water

MDPI

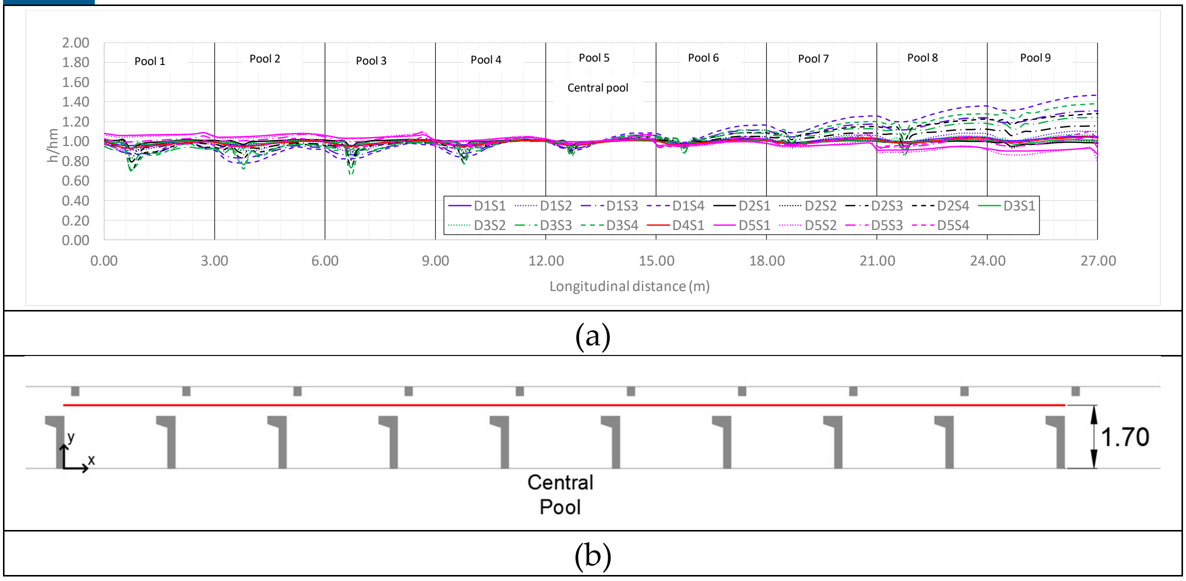

Figure S1. Flow depth along the model (a) Dimensionless flow depth along the nine pools of the model; (b) Line where the flow depth were evaluated – Pools 1 to 9 (left to right).

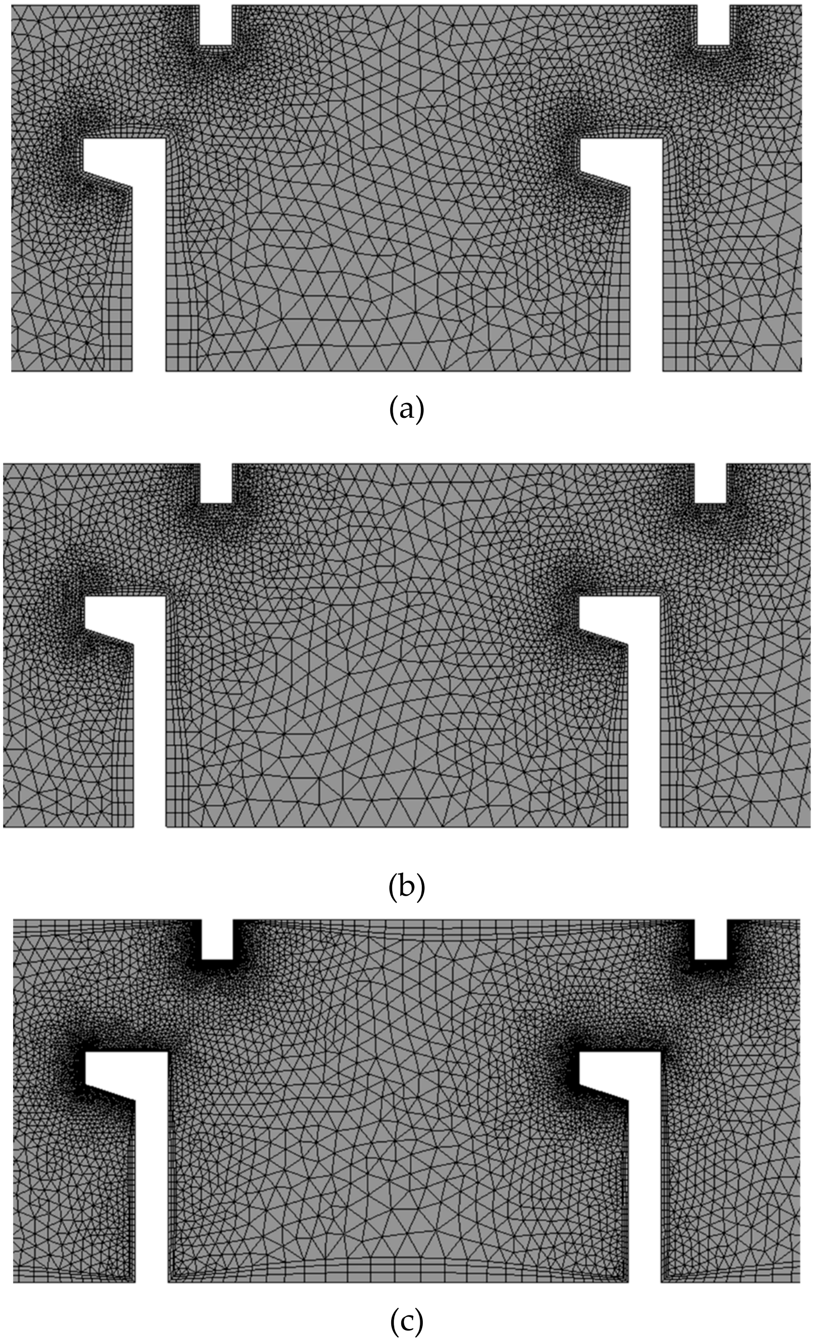

Figure S2. Details in one pool of the three meshes tested: (**a**) coarse mesh (1.8×10^6 elements); (**b**) medium mesh (2.7×10^6 elements) and (**c**) fine mesh (6.1×10^6 elements). The medium mesh was the selected for the simulations.

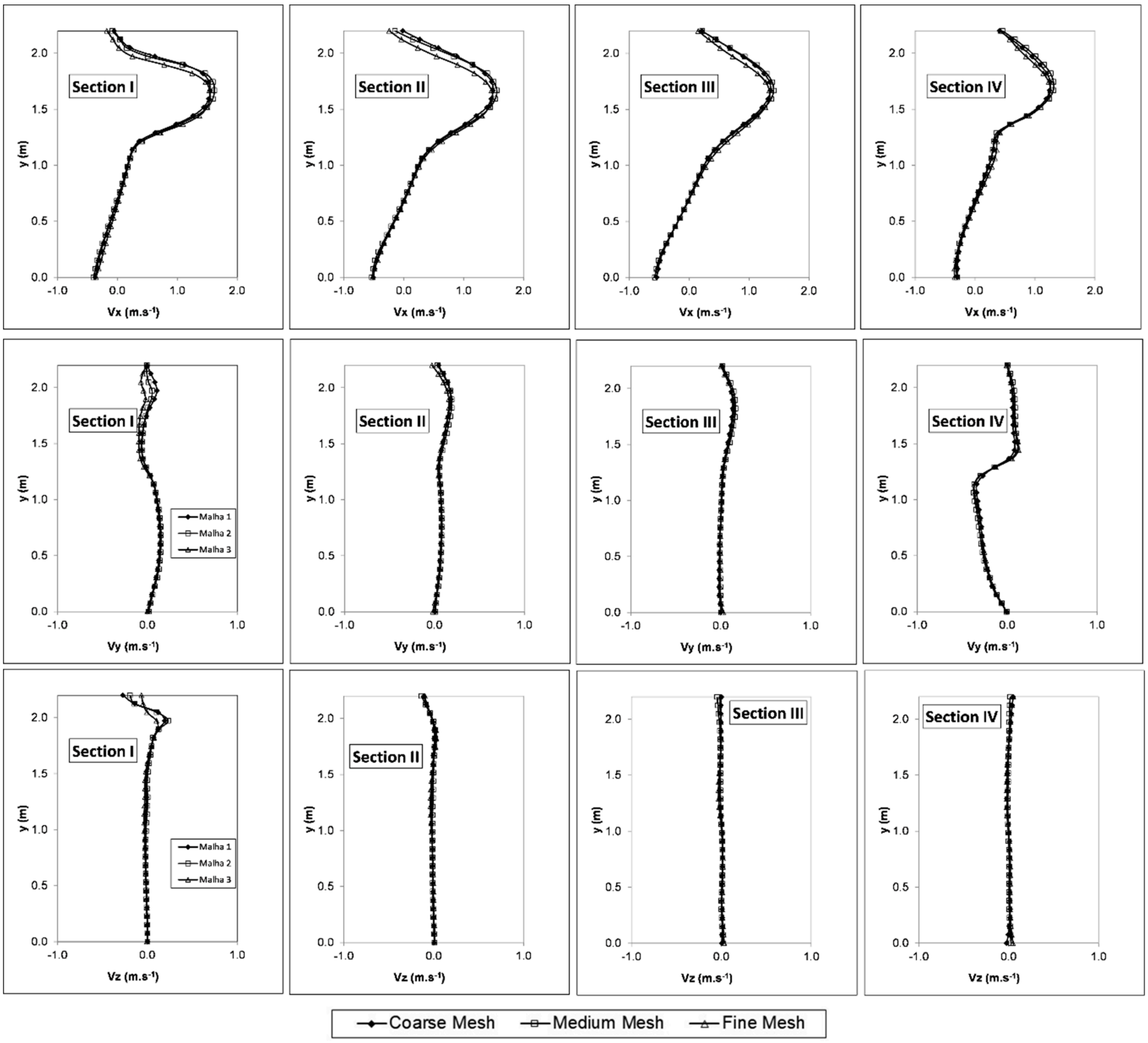

Figure S3. Longitudinal velocity, Vx (m·s^{-1}), transversal velocity, Vy (m·s^{-1}) and z-component of velocity, Vz (m·s^{-1}), at four cross sections I, II, III and IV (according Figure 3), in the plane z/h = 0.4 from the bottom, for discharge 0.800 m·s^{-1}, design D1S1 obtained through three meshes: coarse, medium and fine.

S4. Model Validation

In an initial analysis, the results of the simulations performed in this study were compared with experimental results and the numerical simulation presented by Bombač et al. [15], with the same geometry, slope and discharge. Figure S4 presents the results of the mean velocities in the longitudinal direction (Vx) and the transverse direction (Vy) of the pool's fishway, obtained for geometry D1S1, compared to experimental and numerical results presented by Bombač et al. [15]. In this figure, the values refer to four cross sections: I, II, III and IV (x = 0.60 m, 1.20 m, 1.80 m and 2.40 m) for a plane parallel to the bottom at the position z/h = 0.4 (z is the distance from the bottom to the plane, and h is the mean depth of the flow, measured in the middle of the central pool). It can be observed that there is a good agreement between the results obtained in the present study and the values measured experimentally.

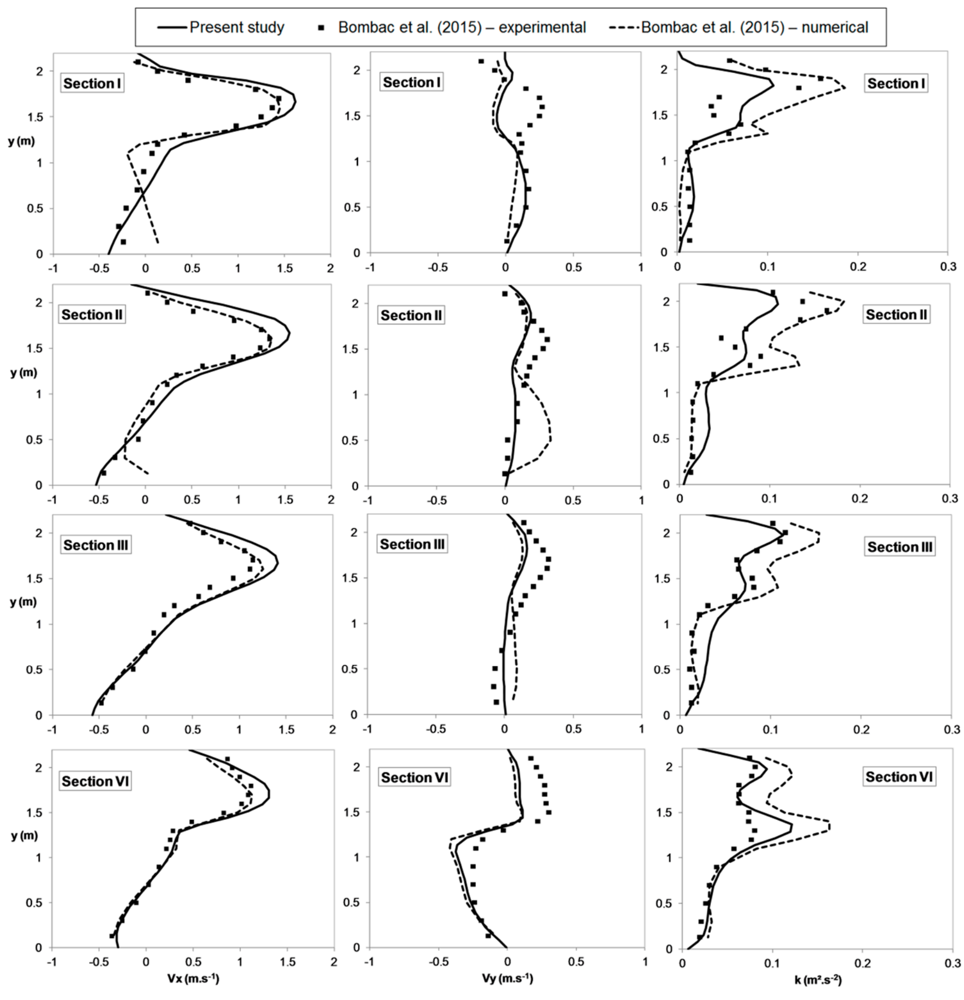

Figure S4. Comparison of Vx, Vy and turbulence kinetic energy (k) simulated in the present study, for the design D1 (Figure 2a), slope S = 1.67%, Q = 1.000 m^3/s with the numerical and experimental results of Bombač et al. [15] in four cross sections (I, II, III and IV) presented in Figure 3.

The analysis of the longitudinal velocities (Vx) indicates that the simulations carried out in the present study represent well the main jet, where the maximum velocities are located, as well as the region opposite the main flow, which represents the recirculation zone between the larger baffles. The maximum values of the mean velocities in the longitudinal direction, Vx, were higher than those obtained by Bombač et al. [15] by up to 12% in section I and 18% in section IV. In the regions adjacent to the main jet, the results of the present work found experimental velocities with differences smaller than those observed in the main jet. In the analysis of the transversal velocity component, Vy, the values of the present study were very close to those of the numerical simulation and to the experimental values of Bombač et al. [15].

The comparison of our turbulence kinetic energy results with the experimental data of Bombač et al. [15] indicates that our simulations properly represent this flow characteristic (Figure S4). The analysis of these results and the evaluation of the mean velocities of the flow allow us to conclude that the numerical model proposed in this work, as well as the boundary conditions, turbulence model and mesh, are adequate for the simulation of flow in these structures, since there is good agreement between the simulation results of the present study and the experimental data of Bombač et al. [15].