Numerical Computation of Inception Point Location for Steeply Sloping Stepped Spillways

Mohammad Sarfaraz[1], Jalal Attari[2] and Michael Pfister[3]

1- Graduate Student, Department of Civil Engineering, Sharif University of Technology, Iran
2- Assistant Professor, Power and Water University of Technology, Iran
3- Ph.D., Laboratory of Hydraulic Constructions (LCH), Ecole Polytechnique Fédérale de Lausanne (EPFL), Lausanne, Switzerland

mohammad.sarfaraz@gmail.com

Abstract

Stepped spillways have gained much interest in recent decades because of their compatibility with Roller Compacted Concrete (RCC) dams. Hydraulics of stepped spillways is not simple considering different flow regimes and regions along the chute. Estimation of flow characteristics of the stepped chutes is presently carried out by using some empirical formulae and physical modeling, which are partially inaccurate if scaled up due to scale effects produced by high interaction between air and water. However, this can be improved by application of Computational Fluid Dynamics (CFD) models. The inception point of aeration of stepped spillways is placed further upstream than on smooth spillways. The position of this point is relevant to cavitation potential, flow losses and flow depth; hence an accurate approximation of the inception point location is essential. In this paper, flow characteristics with emphasis of investigating air concentration in a stepped spillway having a steep slope, was computed by a commercial CFD program. A comparison of the numerical and physical model results showed a relatively good agreement. The study indicates that the turbulence numerical simulation is an effective and useful method for the complex stepped spillway overflow.

Keywords: Stepped Spillway, Air Entrainment, Inception Point, Numerical Modeling, Large Eddy Simulation

1. INTRODUCTION

Stepped spillways are found to be effective for energy dissipation of excess flood released from dams. Many studies have shown that favorable design of stepped spillways can decrease the size of the stilling basin significantly and thus saving on construction costs [1, 2]. Stepped spillways have gained much interest in recent decades because of their compatibility with Roller Compacted Concrete (RCC) dams, hence having a steep slope of more than 50°. Once a stepped chute is located on the body of a RCC dam, it offers additional constructional and economic advantages [3].

On stepped chutes with skimming flow regime, the flow is highly turbulent. Once the outer edge of the turbulent boundary layer reaches the free surface, natural air entrainment commences [2]. L_i designates the distance, along the chute, between the ogee crest and the section where the air entrains the free surface. The thickness of the aerated zone then increases, protruding to the steps, so that the entire flow turns out to be aerated [4]. L_{ib} assigns the distance, along the chute, between the ogee crest and the section of developing fully aerated flow (Figure 1). Sufficient air entrainment about 5% at the spillway surface eliminates cavitation risk and will affect training wall design [3, 4]. Hence a proper estimate of this parameter is essential in the first design of stepped spillways, to assure that air entrainment will appear on the chute and its magnitude is high enough to prevent cavitation damages.

Engineers and researchers are often challenged with stepped spillways. There are only few empirical formulae for estimating inception point location (L_i), which can be used for wide range of stepped spillways [e.g. 5, 6]. Also because of complex existing hydraulic of air-water flow in stepped spillways, no explicit criterion has been presented to design in term of air concentration distribution to prevent cavitation risk [3, 5,].

Physical modeling is limited in investigating flow characteristics in all spatial details, because intrusive measurements in the step niches will disturb the vortex generated there. On the other hand, due to highly aerated flow in stepped spillways, scaling up their physical modeling results will result in scale effects if not adequately performed [2, 3]. Nowadays, with availability of high performance computers and CFD codes, flow characteristics over hydraulic structures can be accurately estimated, which are highly needed for design purposes. The aim of this research is to (1) assure that numerical modeling can be reliable as physical