APCOM & ISCM
11-14th December, 2013, Singapore

Exploring Appropriate CFD Model and Impact Scale for Non-submerged Spur Dikes

Z.H. Gu[1], X.M. Cao[1], and Jane W.Z. Lu[2]*

[1]Colledge of Civil Engineering and Architecture, Zhejiang University, Zhejiang, China
[2]Department of Civil and Architectural Engineering, City University of Hong Kong, Hong Kong, China

*Corresponding author: bcwzlu@cityu.edu.hk

Spur dike is a common hydraulic structure, which is widely used in hydraulic engineering such as channel regulation, flood prevention and river diversion for preserving the desired water depth, deflecting the main current and protecting river banks. In actual operations, spur dikes exert influence on river system usually in the form of groups, and the flow field around spur dike group is complicated and not well understood yet [1-2]. Compared with experimental investigation, numerical simulation can provide more details with reasonable cost. However, it still depends on observed data as well as appropriate choice of turbulence models and boundary conditions [3-4]. In this study, firstly, the flow field around non-submerged spur dike groups which were eight spur dikes in staggered arrangement was selected as the research objects for understanding the performance of the CFD models of spur dike group flow based on flume experiments [5-6]. A set of flume experiment was conducted in laboratory, based on its observed data the numerical simulations were carried out through Finite Volume Method (FVM) and three turbulence models -- standard k-ε model, Reynolds Stress Model (RSM) and Large Eddy Simulation (LES). In each numerical model, the free surface boundary respectively employed two methods -- rigid-lid assumption and VOF (volume of fluid) model. The comparisons between computation results from CFD and observed data from flume experiment showed that the three turbulence models can simulate the three-dimensional flow around spur dike group in a certain extent. Specifically, the simulation effect of standard k-ε model and RSM has no significant difference under either rigid lid assumption or VOF method, both of them can preferably simulate flow field of mainstream area, while the simulation performance of recirculation zone near the spur dikes is poorer relatively; in contrast, the simulation effect of LES is best especially in the region of recirculation. For computation time, k-ε model is shortest while LES is longest, and the gap between the two is huge. Therefore, to rapidly evaluate the field characteristics of mainstream, standard k-ε model with rigid lid assumption is recommended; while to achieve the fine simulation of spur dike field, especially the flow pattern in recirculation zone, LES with VOF method is appropriate. On the basis of the previous, numerical models based on standard k-ε model, finite volume method and rigid lid assumption were built to classify the impact scale of non-submerged spur dikes on river systems. The conceptions of spacing threshold of non-submerged double spur dikes with ipsilateral layout and alternate layout in a straight rectangular channel were proposed respectively, and further quantitative investigation were carried out. Through dimensional analysis, three dimensionless indices -- Fr (Froude number), B/b (channel width to dike length) and B/h (channel breadth to water depth) were identified as the main influencing factors of the spacing threshold. Under the two spur dike layouts, both of the simulation results showed that Fr had the least impact among the three factors; Sc/b (dike spacing threshold to dike length) increased monotonically as B/b increased, and their relationship was close to exponential function; Sc/b decreased monotonically as B/h increased, and their relationship was power function approximatively. Furthermore, the dimensionless empirical formulas of spacing threshold of non-submerged spur dikes with ipsilateral layout and alternate layout were fitted by multivariate regression, and four sets of testing conditions were conducted by CFD model to verify the accuracy of the two empirical formulas. The verification demonstrated that calculation results from empirical formula and CFD were in good agreement, and the efficiency of the empirical formulas used in interpolation conditions were better than in epitaxial conditions. Comparing the two empirical formulas, it can be found that the spacing thresholds of alternate spur dikes are almost generally smaller than ipsilateral spur dike, which implied the flow passing by alternate spur dikes was easier to recovery under the same conditions. This work can provide a reference for the simulation of the turbulent flow with free surface. And the two empirical formulas can be used to determine the impact scale of spur dikes to river systems. They are very useful for the research on river health in macro-scale and cumulative effect to river systems from river projects-related. In the future, more factors, such as bottom slope of flume, erosion bed, spur dike direction, spur dike with head slope and abutment spur dike etc., will be considered to perfect the empirical formula and broaden its applicability to field conditions.

Keywords: Spur dike, CFD Model, Impact scale, Flume experiment, Dike spacing, Threshold

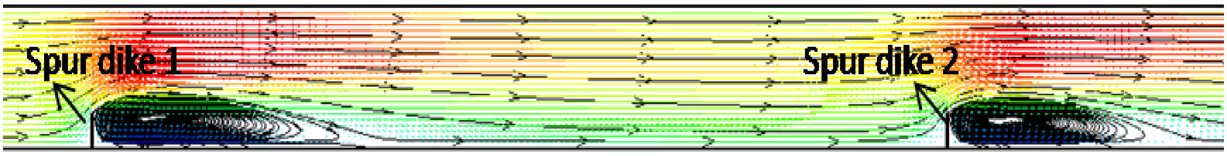

Figure 1 Flow structure around double spur dikes

Acknowledgment

The project supported by Hong Kong RGC-GRF Grant (CityU 118212), Strategic research grant, City University of Hong Kong [Projects No. CityU-SRG 7002718], National Natural Science Foundation of China (Grant No. 50909085) & Science and Technology Project of DWRZJ (Grant No. RC1106).

References

1. Acharya A, Duan JG (2011) Three dimensional simulation of flow field around series of spur dikes. World Environmental and Water Resources Congress 2011: 2085-2094
2. Azinfar H, Kells JA (2008) Backwater prediction due to the blockage caused by a single, submerged spur dike in an open channel. Journal of Hydraulic Engineering 134:1153-1157
3. Galvan S, Reggio M, Guibault F (2011) Assessment study of k-epsilon turbulence models and near-wall modeling for state swirling flow analysis in draft tube using FLUENT. Engineering Applications of Computational Fluid Mechanics 5(4):459-478
4. Han SS, Biron PM, Ramamurthy AS (2011) Three-dimensional modelling of flow in sharp open-channel bends with vanes. Journal of Hydraulic Research 49(1):64-72
5. Gu, Z.H., Liu, G.P., Tang, H.W., Zhou, Y.L., 2007. Study on numerical simulation of water transfer process in tidal river networks. Journal of Hydroelectric Engineering, 26(4), 76~85.
6. Wang, XY, Yang, QY, Lu, WZ, Wang, XK, 2011. Effects of bed load movement on mean flow characteristics in mobile gravel beds, Water Resources Management, 25(11): 2781-95, doi: 10.1007/s11269-011-9838-6