

Validation of Flow3D CFD by using physical modelling

Applicant will learn about Computational Fluid Dynamics, CFD, capabilities in the field of river sediment transport. This will be applied to the simulation of free surface flow through an intake/outlet structure in the lower reservoir of a Retrofitted Pumped Storage Hydropower plant, pertaining to power generation activities.

This is aimed at investigating hydropower operation effects on reservoir loose bed. To this objective, the inflow/outflow momentum and the produced sediment dynamics will be simulated using the CFD Flow3D Hydro by Flow Science, Inc.

Learning outcomes entail CFD using, performance evaluation and sensitive analysis. The performance in reproducing sediment dynamics will be tested by comparing different turbulence modelling: i) Renormalization Group (RNG) K-Epsilon version of Reynolds Averaged Navier Stokes (RANS) turbulence model, and ii) Large Eddy Simulation (LES).

Quantitative evaluation of CFD performance will be achieved by means of the scaled physical modelling reproducing scouring and depositions. This physical model has been developed in the DICAM Hydraulic laboratory, LIDR. The benchmarking dataset comprises time averaged velocity field, turbulence oscillation, suspended sediment concentration, erosion rate and final morphology. Specifically, CFD to scaled model matching will require boundary conditions controlling and replication. Furthermore, scaling and CFD approximation effects will be considered.

Learning program will consist in implementing steady and rump-up boundary conditions. The former set will be applied to characterize turbulence energy spectrum and correlations diagram under steadiness and clear water conditions (fixed bed), whereas rump-up modelling will trigger the actual sediment dynamics and consequent scouring.

Froude scaling of the physical modelling will result in two orders of magnitudes lower Reynolds number from prototype 10^7 value. CFD modelling approximation entails grid averaging, sub-grid turbulence simplification, and the evaluation of critical shear stress for the represented sediment fraction.

Fine sand will be applied in the physical model that would correspond to gravel for the prototype by considering 1:100 geometrical scaling, although critical shear stresses might scale differently because of the applied sediment transport model in the CFD.

Therefore, the learning plan for applicant will regard: i) the implementation of a set of boundary conditions in Flow3D, ii) analysing of turbulence features in the spectral and correlation domains, iii) sensitive analysis of applied parametrization by CFD to eventually simulate sediment dynamics.