*Journal of Ferdowsi Civil Engineering, Vol.33, No.2, 2020.*

# Three Dimensional Simulation of Vortex Shedding Phenomenon and Sediment Transport around Obstacles Subjected to Flow using Flow3D

Nazila Kardan[1] Habib Hakimzadeh[2]

Yousef Hassanzadeh[3]

## 1. Introduction

Scour or the removal of material from the bed and banks of streams and near the piers and abutments, always occurs due to flow acceleration, turbulence and erosive action of flowing water. This process is affected by a large number of variables, mainly, the flow, fluid, pier, and sediment characteristics. It can be divided into three main forms of general scour, contraction scour, and local scour. However, general scour (or evolution of the waterway) occurs naturally in river channels by aggradation and degradation of the river bed.

A change in the river hydraulic parameters is known as the major cause for this type of scouring. Contraction scour occurs by reduction of channel's cross-sectional area at the location of water structures (like bridge piers and abutments) that increases the flow velocity and bed shear stresses, and the transport of the bed sediments. Local scour is the third type of scouring that occurs in the vicinity of bridge piers or abutments. In such a scouring, downward flow is induced at the upstream end of the pier and leads to a localized erosion around the pier. It depends on the balance between streambed erosion and sediment deposition.

Based on this fact and the mode of sediment transport in the approaching flow, two types of local scour can be distinguished: clear water scour and live bed scour. In clear water scour, no sediments are delivered by the river approaching flow, while an interaction exists between sediment transport and the live bed scour process. Local type of scour has long been acknowledged as the major cause of the bridge failures around the world, causing major human and financial losses.

Monitoring local scour is a way to avoid major damages that may occur and to ensure the continued safe operation of the structure. Most of traditional monitoring techniques are based on installation of expensive underwater devices that may be damaged during a flooding event, when the highest risk of scouring exists. In such circumstances, computerized simulation of the scour occurrence can be introduced as a low cost/time consuming alternative for the prediction of the probable failures. The application of simulation software is in many ways similar to the set-up of an experiment, in which Computational Fluid Dynamics (CFD) methods are always used for the simulation of flow process by discretization and solving of Navier-Stokes and continuity equations for the computational cells. Flow-3D is CFD software that employs numerical techniques to solve the fluid motion equations to obtain transient, three dimensional solutions to multi-scale, and multi-physics flow problems. An array of physical and numerical options allows users to apply Flow-3D to a wide variety of fluid flows and heat transfer phenomena. This software is used widely for solving different hydraulic problems.

The results showed that the maximum shear stress occurred when the foundation level was at bed level and the maximum shear stress exerted on the bed decreased by factors of 17% and 53% in the cases of foundation level to be below and above bed levels, respectively. In addition, the amount of vortex flows increased in upstream piers group and near bed in the case of setting the foundation above the bed. This is because of the fact that the volume of piers group acting as an obstacle against flow was more than other level settings. In a study, experimental results were compared with numerical simulation of bed shear stress around bridge abutment in a compound channel by Flow-3D model. In this study, scour simulation around different pier models will be further discussed. The main purpose of this study is to evaluate the effect of two key factors causing local scouring, flow separation, and downward flow in upstream of the pier, on maximum value scouring around 10 defined pier models. The countermeasures that considered in this regard include berm, cross-section, and middle slot.

## 2. Materials and Methods

Flow-3D uses the Volume of Fluid (VOF) method to track fluid-fluid or fluid-sediment interfaces for solving the nonlinear Navier-Stokes equations in a three dimensional space. Also, Fractional Area/Volume Obstacle Representation (FAVOR) method is used to illustrate the complex boundaries of the solution domain. Flow-3D also allows for several turbulence closure schemes to be incorporated and tested. These closure schemes include simple eddy viscosity, one-dimensional Prandtl mixing length, two-equation k-e, large-eddy, and four-equation ReNormalized Group (RNG) models. The 3D-CFD model implemented in Flow-3D represents a conventional channel with 0.456 m-wide rectangular inlet with the length of 6.0 m. A vertical cylindrical pier with the diameter of 0.051 m and height of 0.5 m was inserted in the center of the channel as a solid standard component. A packed sediment type component was used for the channel bottom with dimensions of 2.0, 0.456, and 0.2 m for length, width and height, respectively. Particle size diameter was 0.358 mm with a

[1] Corresponding Author: Assistant Professor, Department of Civil Engineering, Azarbaijan Shahid Madani University, Tabriz, Iran.
Email: n.kardan@azaruniv.ac.ir

[2] Professor, Faculty of Civil Engineering, Sahand University of Technology, Tabriz, Iran.

[3] Professor, Department of Water Engineering, Center of Excellence in Hydroinformatics, Faculty of Civil Engineering, University of Tabriz, Tabriz, Iran.

density of 2680 kg/m$^3$ with critical shields number of 0.031 and drag coefficient of 0.5. A mesh block of 360000 cells was fitted to the model geometry.

To increase the model accuracy around the pier, two mesh planes with finer resolutions were defined for both sides of the pier in x and y directions, separately. Also, two solid components were settled at the outer sides of the geometry (both for beginning and end parts of the channel) to prepare an inflow bottom at the top edge of the sediment (elevation of 0.2 m) in order to prevent against upward movement of sediments in the beginning of simulations. Boundary conditions were volume flow rate at the inlet, outflow at the outlet, wall on the bottom, right and left sides, and symmetry on the top of the block.

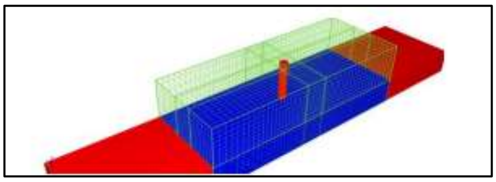

**Figure 1. Meshing structure of the model for simulation of scouring**

## 3. Results and Discussion

The proposed models are classified into four general groups. The first group (A) consist of different geometric sections (circular, round-nosed rectangular, elliptical, and lenticular), to determine the effect of flow separation and lee-wake vortices. For the second group (B series), four different levels of berms in circular geometry (i.e., 5, 6, 7, and 8 cm from the bed level) have been considered in the circular cylindrical pier to evaluate the appropriate level of the berm in controlling the down-flow at upstream of the pier. In the related literatures, several values have been suggested for the pier berm height. In the third group (C series), a new combination of the two most effective countermeasures (i.e., berm and geometry) was used. In the fourth group, combination of berm, geometry, and centered slot is considered (D series). The slot with a length of two times the berm diameter (i.e., 12 cm) is located near bed and extended into the depth of the sand recess (i.e., 5 cm above the bed and 7 cm under the bed).

The clear-water maximum scour depth at the equilibrium state for a single unprotected circular pier under the steady and uniform flow condition is located in front of the pier. In the presence of a countermeasure, the maximum scour depth occurs at the pier front. Therefore, the efficiency of the countermeasure ($r_{de}$) is defined in terms of the reduction of the equilibrium maximum scour depth observed in proximity to the pier as:

$$r_{de}(\%) = \frac{d_{s0}-d_s}{d_{s0}} \times 100 \tag{1}$$

where $d_{s0}$ and $d_s$ are the equilibrium maximum scour depth in front of the unprotected and protected piers, respectively.

The lenticular-shaped pier had the most effect on flow pattern, therefore, minimum scouring depth achieved by this model. This pier was streamlined and no flow separation was observed at its sides. The elliptical-shaped model considered as a streamlined form pier and had a relatively similar scouring process to the lenticular model. In Group B, the results showed that although the gibbosity of the berm prevented the down-flow to hit the bed, the intensified flow separation due to the increased diameter affects the efficiency of the berm. An appropriate value for the height of the berm may be considered quarter to third of water depth.

## 4. Conclusion

According to the results obtained in this study, it was concluded that Flow-3D is a capable model for simulation of different flow attributes (velocity and scouring depth) with/without the presence of an obstacle (like bridge pier) at different dimensions of 1D, 2D and 3D. But it has some constraints for simulation of sedimentation/scouring processes.

Numerical simulation of scouring around 10 combined model piers in clear-water condition was conducted. Identifying the effect of new proposed countermeasures on local scouring variation around non-uniform bridge piers with special attention to the two effective parameters of scouring, flow separation and down-flow, is considered. The model piers were categorized into four groups and the achieved results can be summarized as follows:

1. In A series models, the lenticular cross-section has the minimum scouring depth.
2. For B series models, an appropriate value for the berm height may be considered quarter to third times of the water depth. So, the results of A and B series reveal that the scour depth is more affected by the pier cross-section rather than the berm.
3. Combination of the berm and different cross-sections shows that adding a berm to models having various cross-sections may not be an effective tool against scouring.
4. A combination of the berm, cross-section, and a centered slot shows that intensification of bed erosion around the edges of the slot can be regarded as the main disadvantage of the slot when applied to the circular pier.