9th International Congress on Civil Engineering, May 8-10, 2012
Isfahan University of Technology (IUT), Isfahan, Iran

Numerical simulation of flow pattern around spur dikes series in rigid bed

Hossein Basser[1], Abdollah Ardeshir[2], Hojat karami[3]
1- M.Sc. graduate of Hydraulic Engineering, Amirkabir University of Technology
2- Associate Professor, Amirkabir University of Technology
3- Assistant Professor, Semnan University

basser@aut.ac.ir

Abstract

Spur dikes are used to protect river banks from erosion and also keep the main channel navigable. These structures are built from the river bank into the stream flow and usually they are built in group. A spur dike may be submerged during the flood conditions or to be exposed during the low flow. Spur dikes may be classified into two types: impermeable and permeable. Present paper illustrates results of numerical study on flow field around a series impermeable and non-submerged spur dikes with the vertical attitude to the flow axes. In this study the CFD program namely FLOW-3D has been used to simulate the flow field around the spur dikes. In additions, several turbulence models have been applied in order to achieve the best numerical simulation. Finally longitudinal, transverse and vertical component of flow velocity, velocity vectors and streamlines caused by numerical simulation were compared to experimental results. Achieved results can prove that Flow pattern simulation around the spur dikes could be done by FLOW-3D numerical modeling, very well.

Keywords: Numerical simulation, flow pattern, spur dikes, turbulence model, FLOW-3D.

1. INTRODUCTION

River banks are often exposed to erosion and destruction. Destruction of banks depends on the type of erosion and characteristics of banks like their shapes and their mechanical properties. Using spur dikes as an indirect technique for erosion protection is one of the common and economic methods. A spur dike is a structure that projects from a stream bank into the river channel and causes a redirection of flow away from the bank toward the tip of the spur dike. These structures are usually preferred to be built in group. Spur dikes benefit the stream by reducing velocities near the banks and create still water areas that encourage deposition and channeling flow to reduce the width and create a defined channel. Construction of spur dike against the flow causes significant changes in flow pattern in channel. For example, spur dike causes a difference in hydrostatic pressure at upstream and downstream of the structure which will cause a whirlpool disturbance around it. These changes result in scour phenomenon around spur dikes which may lead to structure failure. Therefore investigating the characteristics of flow pattern around spur dikes have been a great interest in river engineering. Numerous researchers like:, Hao Zhang et al (2009), Beheshti (2010), Duan (2009), Naji (2010), Karami (2011) [1,2,3,4] made a variety of experiments in order to determine the flow pattern around spur dikes. Most of these researchers studied effect of single spur dike, while using series of spur dikes is more effective in protection of rivers. Besides experiment studies, variety of CFD models have been developed for computing flow pattern around hydraulic structures; like Fluent, FLOW-3D and SSIIM. Herein FLOW-3D numerical modeling has been used to investigate flow pattern around a series of spur dikes and streamlines and components of velocities have been studied.

2. EXPERIMENT

For verification the numerical model, the results of experiment which was carried out by Karamil (2010) was used. The experiment was carried out at the porous media laboratory of Amirkabir University of Technology (Tehran polytechnic). A rectangular section flume of 14 m length, 1 m width and 1 m depth was used for the experiments. The bed and sides of the flume were made of glass supported by a metal frame. Three 25 cm length, impermeable, non-submerged and perpendicular to the flow direction spur dikes were installed in the flume. After the first spur dike was installed in 6.16 meter distance from the flume entrance, the space