10[th] International Congress on Civil Engineering, 5-7 May 2015
University of Tabriz, Tabriz, Iran

Numerical Investigation of Turbulent Flow over Ogee Stepped Spillways

Azam Dolatshah[1], Hasan Imani Khoshkho[2], Mahmoud Mashal[3], Najaf Hedayat[4]
1, 4- Islamic Azad University, Dezful Branch, Dezful, Iran
2- Islamic Azad University, Harsin Branch, Harsin, Iran
3- University of Tehran, Tehran, Iran

azam.dolatshah@gmail.com

Abstract

Present study was conducted with application of numerical methods for investigating the turbulent flow over ogee stepped spillways. For this purpose, the Finite Volume-Finite Difference Method (FV-FDM) was employed for numerical modelling of an ogee stepped spillway physical model belonging to a previous research study. Various parameters over the steps like water surface profile, pressure, and velocity of flow over steps were studied. The perfect conformity of numerical and laboratory results is clearly observed both visually and statistically. Then, as a result of development of the numerical model, steps' dimensions and geometries were changed and various spillways with smaller, larger, curved, obstacle, and steeped steps (and even a smooth ogee spillway) were investigated. Different properties of flow over steps were studied more and energy dissipations were obtained using the relevant hydraulic formulation. Results were compared with each other to survey the flow over the spillways. Results show that specific forms of steps make the flow more turbulent, and dissipate the energy better in addition to reduce the cavitation risk.

Keywords: stepped spillways, turbulent flow, Computational Fluid Dynamics (CFD), Finite Volume Method (FVM), Finite Difference Method (FDM).

1. INTRODUCTION

A stepped spillway is a hydraulic structure, which is an integrated part of the dam, allows the safe passage of overtopping flows. Due to increased air entrainment (i.e. aeration flow), stepped spillways dissipate the energy of the flow. This means that they reduce the level of energy in dissipater structures downstream from the spillway and lower the risks of cavitation. In recent years, the development of the Roller Compacted Concrete (RCC) technique has improved stepped spillways due to the low-cost and relatively high-speed construction this method enables.

Most studies investigating the flow over stepped spillways have been undertaken utilizing both laboratory experiments on scaled models and analytical and numerical approaches. For example, Degoutte et al. [1] suggested that the energy dissipation is higher in the jet flow than the skimming flow. They also discovered that there are two types of jet flow: one with fully developed hydraulic jumps and the other one with partially developed hydraulics jumps. Gonzalez and Chanson conducted an experimental study to gain a better understanding of the flow properties in stepped chutes with slopes typical of embankment dams. Their work yielded a new design procedure including some key issues not foreseen in prior studies [2]. Pfister and Hager presented visual observations made with a high-speed camera and air concentration measurements in the vicinity of the pseudo-bottom air inception point on a stepped model spillway [3]. Felder and Chanson conducted a physical study in a moderate slope-stepped chute and tested five configurations. The results were compared in terms of flow patterns, energy dissipation, and flow resistance [4]. Chanson and Felder also studied the two-phase gas-liquid flow properties of high-velocity open channel flows in a large-size channel experimentally. The results demonstrated high levels of turbulence in the high-speed, highly turbulent free-surface flows [5]. Barani et al. [6] used the feasible direction method and a wooden physical model to determine the optimum slope and step height of a stepped spillway and to calculate the dissipated energy of the flow. Si-ying et al. [7] studied the effect of aerator type on countering cavitation damage in the stepped chute at the Murum Hydraulic Power Station. They optimized the shape of the aerator and observed the aeration effects by hydraulic modelling.

With the advent of high-performance computers and the development of robust Computational Fluid Dynamic (CFD) software, the development of complementary analytical tools for resolving the intricacies of the flow pattern have become feasible, for examples see Chen et al. [8], Tabbara et al. [9], and Naderi Rad