Geophysical Research Abstracts
Vol. 19, EGU2017-18472, 2017
EGU General Assembly 2017

Analysis of a hydraulic a scaled asymmetric labyrinth weir with Ansys-Fluent

Andrés Humberto Otálora Carmona (1) and Germán Ricardo Santos Granados (2)

(1) Escuela Colombiana de Ingeniería Julio Garavito, Bogotá, Colombia (andres.otalora@escuelaing.edu.co), (2) Escuela Colombiana de Ingeniería Julio Garavito, Bogotá, Colombia (german.santos@escuelaing.edu.co)

This document presents the three dimensional computational modeling of a labyrinth weir, using the version 17.0 of the Computational Fluid Dynamics (CFD) software ANSYS - FLUENT. The computational characteristics of the model such as the geometry consideration, the mesh sensitivity, the numerical scheme, and the turbulence modeling parameters. The volume fraction of the water mixture - air, the velocity profile, the jet trajectory, the discharge coefficient and the velocity field are analyzed.
With the purpose of evaluating the hydraulic behavior of the labyrinth weir of the Naveta's hydroelectric, in Apulo - Cundinamarca, was development a 1:21 scale model of the original structure, which was tested in the laboratory of the hydraulic studies in the Escuela Colombiana de Ingeniería Julio Garavito.
The scale model of the structure was initially developed to determine the variability of the discharge coefficient with respect to the flow rate and their influence on the water level. It was elaborate because the original weir (labyrinth weir with not symmetrical rectangular section), did not have the capacity to work with the design flow of 31 m^3/s, because over 15 m^3/s, there were overflows in the adduction channel. This variation of efficiency was due to the thickening of the lateral walls by structural requirements.
During the physical modeling doing by Rodríguez, H. and Matamoros H. (2015) in the test channel, it was found that, with the increase in the width of the side walls, the discharge coefficient is reduced an average by 34%, generating an increase of the water level by 0.26 m above the structure.
This document aims to develop a splicing methodology between the physical models of a labyrinth weir and numerical modeling, using concepts of computational fluid dynamics and finite volume theories. For this, was carried out a detailed analysis of the variations in the different directions of the main hydraulic variables involved in the behavior, such as, the components of the velocity and the distribution of pressures,
For the numerical development, we worked with ANSYS - FLUENT software modeling version 17.0. Initially, a digital model of a conventional triangular weir with a vertical angle of 102° was developed in order to find the most appropriate numerical scheme and conditions. The numerical results were compared with conventional theories, evaluating the path and discharge coefficient.
Subsequently, one of the five cycles that compose the labyrinth weir was simulated, evaluating the behavior of the discharge coefficient, the water level, the streamline and the velocity field, with the purpose of understanding the hydraulic variables that are related in these geometries.
According to the previous results, the numerical modeling of labyrinth weir was performed, comparing the obtained results with the data of the physical scale model, analyzing the variation of the discharge coefficient, the streamline, velocity field, pressure distribution and shear stress.
Finally, based on the lessons learned from physical and numerical modeling, a methodological guide was created for any user with a computational and hydraulic fluid mechanics knowledge to develop a good practice of a computational and physical modeling.