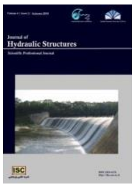
Journal of Hydraulic Structures
J. Hydraul. Struct., 2024; 10(1):119-126
DOI: 10.22055/jhs.2024.19078



The effect of slot in reducing scour depth of cylindrical bridge pier using Flow3D software

Milad Kazemian [*1]
Kazem Esmaeili [1]

Abstract

Scouring is one of the most important factors in the destruction of bridge piers. One of the ways to reduce scouring depth is to use a slot in the bridge pier. In this research, the Flow 3D model was used to simulate the effect of a slot with the same dimensions in two positions parallel to the bed and close to the water surface. The tests were performed at two relative speeds of 0.87 and 0.92. Uniform and non-cohesive bed sediments with a geometric standard deviation of 1.4 and an average diameter of 1 mm were used. The results showed that the slot created in the pier of the bridge weakens the eddies around the pier and is effective in reducing scour depth. The slot close to the surface, on average, about 6%, and the slot parallel to the bed, about 26%, are effective in reducing the maximum scour depth compared to the control pier. Also, with the increase of Froude number from 0.376 to 0.380, scour depth has increased by 11% on average.

Keywords: Scour, Bridge Pier, Near-Surface Slot, Bed-Level Slot, Flow 3D Model.

Received: 09 November 2023; Accepted: 16 April 2024

1. Introduction

Scouring is a phenomenon that occurs due to the interaction of flow conditions and the movement of sediments on the rivers and waterways due to the passage of the current. The depth caused by bed erosion compared to the original bed is called scour depth [1]. Over the past 30 years, more than 1,000 of the 600,000 bridges built in the United States have been destroyed, 90% of which were due to scour [2].

reduction methods of scouring are divided into two main categories. In the first method, the bed is reinforced and its resistance to driving forces is increased, such as the stonework of concrete protection layers, and the second method changes the flow pattern around the pier to reduce the downward flow and vortex systems that Those methods of modifying the flow say that this method can be referred to the use of Piling, collar and slot [3]. Azhari et al. (2010) simulated the scour depth around a series of three cylindrical vertical bridge foundations in a river using a three dimensional numerical model and a physical model.

---

[*] Email: m.kazem19717730@gmail.com (**Corresponding Author**)

[1] Department of water structures, Ferdowsi University of Mashhad, Mashhad, Iran.

The results showed that the maximum depth of scour occurred around the first pier and the results of the numerical model are in good agreement with the information obtained from the physical model [4]. Qaderi and Abbasi (2019) investigated the local scouring of the airfoil pier by considering the collar using a numerical model. The results showed that the use of the collar in the pier reduces the maximum scour depth in front of the bridge pier and reduces the eddies behind the pier [5]. Nazari Sharabian (2020) investigated the effect of piles with different configurations on reducing scour depth of bridge pier using folw 3D model. The results showed that the models that have a single pile installed in front of the pier and at a distance of five times the diameter of the pier are more effective in reducing scour depth [6]. Hassanzadeh et al. (2020) Using a numerical model, investigated the effect of the distance and placement of adjacent piers on the flow pattern and erosion of the sand bed around the bridge piers. By examining the changes in the distance between the piers in two cases, it was found that by reducing the distance, due to creating a wider barrier against the flow and creating a water jet between the piers, scour increases [7]. Danesh Faraz (2022) conducted the effect of cable on reducing scouring of the bridge pier using artificial neural networks and ANFIS methods. They used two types of cables with diameters of 10 and 15% of the pier diameter and twist angles of 15 and 12 degrees, respectively.

The results showed that the use of cable can reduce scouring depth by 10 and 22% respectively. And conducting tests with angles of zero, 10 and 15 degrees to the direction of the flow, showed that the pierss with an angle of 15 degrees to the flow have the greatest scour depth[8].

Many researches have been done using Flow3D numerical model on the methods of reducing scour depth around the pier of cylindrical bridge by using collar, slot in center of pier, etc. In this research, the effect of the slot around the pier body of the cylindrical bridge, in order to reduce the scour depth in two positions parallel to the bed and close to the water surface, was investigated using the Flow 3D model, which is unique from this point of view. In this research, the effect of the slot around the pier body of the cylindrical bridge, in order to reduce the scour depth in two positions parallel to the bed and close to the water surface, was investigated using the Flow 3D model, which is unique from this point of view.

2. Material and Methods

In the current research, the Flow 3D model was used to simulate the scouring of the bridge pier and the effect of the slot on it, which will be discussed further. Flow3D model is one of the most powerful models in the field of fluid dynamics. This model has the capability of three-dimensional analysis of the flow field and also has a very wide application range in problems related to fluids. And to solve the turbulent, it uses five methods of Parantel's mixing length, one-equation model, two-equation K-ε model, RNG and LES. The differential equations that must be solved in the Flow-3D model can be written in both Cartesian and cylindrical coordinates. The three-dimensional analysis of the Navier-Stokes-Reynolds equations in the Cartesian coordinate system which are solved by the finite volume method in the incompressible fluid state is in the form of relations 1 to 4.

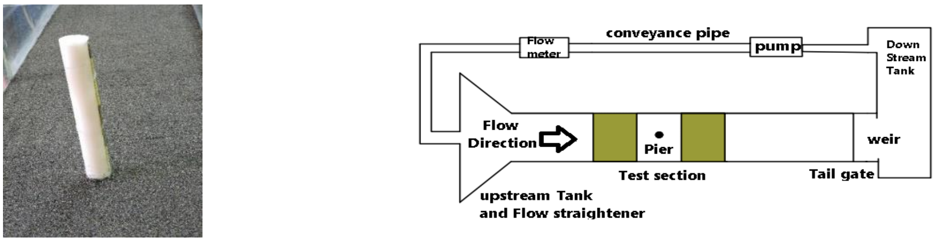
$$V_F\frac{\partial\rho}{\partial t}+\frac{\partial(\rho uA_x)}{\partial x}+\frac{\partial(\rho vA_y)}{\partial y}+\frac{\partial(\rho wA_Z)}{\partial z}=R_{SOR} \tag{1}$$

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}(uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z})=-\frac{1}{\rho}\frac{\partial P}{\partial x}+G_x+f_x \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z})=-\frac{1}{\rho}\frac{\partial P}{\partial y}+G_y+f_y \tag{3}$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z})=-\frac{1}{\rho}\frac{\partial P}{\partial z}+G_z+f_z \tag{4}$$

where V_F is the fraction of the volume associated with the flow, ρ is the fluid density, (u,v,w) are the velocity components, (Ax, Ay, Az) is the fraction of the area associated with the flow, (Gx Gy, Gz) is the mass acceleration, R_{SOR} The Source term, (Fx, Fy, Fz) is the viscous acceleration in the (x, y, z) directions and p is the pressure. In this research, the laboratory results of Kazemian et al. (1400) have been used [9]. The laboratory results obtained for the control pier were carried out in a laboratory flume with a length of 10.5 meters, width and height of 0.5 meters. The flume was made of glass and polyglass and its slope was 0.001. The location of the pier was a Teflon plastic plate with a height of 0.02, a width of 0.5 and a length of 1.06 meters, and two platforms with a height of 0.12, a width of 0.5 and a length of one meter were located upstream and downstream of it. Figure 1 shows a picture of the general design of the channel and the control pier used in the laboratory.



(A) **(B)**

Figure 1. control pier (A) and Flume schematic (B)

The pier used in the experiment has a diameter of 4 cm. Relative velocity is defined as the ratio of flow velocity to critical velocity. In this experiment, two relative speeds of 0.870 and 0.92 were used.

Bed sediments were used in a uniform and non-cohesive form with an average diameter of 1 mm and a geometric standard deviation of 1.4. And also the sediment beds were 15 cm high.

In this research, AutoCAD software was used to draw the geometry of the channel and pier. To save the calculation time, it was simulated in about 1.5 meters of the length of the laboratory channel. Figure 2 shows the geometry of the created channel, bed and pier.

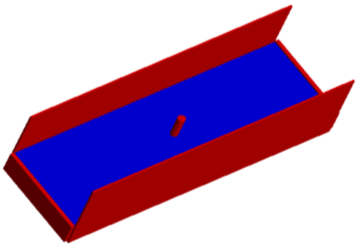
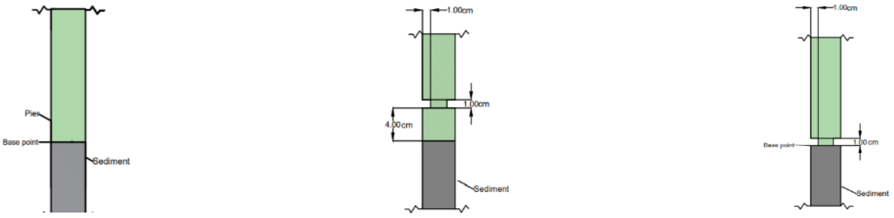


Figure 2. Geometry created by AutoCAD software

The piers used in the numerical model include three types of piers. The S0 pier without slot as a control pier has the same characteristics as the laboratory sample in order to validate the results. The height and indentation of the slot was introduced as a measure of the diameter of the pier and it was considered as a quarter of the diameter of the pier. The pier S1 has a slot with a height of one centimeter all around the pier and the position is 4 cm away from the bed. The S2 base has a slot with the same specifications as the S1 pier, and is positioned level with the bed. The indentation depth of the slot in both types of sloted pier is one centimeter. In such a way that the diameter of the base at the place of the slot is 3 cm with a decrease of one centimeter. The specifications of the piers in Figure 3 are shown.



(A) (B) (C)

Figure 3. control pier (A), pier with a slot close to the surface (B) and pier with a slot parallel to the bed (C)

The information related to sediments, fluid and other effective parameters were entered into the software.

Also, flow rate and flow depth were introduced to the software as initial conditions. The geometry of the model was entered into the software, and for more accuracy in obtaining scour depth and showing the characteristics of the pier, a another block was considered around the bridge pier. The boundary conditions defined in this study are shown in Figure 4, where the inlet boundary condition (Xmin) was considered as a constant discharge (Q). In the laboratory model, at the inlet boundary, water flows with a known depth and discharge, and for this reason, the inlet condition of a constant discharge with a constant water height was used. For the outlet of the channel (Xmax), the boundary condition was considered in such a way that the fluid is in contact with the air. As mentioned, to reduce the calculation time, 1.5 m of the length of the laboratory channel was simulated in such a way that the pier is placed in the middle of the channel. The boundary of meshing on the bed of the channel is one centimeter above the bed (within the sediments), and in the width of the channel one centimeter on both sides of the channel wall (according to the distance of one centimeter from the width of the channel on both sides, the boundarys of the meshing are inside the water and instead of 50 cm 48 cm was considered.) was placed. These boundaries were defined as symmetry (the conditions outside the solution network are considered exactly the same as the conditions on the internal boundary of the domain). This type of boundary condition is used in symmetric simulations. The free surface of the fluid was also considered symmetrical.

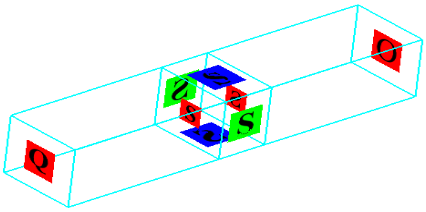


Figure 4. created boundary conditions

RNG model of turbulence has been used to model turbulence in this research. Compared to the k- Ɛ turbulence model, this model needs fewer experimental constants and has shown better performance for simulating regions with flow separation [10]. In addition, three models k-Ɛ, RNG and LES were investigated to obtain the maximum scour depth under the same conditions, and the results showed that the RNG model has a better match with the experimental observations.

3. Results and Discussion

One of the most important parts of numerical solution is determining the size of computing domains or the size of cells in meshing. The number of cells in a mesh depends on the determined range and size and affects the accuracy and time of calculations. As a result, it is necessary to adopt a suitable meshing that both meets the accuracy required in the calculations and is within the conventional limits in terms of the calculation time. Cartesian meshing is used in this numerical simulation. Appropriate meshing was investigated in two modes of uniform and non-uniform meshing, and then according to the geometry of the model and also the sensitivity to observe as well as possible the maximum depth of scour and its location, non-uniform meshing was done around the pier of the bridge. So that the pier and its surroundings have a finer mesh.

To determine the optimal number of meshes, the meshes started from 50,000 and increased with steps of 50,000. The results showed that for meshes less than 400,000, the rigid boundaris of the channel as well as the slot in the pier could not be identified completely by the model. But gradually, with the increase in the number of cells, the detection ability increased. For example, the identified solid boundaries for the channel, bridge pier and bed with the number of 200 thousand meshes and 600 thousand meshes are presented in Figure 5. Research showed that for a mesh number of more than 500,000, the software can fully recognize all parts of the channel. For this reason, for meshes more than 500 thousand, the compatibility of the model results with the laboratory results was considered. For 500, 550, 600, 650, 700 and 750 thousand meshes, the relative error between the modeling results and the laboratory results related to the control pier for the relative flow speed of 0.92 is presented in Table 1.

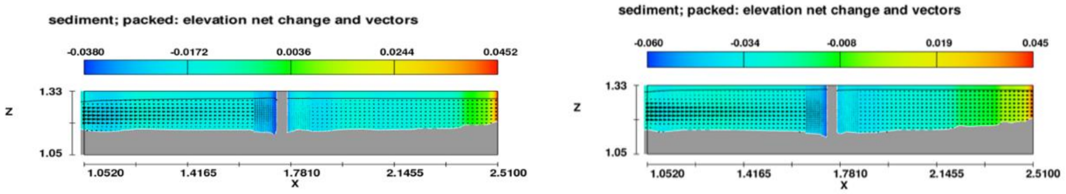
Table 1. The maximum scour depth in the exoerimental model and the numerical model with a relative speed of 0.92

The number of meshes	Maximum scouring depth in the experimental model, cm	Maximum scouring depth in the numerical model, cm	Relative error, %
500000	6.2	3.2	48
550000	6.2	3.3	46
600000	6.2	4.6	25
650000	6.2	6	3
700000	6.2	6	3
750000	6.2	6	3


(A) (B)

Figure 5. Shows the mesh created in the software for 200 thousand meshes (A) and 600 thousand mesh (B)

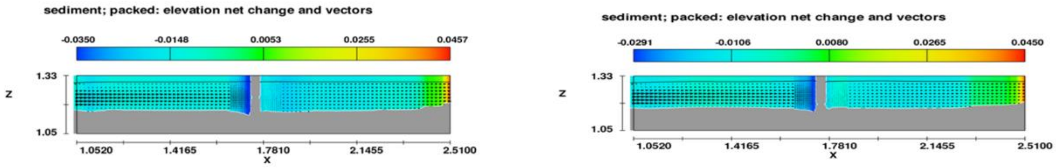
For the number of mesh equal to 650 thousand, the relative error is equal to 3% and with the increase of the number of mesh to 750 thousand, there was no significant change in the maximum scour depth. Therefore, to reduce the calculations, the number of meshes equal to 650 thousand was chosen as the optimal number of meshes, and then all the numerical simulations were performed with 650 thousand meshes. It should be noted that the data of the maximum scour depth in the above table for the numerical model are in the test time of 800 seconds. The duration of the test should be chosen in such a way that the maximum scour depth can be reached and balance can be created in the scour hole and at the around pier. For this purpose, the test time was done starting from 400 seconds and adding a step of 200 seconds for different meshing. The results showed that for a time of 800 seconds and an optimal mesh of 650 thousand, the relative error is about three percent. Also, by increasing the test time to 1000 seconds, no effect was observed on the test results. Therefore, this time was considered as the test time. The purpose of this research is the effect of slots in reducing the scour depth of the bridge pier and the comparison of scour in the sloted pier and control. Therefore, considering that 60% of the scouring depth occurred in 400 seconds, this time was considered as the test time to save calculation time. Figure 6 shows the maximum scour depth with the optimal mesh of 650,000 at times of 400 seconds and 800 seconds.



(A) (B)

Figure 6. The maximum scouring depth after 400 seconds (A) and 800 seconds (B)

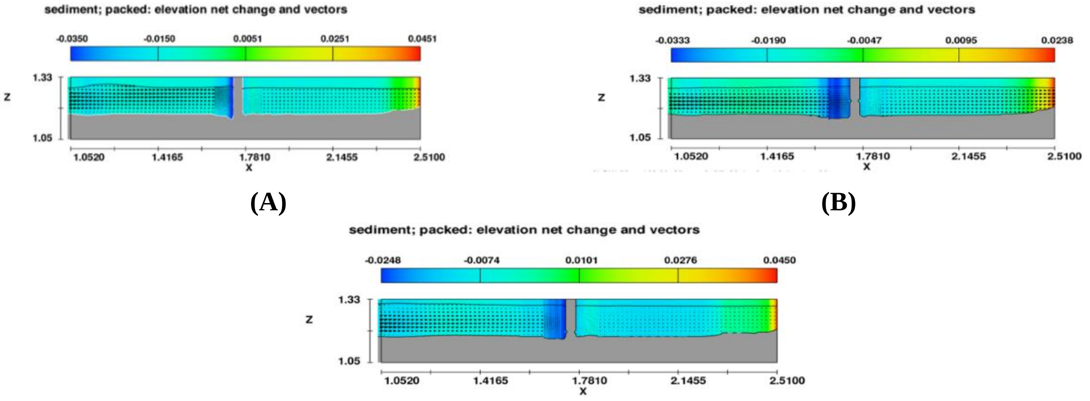
The comparison of the maximum scour depth at the relative speed of 0.92 in piers S_1 and S_2 in 400 seconds is shown in Figure 7. According to the results obtained at the relative speed of 0.92, the pier with the slot close to the surface (S_1) is about 8% and the pier with the slot parallel to the bed (S_2) is about 24% compared to the control pier had effective in reducing the maximum scour depth.



(A) (B)

Figure 7. The maximum scouring depth in pier S1 (A) and pier S2 (B) at relative speed of 0.92

The comparison of the maximum scour depth in foundations S_0, S_1 and S_2 with a relative speed of 0.87 is shown in Figure 8. According to the results obtained at a relative speed of 0.87 the slot close to the surface has an effect of 4.8% and the slot parallel to the bed has an effect of 29% compared to the control pier in reducing scour depth. Also, with the increase in flow rate and as a result the relative speed increase, scouring depth has increased in different pier models.



(A) (B)

(C)

Figure 8. The maximum scouring depth with a relative speed of 0.870 in the control pier (A), the pier with a slot close to the surface (B) and the pier with a slot close to the bed (C)

Froude number in the rectangular channel is defined as the relation 5.

$$Fr = \frac{V}{\sqrt{gy}} \tag{5}$$

where V is the speed of the flow, g is the acceleration of gravity and y is the depth of the flow. Figure 9 shows the effects of increasing the Froude number on the maximum scour depth in the control pier (S_0), the pier with a slot close to the surface (S_1) and the slot parallel to the bed (S2). The results showed that with the increase of the Froude number from 0.376 to 0.380, the maximum scour depth in piers S_0, S_1 and S_2 has increased by about 8, 6 and 20 percent, respectively.

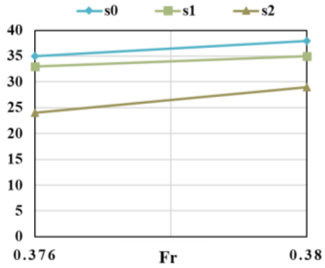
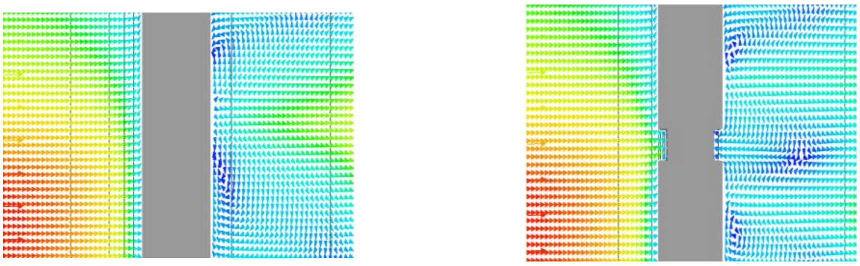


Figure 9. The effects of increasing Froude number on scour depth in different piers

Figure 10 shows the velocity vectors in the control pier and the pier with a slot close to the surface. According to the figure, the results showed that the use of slots around the bridge pier weakens the eddies around the bridge pier as the main factor of scouring phenomenon and also reduces the flow speed and Froud number in that area.



(A) (B)

Figure 10. Velocity vectors around the control pier (A) and the pier with a slot close to the surface (B)

4. Conclusion

In general, it can be said that with the increase in the flow intensity and consequently the Froude number and flow speed, the scour depth will increase. With the increase of relative speed from 0.87 to 0.92, scouring depth has increased by 11% on average. The results showed that in different relative velocities, on average, the slots around the pier in the position close to the water surface (S1) are 6% and in the position parallel to the bed (S2) are effective in reducing the maximum depth of scouring by 26%. In general, the use of slots around the bridge pier weakens the eddies around the bridge pier and the speed in that area and reduces scour depth.

References

1. Shafaei bajestan, M. (2013). Hydraulic of sediment transport. 4nd ed., Shahid Chamran University of Ahvaz Publications. (in Persian)
2. Sumer, B.M. and Fredsoe, J. (2002). *The mechanics of scour in the marine environment*. Advanced series on ocean engineering,vol 17.World Scientific.
3. Mashahir, M. B and Zarrati, A. R. (2002). Effect of Collar on Time Development of Scouring around *Rectangular Bridge Piers*. International Journal of Engineering, Transactions B: Applications 25(1©):11-17.
4. Azhari, A. and Saghravani, S.F. and Mohammadnezhad, B.A., (2010), "*3D Numerical modelling of local scour around the cylindrical bridge piers*," XVIII International Conference on Water Resources CMWR 2010 J. Carrera (Ed) CIMNE, Barcelona.
5. Ghaderi, A., & Abbasi, S. (2019). *CFD simulation of local scouring around airfoil-shaped bridge piers with and without collar*. *Sādhanā*, *44*, 1-12.
6. Nazari-Sharabian, M., Nazari-Sharabian, A., Karakouzian, M., & Karami, M. (2020). *Sacrificial piles as scour countermeasures in river bridges a numerical study using flow-3D*. Civil Engineering Journal, 6(6), 1091.
7. Hasanzadeh, Y. Kardan, N. and Tamizi, A., (2020). *Numerical study of the effect of the distance and the placement of the adjacent piers on the flow pattern and erosion of the sand bed around the bridge piers.* Iran Water and Soil Research (Agricultural Sciences of Iran), 51(7), 1753-1737. SID. (in Persian)
8. Daneshfaraz, R., Abam, M., Heidarpour, M., Abbasi, S., Seifollahi, M., & Abraham, J. (2022). *The impact of cables on local scouring of bridge piers using experimental study and ANN, ANFIS algorithms*. *Water Supply*, *22*(1), 1075-1093.
9. kazemian, M., Gohari, S., & shahverdi, K. (2021). *The Effect of Height and Vertical Position of Slot on the Reduction of Scour Depth around Cylindrical Bridge Pier in Uniform and Non-uniform Sediments*. *Journal of Hydraulics*, *16*(4), 37-48. doi: 10.30482/jhyd.2021.285672.1527. (in Persian)
10. Sicilian JM, Hirt CW and Harper RP, (1987). FLOW-3D: *Computational modeling power for scientists and engineers*. Report FSI-87-00-1, Flow Science, Los Alamos, NM.