdot}10^{-5}$ s.

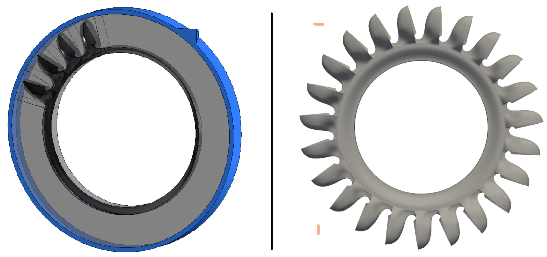

Fig. 1: Computational domains considered in ANSYS CFX (left), Particleworks and DualSPHysics (right).

B. Particleworks

The geometry of the turbine was discretized into Particleworks without applying any modifications or simplifications to the original CAD geometry. The entire turbine and the two jets were considered in the simulation. Figure 1 on the right shows the prescribed boundary conditions and the computational domain. After a sensitivity analysis, the particle size, i.e., the dimension of the computational volume, was set to 2.0 mm. An implicit (Divergence Free) solving method was used for the pressure component, in order to resolve with higher resolution the pressure developing in the flows. The initial time step was imposed to $7{\cdot}10^{-6}$ s. To further stabilize the calculations, a limiter was applied to the Courant–Friedrichs–Lewy (CFL) number. In this way, every particle moving at velocities greater than 5u_{max} (the maximal expected velocity) would be rescaled to 5u_{max}. The simulation process was run in parallel processing, enabled by the graphics processing unit (GPU) solver.

C. DualSPHysics

Analogously to Particleworks, the geometry was set without any modifications. The computational domain is the same considered in Particleworks, where the entire turbine and the two jets are simulated. The initial inter-particle distance was set to Δ = 2.0 mm and the Wendland smoothing kernel was chosen as also suggested in [2]. The value of the α coefficient in the viscosity term was set to 0.01. The numerically stable two-stage Symplectic scheme was used in the simulation. A variable time step is adopted with a CFL = 0.2, an initial time step Δt_{in} = 1.0 10^{-4} s, and a minimum time step Δt_{min} = 1.0 10^{-7} s. The modified dynamic boundary condition (mDBC) [6] was used for the geometry of the runner, initialized with several layers.

IV. Results

The simulation results were analyzed by observing the interaction between the jets and the runner, and by calculating the torque, power, and efficiency of the turbine. Fig. 2 shows the comparison between jet flows patterns obtained from the three software used. As these three panels imply, the interaction between jet–buckets is similar and well captured in all three cases.

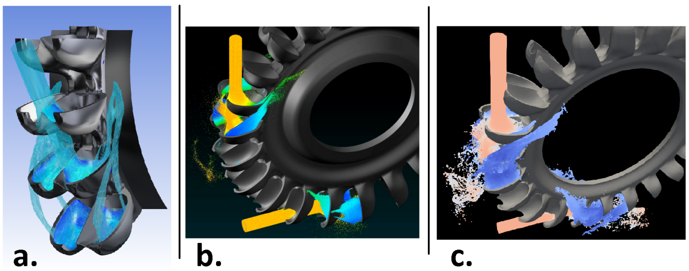

Fig. 2: Visual comparison: **a.** surface of the water jet reconstructed using ANSYS CFX; **b.** and **c.** water jets simulated with Particleworks and DualSPHysics, respectively.

The total torque at the shaft was calculated both in Particleworks and DualSPHysics directly from the post-processing, while in ANSYS CFX the torque of the third bucket (i.e., the second last bucket that intercepts the jet in the simulation) was computed, and cast into its full geometry composition for a single bucket. Then, this torque curve was shifted along the x-axis by 23 times (i.e., the total number of buckets), in order to represent the torque curve of all buckets over time. Therefore, the torque calculation time is significantly longer using ANSYS CFX compared to Particleworks and DualSPHysics. All calculated torque values were normalized to the theoretical turbine torque value. Fig. 3 shows the normalized torque curves calculated by ANSYS CFX, Particleworks, and DualSPHysics.

The normalized torque pattern is similar, but the torque amplitude is greater in Particleworks and DualSPHysics. Moreover, in DualSPHysics the torque curve fluctuates greatly and is unstable.

Regarding the efficiency, Particleworks and ANSYS CFX overestimated the efficiency, within a 0.3% and 0.2% percentage error, respectively, while DualSPHysics underestimated the efficiency, within a -8.6% error margin. This reduced accuracy is to be expected considering the relative low resolution, as compared to [2], where higher resolutions were used. The

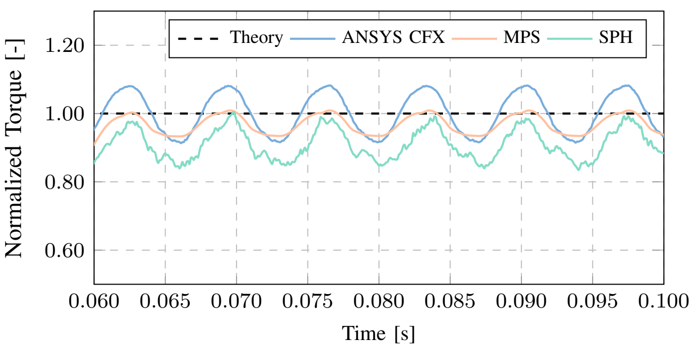

Fig. 3: Normalized torque curves obtained from DualSPHysics, Particleworks, and ANSYS CFX.

computational cost is higher for the simulation performed in ANSYS CFX compared to the Lagrangian methods: 3 days to run the simulation in ANSYS CFX (with Intel Xeon X5650 @2.67 GHz (12 threads), 96 GB RAM), 3 hours in Particleworks (1 CPU + 1 GPU NVIDIA V100), and 3 hours in DualSPHysics (1 GPU NVIDIA RTX 4090). In particular, ANSYS CFX has the possibility of parallel processing only with CPU solver, in contrast with Particleworks and DualSPHysics, which exploit the GPU-based solvers.

V. CONCLUSIONS

A comparison between Pelton turbine simulations performed with a classical Eulerian method (homogeneous multiphase flow model – ANSYS CFX), and two Lagrangian methods (MPS – Particleworks, SPH – DualSPHysics), was carried out. The low computational cost of both Particleworks and DualSPHysics allows to simulate the entire geometry of the turbine and to study long-term jet-runner interactions, in contrast to ANSYS CFX. The water film and the jet–bucket interaction captured by all simulations has proved to be similar. ANSYS CFX and Particleworks provide results in good agreement with theoretical values, providing an error in efficiency of less than 1%. DualSPHysics has the worst accuracy in predicting efficiency, with an error of 8.6%. This comparison places the SPH on a par with other methods for engineering applications, based on the reported runtime (**competitiveness**).

The SPH results appear to be off due to some limitations in the DualSPHysics numerical set-up. In particular, a sensitivity analysis on particle distance is missing, which should help to significantly improve the results as the particle distance decreases, as demonstrated by [2]. Therefore, simulations with reduced particle distance and a complete post-processing, including pressure distribution on the buckets, will be performed in future developments. Other promising developments in DualSPHysics include the introduction of casing and analysis of the runaway turbine speed, where maneuver under shut-down conditions are considered, which would require the investigation of a transient response of the turbine. (**novelty**).

REFERENCES

[1] P. Koukouvinis, J. Anagnostopoulos, and D. Papantonis, "Sph method used for flow predictions at a turgo impulse turbine: Comparison with fluent," *World Academy of Science, Engineering and Technology*, vol. 5, p. 528, 07 2011.

[2] J. Sun, X. Ge, and Y. Zheng, "Sph method used for characteristic predictions at pelton turbine buckets: comparing with the mesh-based method," *Engineering Computations*, 2023.

[3] J. M. Domínguez et al., "DualSPHysics: from fluid dynamics to multiphysics problems," *Computational Particle Mechanics*, vol. 9, no. 5, pp. 867–895, September 2022.

[4] M. Minozzo, R. Bergamin, M. Merelli, and M. Galbiati, "Cfd study of a pelton turbine runner," *Geoengineer*, 2020.

[5] A. Rossetti, G. Pavesi, G. Ardizzon, and A. Santolin, "Numerical analyses of cavitating flow in a pelton turbine," *Journal of Fluids Engineering*, vol. 136, no. 8, p. 081304, 06 2014.

[6] A. English, J. M. Domínguez, et al.,, "Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems," *Comp. Part. Mech.*, vol. 9, no. 5, pp. 911–925, September 2022.