Non-viscous flow simulations of six jet Pelton turbines: The effect of jet interference and of erosion on the performance of the runner

M Mettille[1*], J Decaix[1], A Bullani[2], B Valluy[3], C Münch-Allligné[1]

[1]Institute of Sustainable Energy, School of Engineering, HES-SO Valais-Wallis, Switzerland
[2]Mhlyab, Mini-hydraulic laboratory, Switzerland
[3]Alpiq, Switzerland

*mathieu.mettille@hevs.ch

Abstract. The computation of the flow in Pelton turbines remains a challenge mainly due to the two-phase flow in the bucket and in the casing. The present study focuses on simulations of two Pelton turbines, applying the finite volume method with the ANSYS® CFX® software and is divided into two parts: first, the degradation turbine efficiency due to jet interference (called "falaise" effect), and second, the performance losses due to erosion of the bucket splitter. Based on the simulations of six buckets using a non-viscous approach, the total torque of the runner and the efficiency are analysed. The sharp decrease due to the "falaise" effect is effectively captured within the measured values and the values predicted by the analytical formulas. The simulated efficiency decrease due to the erosion is in the same order of magnitude than the analytical formulas.

1. Introduction

As of 2009, Pelton turbines represent 31% of the European hydropower plants larger than 50 MW [1] [2]. Pelton turbines are known for their high discharge operating range but a lower capability of maintaining efficiency at significantly higher or lower head. This capability to operate at lower than design heads can potentially unlock unused storage capacity. The main components of this impulse turbine are a manifold, one to six injector(s) and a runner made-up usually of 17 to 23 buckets. Pelton turbines exhibits a variety of flows physic. From pressurized flow into the injectors, it becomes a two-phase flow during the development of the free waterjet, buckets' load and water sheets outflow. With such multitude of physics, simulating the flow into a Pelton remains a challenge. Different numerical approaches have been developed to correctly predict the flow and the performances of Pelton runners. The finite volume method has been widely used [3], but this mesh-based method faces challenges such as the meshing of complex geometries including rotating part or the modelling of the free surface flow and droplets. Mesh-less methods such as Lagrangian methods [4] or hybrid approaches [5] expend the scope of the possibilities during the last years. The present study investigates the capability of non-viscous simulations using the cell-vertex finite volume method computed with the ANSYS® CFX® software. Non-viscous simulations do not consider a turbulence model, nor wall friction but significantly reduce the computational time. This enables to perform set of simulations aiming at capturing the complexes interactions. Some dynamical trends can be analysed through parametric simulations.