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 between them was fixed equal to twice of the spur dike length, (2L, L is spur dike's length). The flow discharge was regulated by an inlet valve. It was measured by a rectangular weir at the flume. An Acoustic Doppler Velocimeter (ADV) was used to measure the three components of velocity in longitudinal, vertical and lateral directions. Figure (1) indicates the experimental spur dikes and ADV in laboratory. Figure (2) shows a schematic view of the experimental flume and the spur dikes.

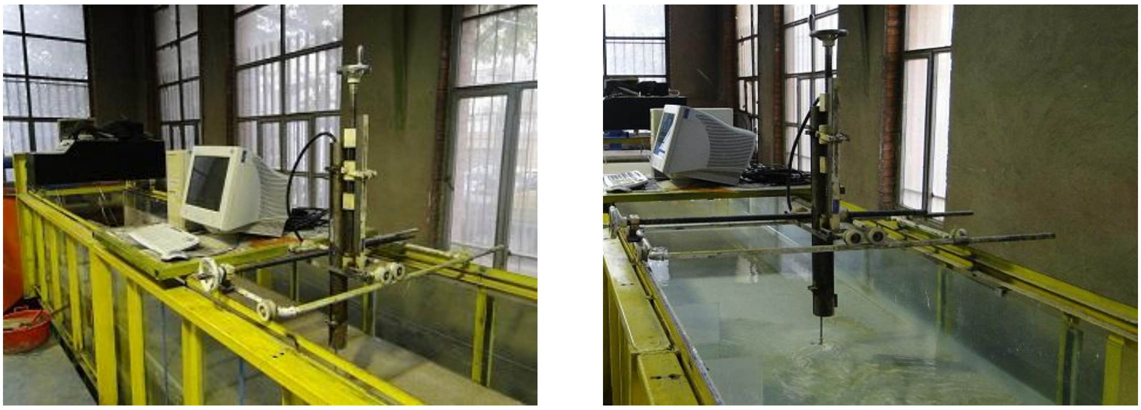


Figure 1. Experimental spur dikes and ADV in Laboratory

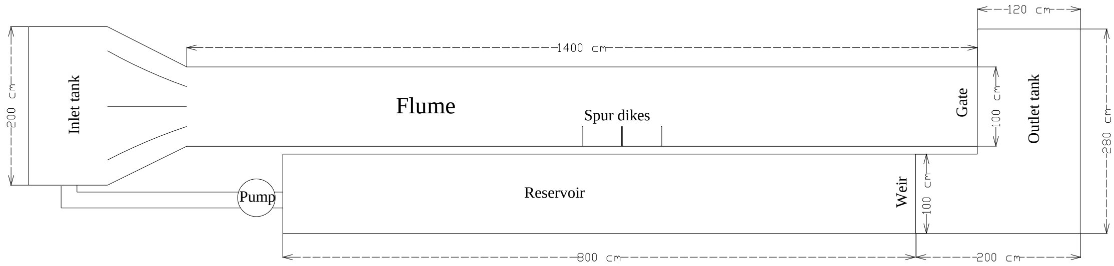


Figure 2. Schematic view of the experimental flume and spur dikes

3. NUMERICAL

In the present study a CFD model namely FLOW-3D was used to solve the flow around spur dikes. The commercially available computational fluid dynamics software, FLOW-3D solves the transient Navier-Stokes equations by the volume-of-fluid method [5]. In addition to the basic abilities, the model supports several turbulent closure using a number of advanced and widely accepted numerical schemes [6,7,8] for simulating turbulent flows. These include Prandtl's mixing length theory, one equation turbulent energy (k), the two-equation (k-ε), Renormalization-Group (RNG), and Large Eddy Simulation (LES) closure schemes. The RNG k-ε model of Yakhot and Orszag (1986) is an extension of the k-ε model [7,9]. However, it requires less reliance on empirical constants and provides a better solution in areas affected by high shear [10]. Standard values of turbulence parameters [7,9] have been used to develop FLOW-3D. The model also has a number of other features, such as the ability to construct non-uniform grids, automatic time-step selection, graphical post-processing, etc. Several turbulence models have been used in simulating three-dimensional flow field around spur dikes. Mendoza-Cabrales (1993) used the k-ε turbulence closure model to solve three-dimensional flow around circular piers and computed the associated bed shear stress [11]. Melville (1975) found a large discrepancy between the experimental data and his numerical model results showing inadequacy of the k-ε turbulence model for vertical three-dimensional flow [12]. Smith and Foster (2005) compared the result of LES and the standard k-ε turbulence model to solve three-dimensional flow around a cylinder over a scoured bed and predicted the accuracy of the k-ε model [13]. Haltigin et al. (2007) and Bradbook et al. (2001) showed that the RNG k-ε model gives a good estimation of the velocity field [14,15]. The present study will examine various turbulence models using the FLOW-3D software and the appropriate turbulence model will be used for simulating flow field around the spur dikes.
In this study the geometry of channel was made in Auto Cad an then it transferred to FLOW-3D for generating the grids in channel. After generating the grid a sensitivity analysis was carried out for investigation of effect of various turbulence models. It appeared that the RNG k-ε model and standard k-ε model, performed better than the Prandtl's mixing length model and one equation model. Then the two

equation; k-ε model was used. Based on a sensitivity analysis of effect of two k-ε standard and k-ε with some RNG extensions turbulence models, the k-ε turbulence model with some RNG extensions showed the best agreement with experiment measurements. Table (1) indicates the amount of the errors for the turbulence models at several sections around first spur dike for three velocity components. It can be seen that the k-ε RNG model shows best agreement with measured velocities. After selecting the k-ε RNG turbulence model the flow pattern around spur dikes was simulated. The results are showed in next part.

Table 1- Amount of errors between computed velocities and measured velocities around first spur dike

Velocity component	U	V	W
k-ε standard	6.8	22	29
k-ε RNG	8.5	28	36

4. NUMERICAL RESULTS

Figure (3) indicates comparison of numerical values with experimental values and Prandtl values at x=5.5 m , y=0.5 m, it can be seen that numerical model could simulate the velocity in flow direction with sufficient accuracy. Figure (4) illustrates amount of computed velocities components and measured velocities components around first spur dike at two sections, and figure (5) indicates the streamlines achieved from experimental method and numerical method. These results indicate the accuracy of the numerical model in calculating the flow pattern around spur dikes.

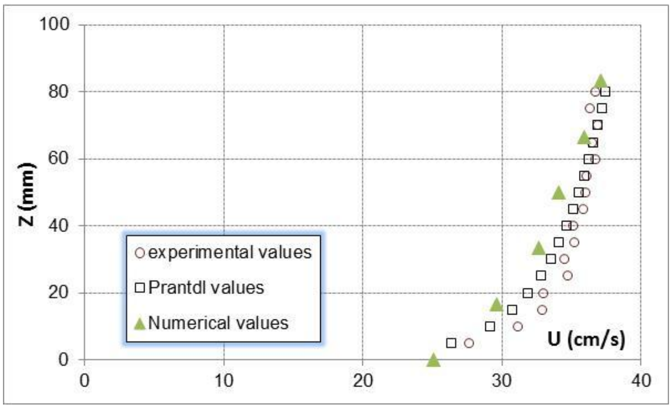
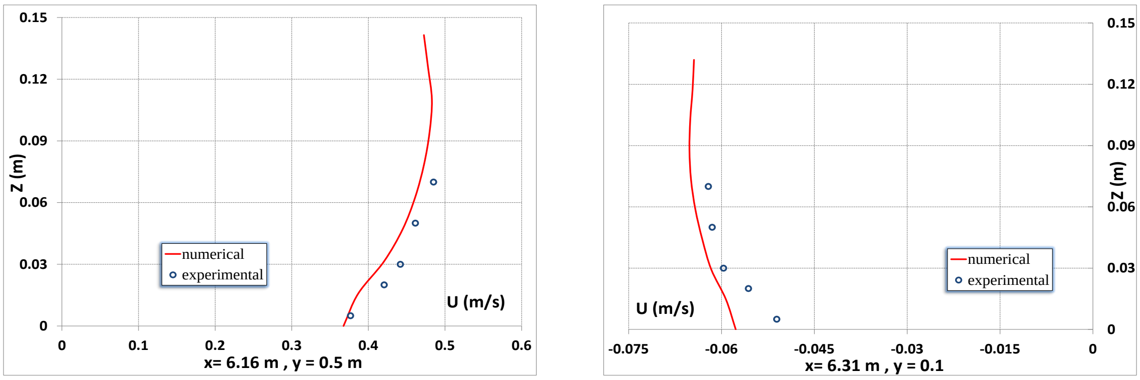
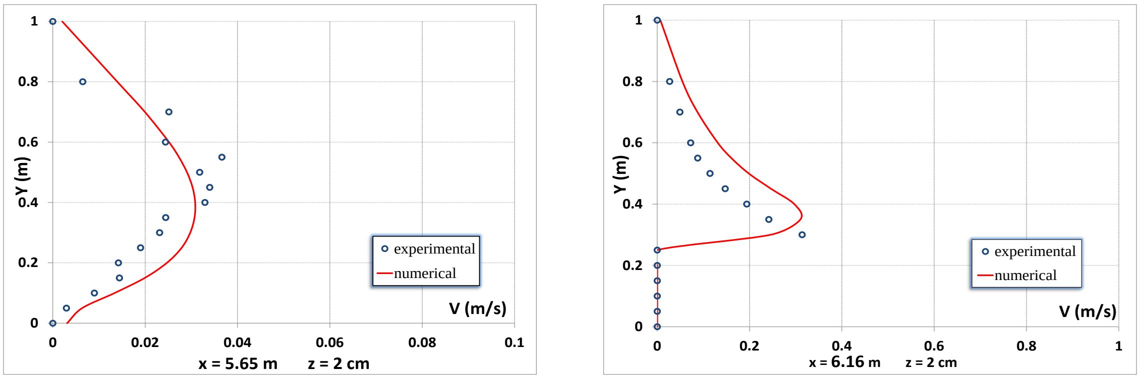


Figure 3. Comparison of numerical values with experimental values and Prandtl values at x=5.5 m , y=0.5 m.

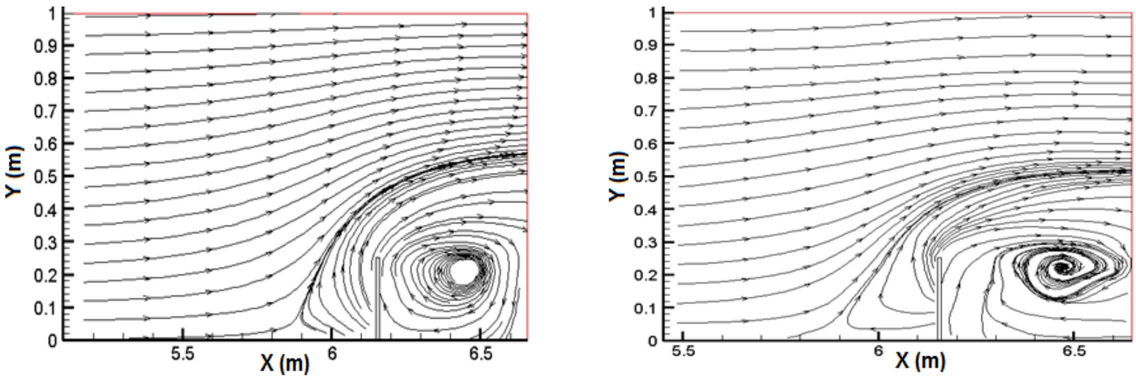


(a)



(b)

Figure 4. Comparison of computed velocities and measured velocities around first spur dike: (a) U (velocity in flow direction) (b) V (velocity in lateral direction)



(a) **(b)**

Figure 5. Stream lines at z=2cm : (a) Numerical (b) Experimental

It can be seen in figure (5) that there is a backward water area at upstream side of the first spur dike. It can be seen in this figure that a still area and clockwise horizontal vortex is formed within the zone between two spur dikes.

Encountering the approaching flow to the first spur dike caused a difference in pressure between the water surface and bed level, which made the flow move downward, then moved in the opposite direction of approaching flow. This phenomenon caused the formation of principal vortex, which is considered as the main reason for scouring around spur dikes. Figure (6) indicates the downflow which achieved by numerical model. The results proved that the maximum amount of velocity was occurred at the tip of the first spur dike and at downstream side of that. Figure (7) indicates the streamlines in X-Y plane, it can be seen that the dimension and power of vortexes is higher at the vicinity of bed surface in compare to water surface.

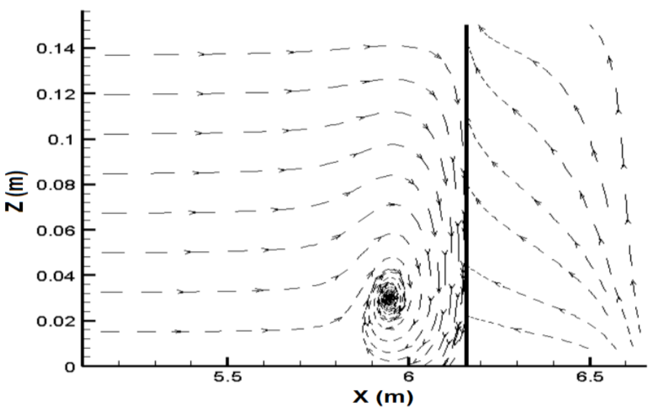


Figure 6. Downward flow at the upstream side of the first spur dike

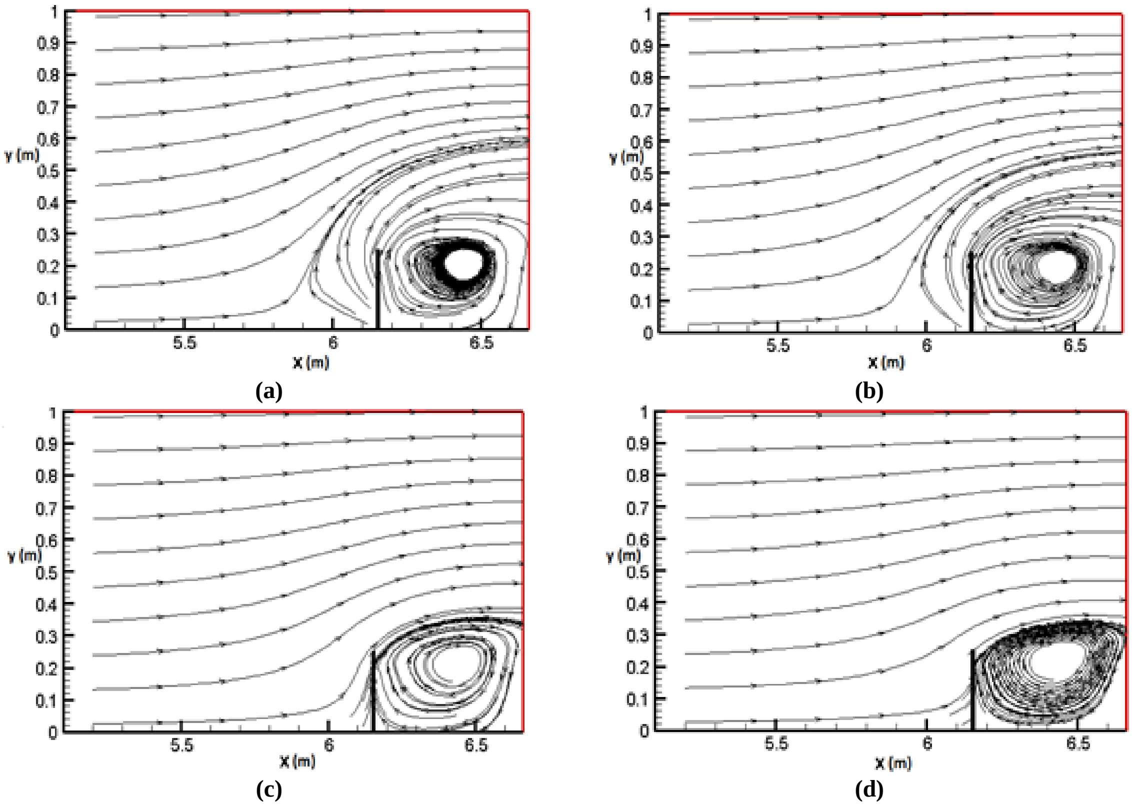


Figure 7. Stream lines at: (a) z = 0.5 cm (b) z = 2 cm (c) z = 5 cm (d) z = 8 cm

5. CONCLUSION

In this research, flow pattern around three non-submerged, impermeable, and perpendicular to the flow direction spur dikes was studied with numerical method. By comparing the 3D model with the experimental measurements, the model was found to reproduce flow pattern around spur dikes with sufficient accuracy. The following main results were found in this research:

- The FLOW-3D model was able to resolve the spur dikes correctly and reproduced the three-dimensional flow field around the spur dikes for the rigid bed condition.
- From the comparison of the simulated velocities with the measurements, the best turbulence closure scheme was investigated. It appeared that the RNG k-ε model and standard k-ε model, performed better than the Prandtl's mixing length model and one equation model for the complex flow field around the spur dikes for rigid bed condition and comparison of the results from the RNG k-ε model with the standard k-ε model showed that the predictions of the k-ε turbulence model with some RNG extensions agreed better than other turbulence models when compared with the existing experimental results for the rigid bed condition.
- It was achieved that maximum velocity in flow direction occurred at the downstream side of the first spur dike and close to the tip of the spur dike.
- It was achieved that the dimension and power of the vortexes in X-Y plane within the zone between two spur dikes is higher close to the bed in compare to water surface.
- It was observed that a downward flow was formed at the upstream corner of the first spur dike which is considered as the main reason for scouring around spur dike.

6. REFERENCES

1. Zhang, H., Nakagawa, H., Ishigaki, T., Muto, Y. (2005), ''Prediction of 3D flow field and local scouring around spur dykes.'' J .Hydraulic Engineering ,49, 1003-1008.

2. Beheshti, A.A., Ataei-Ashtiani, B. (2010), "Experimental Study of Three-Dimensional Flow Field around a Complex Bridge Pier." J.Eng.Mechanics, 136 (2), 143-154.

3. Duan, J.G. (2006), ''Three-dimensional mean flow and turbulence around a spur dike.'' proc .Hydraulic Engineering, ASCE.

4. M.Naji Abhari .er. al. (2010) "Experimental and numerical simulation of flow in a 90° bend" , Flow Measurment and Instrumentation, , 21, 292-298.

5. Hirt, C. W., and Nichols, B. D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries." J. Comput. Phy., 39, 201-225.

6. Launder, B. E., and Spaulding, D. B. (1972). Mathematical models of turbulence. Academic Press, New York, N.Y.

7. Rodi, W (1980). Turbulence models and their application in hydraulics: A state of the art Review. IAHR.

8. Abbot, M., and Basco, D. (1989). Computational fluid dynamics: An introduction for Engineers.

9. Yakhot, V., and Orszag, S. A. (1986). “Renormalization group analysis of turbulence.” J. Sci. Comp., 1(1), 3.

10. Sicilian, J. M., Hirt, C. W., and Harper, R. P. (1987). "FLOW-3D: Computational modeling power for scientists and engineers." Rep. FSI-8700- 1, Flow Science, Los Alamos, N.M.

11. Mendoza-Cabrales, C. (1993). "Computation of flow past a cylinder mounted on a flat plate." Proc., Hydraul. Eng., ASCE Reston, Va., 899-904.

12. Melville, B. W. (1975). „„Local scour at bridge sites.‟ ‟ Rep. No. 117, Dept. of Civil Engineering, School of Engineering., Univ. of Auckland, Auckland, New Zealand.

13. Smith, H. D., and Foster, D. L. (2005). “Modeling of flow around a cylinder over a scoured bed.” Journal of Waterway, Port, Coastal, and Ocean Engineering, 131(1), 14-24.

14. Haltigin, T. W., Biron, P. M., and Lapointe, M. F. (2007). “Predicting equilibrium scour-hole geometry near angled stream deflectors using a three-dimensional numerical flow model.” J. Hydraul. Eng., 133(8), 983–988.

15. Bradbrook, K. F., Lane, S. N., Richards, K. S., Biron, P. M., and Roy, A. G. (2001). “Role of bed discordance at asymmetrical river confluences.” J. Hydraul. Eng., 127(5), 351-368.