Proceedings of the Canadian Society for Mechanical Engineering International Congress
32nd Annual Conference of the Computational Fluid Dynamics Society of Canada
Canadian Society of Rheology Symposium
CSME-CFDSC-CSR 2025
May 25–28, 2025, Montréal, Québec, Canada

Numerical simulation study of the blockage effect on the hydrodynamics of a Francis turbine at best efficiency point and partial load

William Dubé[1], Sébastien Houde[1]
[1]Département de génie mécanique, Université Laval, Québec, Canada
wdub703@gmail.com

ABSTRACT

Hydropower plays a central role in energy decarbonization. As hydraulic turbines are more often used to balance production and demand on electrical grids, the reliability of hydropower installations becomes a critical issue beyond purely economic considerations, making efficient diagnostics essential. This research is part of the MD-Francis project, carried out at the Heki Hydropower Innovation Center at Université Laval. It is conducted in collaboration with a consortium of six industrial partners and four universities, and focuses on detection and diagnostic of anomalies in Francis turbines. Within MD-Francis, various anomalies will be implemented on a structurally homologous model of a Francis turbine. Detailed measurements related to fluid-structure interactions will be performed to identify correlations that could be used to identify and categorize anomalies. This research focuses on the issue of blockages in the inter-blade channels of Francis turbines. These blockages, caused by debris or natural foreign objects (tree trunks, rocks, etc.), can lead to flow imbalances, generating vibrations and affecting overall performance. The blockage causes a head loss, resulting in a reduction of power. Additionally, these vibrations and imbalances can, in turn, cause more significant damage, potentially leading to major shutdowns of the turbine-generator unit. The main objective is to analyze the impact of these blockages on hydrodynamics and performance at the best efficiency point (BEP) and part load (PL) to guide experimental investigations. Numerical flow simulations of the unobstructed Francis, using a RANS approach, are presented and validated against experimental data to form a comparison baseline. Different blockages are then modelled as porous domains located inside an inter-blade channel, allowing for a realistic representation of partial flow restrictions. URANS simulations are used to analyze the hydrodynamic effects of those blockages on the flow. The analysis presented includes the impact on turbine performance, the unbalanced flow distribution in the distributor and runner, the resultant radial forces on the runner, the torque applied to adjacent blades, the rotor-stator pressure signals, and the impact on vortex development in the draft tube.