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

Numerical modeling of scouring around rectangular abutment

Hossein Basser[1], Abdollah Ardeshir[2], Ruhollah Cheraghi[3]
1- M.Sc. graduate of Hydraulic Engineering, Amirkabir University of Technology
2- Associate Professor, Amirkabir University of Technology
3- M.Sc. graduate of Hydraulic Engineering, Amirkabir University of Technology

basser@aut.ac.ir

Abstract

Present paper illustrates results of numerical study on scouring phenomenon around rectangular cross section, impermeable and non-submerged bridge abutment with the vertical attitude to the flow axes. In this study the CFD program namely SSIIM 2.0 has been used to simulate the scouring around the abutment. The SSIIM 2.0 numerical model is based on the finite volume method, using an unstructured grid with dominant hexahedral cells. Based on a sensitivity analysis on turbulence model the k-ε turbulence model with some RNG extensions was used to predict turbulence. The SIMPLE method was used to compute the pressure and Power Law scheme was used to discretize the equations. The sediment transport was computed as bed load in addition to solving the convection-diffusion equation for suspended sediment transport. In additions, variety of grids was applied in order to numerical simulation, and the best grid was chosen by means of available experimental results. Finally, bed changes and scouring phenomena caused by numerical simulation were compared with experimental results. It could be found from achieved results that scouring simulation around the rectangular abutment could be done well by SSIIM 2.0 numerical modeling.

Keywords: Numerical modeling, scouring, abutment, SSIIM 2.0.

1. INTRODUCTION

Local scour at bridge structures has been extensively studied over the past fifty years with both experimental and numerical methods. When an obstacle is placed in a flow on an erodible bed, a scour hole forms at the footing of the obstacle. On river beds, this phenomenon typically occurs in the vicinity of bridge abutments and bridge piers, often leading to the structure collapse.

Construction of an abutment against flow causes a difference in hydrostatic pressure at upstream and downstream of the structure which will cause a whirlpool disturbance around it. These whirlpool flows account for the main local scoring mechanism which produces large vortexes at the vicinity of abutment and this phenomenon may lead to structure's failure.

Local scour hole is formed around abutment due to the action of flow against these obstructions. Estimation of the depth of scour at the vicinity of abutment has been the main concern of engineers and researchers for years. Therefore, knowledge of the anticipated maximum depth of scour for a given discharge is a significant criterion for the proper design of an abutment foundation or utilizing a method for decreasing scour around the structure. Numerous researchers like: Dey (2005), Chiew (1992), Mashahir et al (2004), Hua et al (2006), Kayaturk (2004), Molinas et al (1992), Melville (1992) and Kumar (1990) made variety of experiments in order to investigate the scour phenomenon around vertical abutment [1,2,3,4,5,6,7,8,9]. Beside experiment studies, variety of CFD models have been developed for computing sediment transportation and calculating bed changes in channels and around hydraulic structures or obstructions; like SSIIM, Fluent and Flow-3D. In the present study, SSIIM 2.0 three-dimensional model was used to compute sediment transport and scouring phenomenon around rectangular abutment and its capability for simulation scour around hydraulic structures was investigated.

2. EXPERIMENT

For verification the numerical model, the results of experiment which was carried out by Cheraghi, et al (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