O 130. CFD SOLUTION OF DAM SPILLWAYS

Ahmet Bahadır Ceylan[1], Muhammed Uçar[2], Şerife Yurdagül Kumcu[3]

[1]*Undergraduate Student of Civil Engineering Department, Necmettin Erbakan University*
[2]*Research Asistant, Necmettin Erbakan University, Civil Engineering Department*
[3]*Assoc. Prof.Dr., Necmettin Erbakan University, Civil Engineering Department*

E-mail: yurdagulkumcu@gmail.com

ABSTRACT: Since the design of each dam in hydraulic engineering is special by its shape, purpose, basin characteristics and topography of the dam and so it cannot be made any unque standard project. It is very important for security to test the spillways of the dams which are safety structures of the dams by physical and / or mathematical modeling methods before the construction. In physical modeling, although the effect of scale, time and laboratory conditions, assumptions made in mathematical modeling and initial investment cost are restrictive, it is very useful in prediction of the problems encountered during modeling and construction phase. During physical model studies, a series of experimental studies were conducted; speed profiles, key curve, water surface profiles and pressures at various conditions, working conditions of the energy breaking pool are measured. The development of computer technologies and recent advances in numerical solutions lead engineers to do numerical modeling. In the scope of this study, the hydraulic properties of Alpaslan 2 Hydroelectric Power Plant (HEPP) Project were investigated experimentally with 1/70 scale physical model in the laboratory. Flow depth, flow discharge and pressure readings were measured under the different current conditions. Computational Fluid Dynamics (CFD) simulation was also tested to see if there is a suitable solution for the modeling of numerical modeling in spillway flow. In mathematical modeling, FLOW-3D program was used to solve the Reynolds-averaged Navier-Stokes (RANS) equations. The FLOW-3D program defines the cells in the calculation area by defining partially or completely filled cells. At the end of the study, the results of the project and the results of both models were compared. For the comparison of physical modeling and numerical modeling, the rating curve, flow profiles in the discharge channel, cavitation number, velocity and pressure measurements and energy dissipation structure flow conditions were used. The results obtained from physical and numerical modeling were found to be overlapping each other.

Key words: Spillway design, physical modelling, CFD solutions, spillway modelling, Flow 3D