Numerical simulation of flow velocity around single and twin bridge piers with different arrangements using the Fluent model

Zahra Behrouzi [1], Hossein Hamidifar [1*], Seyed Mohammad Ali Zomorodian [1]

[1]Water Engineering Department, Shiraz University, Shiraz, Iran.

ABSTRACT

Flow structure around bridge pier is a very complicated phenomenon. Due to the special geometric and structural conditions in some cases, it is required that the piers be placed in pairs next to each other with special arrangements, which leads to a more complex flow structure around the piers. In this study, the variations of the flow velocity and turbulent kinetic energy around single and the twin bridge piers with a circular cross section is simulated using the Fluent model. The twin piers are placed in three configurations including tandem, side-by-side, and at inclined with the flow direction. The three-dimensional components of flow velocity, streamlines, and velocity contours have been investigated for both single and twin piers. By comparing the longitudinal velocity between measured and simulated conditions in two selected cross sections, the average error for the single tandem piers was 7.3% and 3.54%, respectively. Also, the longitudinal velocity in the tandem, side-by-side and inclined piers has decreased by 2.34% and 9.27% and increased by 87.8%, respectively, compared to the single pier conditions. In general, due to the minimum values of turbulent velocity and kinetic energy, the side-by-side model is recommended as the most appropriate arrangement of the piers with respect to the flow direction.

KEYWORDS

Flow pattern, velocity, bridge pier, simulation, Fluent.

[*] Corresponding Author: Email: hamidifar@shirazu.ac.ir

1. Introduction

The study of flow pattern around bridge piers is essential because of its effect on the structural failure. Generally, when a structure is located in flow, it creates some changes which increase the capacity of sediment transport, and finally, it causes the scour phenomenon. Ayari and Hakimzadeh (2009) simulated three-dimensional flow pattern around bridge piers with circular, spindle, oval, square, and hybrid (circular-rectangular) sections. They concluded that with increasing longitudinal dimension, the vortex length was reduced, and these vortices were shorter in the hybrid (square-rectangular) section than in the square and rectangular ones. Also, at the back of the spindle pier, there was no recirculating flow. Also, Poorahmadi and Hakimzadeh (2010) investigated the effects of turbulence models on two-dimensional numerical modeling of flow around circular piers using the Fluent software. They showed that the Reynolds Stress Model (RSM) could predict the flow separation around the pier and backflow eddy currents considerably. Ataie-Ashtiani and Aslani-Kordkandi (2013) studied the flow pattern around single and twin bridge piers in rough bed conditions. They measured the flow velocity using an ADV. Their results showed that the presence of a downstream base changes the flow structure significantly, especially in the vicinity of the flow edges. Kardan et al. (2018) simulated the flow around bridge pier models and found that the k-e model is more accurate than other models. In the present study, numerical simulation and comparison of flow pattern around single and twin bridge piers are investigated.

2. Methodology

In this study, the flow pattern has been simulated with Fluent software (v.16.0) around a single pier and two cylindrical piers with different arrangements. The piers were arranged in 3 ways: tandem, side by side, and inclined with the flow direction. The volume of fluid method (VOF) method has been used to study the effect of the free surface. Meshing was performed using Gambit software (V.2.4.6). A finer mesh was considered around the pier. The number of cells in the single-pier models was 1318482, and for the three other models were the same as 1355760. The study includes 3-D flow velocity components, streamlines, velocity vector, and contours of turbulence kinetic energy for single pier and twin piers. Also, the results of the single pier and two piers were compared with laboratory results.

3. Discussion and Results

Variations of the flow velocity in the streamwise direction (u) for single pier and twin piers are presented in Figures 1 and 2, respectively. The results show that fluent software is suitable in simulating flow around piers. The average error between measured and simulated longitudinal velocity in 2 selected cross-sections were 8 and 3.7% for single and twin piers with the tandem arrangement, respectively. Also, a good agreement was observed between the simulated and measured data in terms of velocity magnitude. The flow velocity increases around the pier due to the flow compression, which is well evident in the simulated results. The magnitude of this velocity is lower at 0.02h than that at 0.5h, as the flow velocity increases from the bed to the free surface. At the downstream of the piers, due to the formation of recirculating vortices, the flow is diverted in the opposite direction y, and its velocity is negative, which is well observed in both simulated and experimental models.

a)

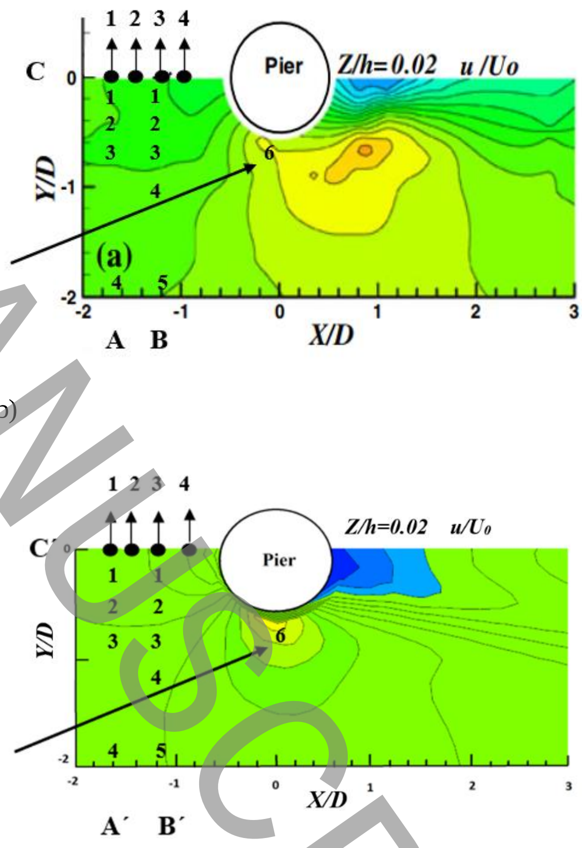

Figure 1. Velocity contours in the *x* direction for single pier, a) experimental, and b) simulated results

a)

b)

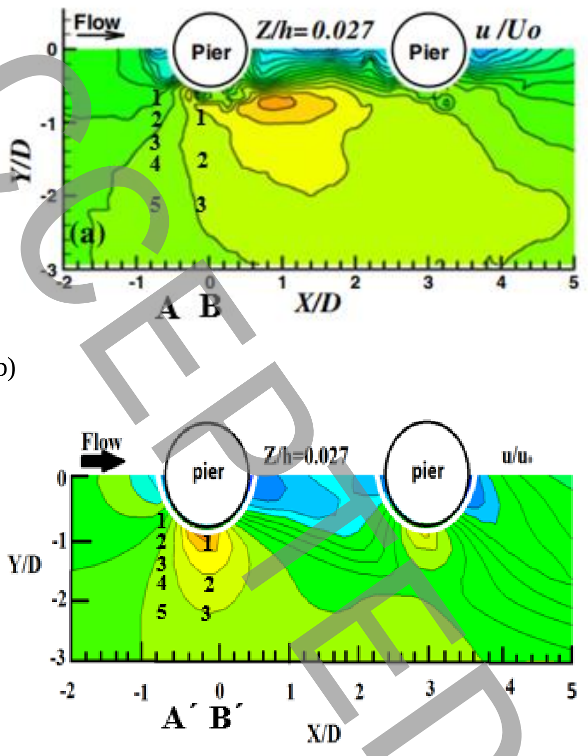

Figure 2. Velocity contours in the *x* direction for twin piers, a) experimental, and b) simulated results

4. Conclusions

By comparing the results, it is found that the best arrangement for piers placement in flow direction is side by side; because minimum velocity and lower kinetic energy occur, and therefore the lowest scour depth adjacent to the piers will be expected.

5. Acknowledgement

The authors are thankful to the staff of Rapid Computing Research Center, Shiraz University, Shiraz, Iran, for providing the computer facilities for the present study.

6. References

[1] Ayari, S. & Hakimzadeh, H., 2009. Investigation of bridge pier shapes on the flow pattern using fluent software. *Proceedings of the 8th International Congress of Civil Engineering*, Shiraz University, Shiraz, Iran. (in Persian)

[2] Poorahmadi, M. & Hakimzadeh, H., 2010. Investigation of flow pattern around cylindrical pier under steady flow using first and second order turbulence models. *Proceedings of the 5th National Congress of Civil Engineering*, Ferdowsi University of Mashhad, Mashhad, Iran. (in Persian)

[3] Ataie-Ashtiani, B. & Aslani-Kordkandi, A., 2013. Flow field around single and tandem piers. *Flow, Turbulence and Combustion*, 90(3), pp.471-490.

[4] Kardan, N., Hassanpour, N. & Hoseinzade-Dalir, A. H. (2018). Experimental and Numerical Investigation of Bed Erosion around Bridge Piers with Different Cross-Sections. *Irrigation and Drainage Structures Engineering Research*, 19(71), pp.19-36. (in Persian)