Numerical modeling of the effect of pile geometry and pile cap leveling on the local scour under inclined pier group

Ramtin Sobhkhiz¹ · Alireza Mardookhpour²

Received: 28 February 2019 / Accepted: 1 November 2019 / Published online: 28 November 2019
© Springer Nature Switzerland AG 2019

Abstract

When the bridge piers are located in front of the water flow, vortices are formed against it and due to their activity, the materials of the river bed are eroded around the bridge piers and the scouring hole is created. If the foundation depth and bridge pier piles are insufficient, the bridge will fail. Now, due to the importance of identifying the mechanisms affecting the scouring on this types of bridge piers, the effect of the geometric shape of the piles that installed under the inclined piers and also the effect of the pile cap leveling in the sedimentary bed have been investigated and the properties of the scouring around the inclined pier group was studied numerically and using the Flow-3D software. The study of total shear stress in the flow bed at different leveling of the pile caps shows that of the highest shear stress was created when the pile cap position is at the same level with the river bed and by installing the pile cap at a lower level than the river bed, the maximum shear stress decreases. This may be due to the fact that in this case the distance between the pier group increases and the presence of the second pier decreases the flow rate in the pier group and different pier in the one pier group act as the two Independent piers in the formation of flow pattern. By comparing the final longitudinal sections of the scouring at different leveling of the pile cap, it is concluded that the largest reduction in scouring depth occurs in aerofoil-shaped pile caps and pile caps with the sharper nose and better aerodynamic shapes are good options to control the horseshoe vortices and will reduce the scouring depth around the inclined pier group.

Keywords Scour · Flow pattern · Crop processing digits · Pile geometry shape · Inclined pier group · Flow-3D

1 Introduction

Scouring is a natural process that is created by the erosion caused by the flow of water on the river bed and also formed near the bridges' piers and supports [1]. Scouring around the head of the piles and the pier of the bridges is one of the major problems of structures such as bridges whose piles are in the water of rivers with erosive flows. The erosion around the bridge piers causes the complete or local failure of some of the bridges. Once a river is in equilibrium state, the amount of input and output sediment is equal to each other. Naturally, a river, in order to achieve such an equilibrium state, shows the behaviors

that ultimately leads to reach a physical equilibrium. Of course, annual floods are one of the most important factors in the destruction of this system and cause annual erosion in the river path.

Scouring occurs when the shear stress between the water passing through the river bed and the wall of the stream is increased from the amount of stress necessary to move the particles forming the river cross section [1]. Research indicates that the local scouring phenomenon around the piles of bridges is the main cause of the failure of the piles. In fact, the flow pattern affects the processes of erosion and sedimentation around the pile. Therefore, one of the important points to be taken into account in

✉ Alireza Mardookhpour, alireza.mardookhpour@liau.ac.ir | ¹Department of Civil Engineering, Qom University, Qom, Iran. ²Department of Civil Engineering, Lahijan Branch, Islamic Azad University, Lahijan, Iran.

hydraulic designing of the bridges is the design of the pile and foundation geometry with regard to scouring. In terms of economic and geotechnical reasons, the inclined pier group is used in design of the piers of the bridges [1]. The first fundamental studies on scouring were made by Engels at the beginning of the twentieth century and at the Zooner Laboratory in Germany (rouz-Inn) [1]. After this research, many studies were conducted by Chabert and Engeldinger at the Chago Lab. The results of this research show that the amount of scouring in different types of transparent waters and flood with live bed.

Hannah [2], conducted studies on the effect of distance between piers on scouring. These studies showed that as the distance between the bridge piers increases, the effect of the reinforcing factor decreases, and when the spacing between the piers is 2.5 times the pier diameter, the reinforcement factor reaches the maximum value.

Melville and Sutherland [3], studied the collision of the water flow to the bridge pier that leads to separation of the water flow, which is the main cause of local scouring in the bridge piers. The results of these studies showed that collision of water to the pier, reduced the water pressure to less than the free flow pressure to the bed, which in turn, produced downward flow. In other words, the direction of the vortex system will be upward. Research indicates that horseshoe vortices and lifting phenomenon have a great influence on the formation of a scouring hole around the bridge's piers.

Also, due to the importance of studying the scouring phenomenon, Melville and Sutherland [3], studied the parameters affecting scouring around the bridge piers. They investigated the effect of geometric parameters such as pier diameter and uniformity of its cross-section, and it was determined that the maximum scouring depth at the pier of the bridge having a uniform cross-section is equal to 2.4 times the pier diameter, If the cross-section of the bridge pier is variable (similar to when the pier is on the foundation, different levels of scouring occur depending on the base level of the bridge foundation. Also, the researchers found that scouring of two piers could fall on each other, and the level of the bed in the rear zone of the front pier was lowered. In this situation, the water flow, moves the particles simply from the scouring hole of the front pier and a large depth scouring has been created in the front of the bridge pier.

Parola et al. [4] investigated the scouring and showed that the maximum scouring depth was reduced when the bridge foundation was at a lower depth in comparison with the maximum scouring depth of a pier. This is due to the fact that the scouring amount will be reduced at

greater depths and scouring hole cannot be replaced with the new hole.

Melville and Radwicky [5], stated that in the real conditions, bridge piers were designed and constructed with different geometric shapes, and the cross-section of many piers have been changed with increasing the height.

In addition, Melville and Chiv [6], studied the effects of hydraulic flow conditions such as velocity and depth of flow on scouring. Their research showed that the effect of the flow depth on the maximum value of scouring is depend to the diameter of the aggregate particles in the river bed. This effect varies from relative depth of 1 for fine grain particles to a relative depth of 6 for coarse grain particles (relative depth is equivalent to the depth of water flow relative to the pier diameter).

Ataie-Ashtiani et al. [7], investigated the methods that presented by HEC-18 and Kollman, which were used to estimate the depth of the scouring around the bridge piles, and presented corrective equations for scouring depth estimation.

Amini et al. [8], also investigated the effect of different distances between the pile and the foundation level on the scour depth, and provided a relationship for assessing the depth of scouring under different conditions.

Esmaili varaki et al. [9], conducted studies in this field, showed that the inclining of the pier to the upstream direction relative to the parallel plane with the water flow would increase the maximum amount of scouring.

Wen and King [10], conducted a numerically calculations and stated that CFD techniques, with the help of powerful water flow image processing method, are capable of displaying flow during scouring, which helps to detect the scouring mechanism and increase scouring over time. By performing these experiments, it was concluded that one of the important parameters for designing.

the bridge pier is the maximum depth of the scour hole, but the position where the maximum depth of the hole is located and the shape of the hole is different from the experimental results. Hydraulic and sedimentary conditions have been discussed in discussions related to rivers and pile and pier of bridges. As a result, it has always been necessary to investigate the scouring phenomenon, and the hydraulic and sediment properties of water flow that passes from piers with different shapes are considered, but so far little research has been done in this context. In this paper, the numerical simulation analysis of the effect of different geometries of piles with fixed foundation and inclined pier group on local scouring is performed.

2 Materials and methods

The first step for simulating the flow is to calculate the equations associated with the scouring phenomenon. Solving flow problems including the continuity equation and the three momentum equations in the direction of three axes of coordinates (Navier–Stokes equations) are the basis of all numerical methods of modeling. In this section of the study, a general description of these equations will be presented. The continuity equation is one of the main equations of fluid mechanics and another form of the conservation of mass law, indicating the mass change of a fluid that includes the control volume over a specific time period. This mass change is equivalent to the difference between the total input and output masses to this control volume [11]. If the selected control volume is in accordance with Fig. 1, the continuity equation is as follow:

$$\dot{m}_{\text{stored}} = \sum \dot{m}_{\text{in}} - \sum \dot{m}_{\text{out}} \quad (1)$$

In Eq. (1), parameter $\dot{m}_{\text{stored}}$ indicates the mass stored in the control volume, parameter $\sum \dot{m}_{\text{in}}$ represents the total sum of the input masses to the control volume surfaces and parameter $\sum \dot{m}_{\text{out}}$ represents the total output mass from the surfaces of the control volume. If Eq. 1 is written in an integral form on the control volume surfaces, Eq. 2 will be obtained:

$$\frac{\partial (\int_V \rho dV)}{\partial t} = - \int_A \rho (\vec{V}_{\text{rel}} \cdot d\vec{A}) \quad (2)$$

In Eq. (2), the relative velocity ($\vec{V}_{\text{rel}}$) indicates the difference between the velocity of the fluid velocity and the velocity vector of the control volume, that is, if the control volume is in motion and its velocity is equivalent to the fluid velocity, the input and output mass is zero. To change the surface integral (Eq. 2) into the volume integral, the divergence theorem should be used. According to this theory, it is possible to change the surface integral of the $\vec{X}$ vector quantity to the volume integral by using Eq. 3:

$$\int_A \vec{X} \cdot d\vec{A} = \int_V \text{div}(\vec{X}) dV \quad (3)$$

If it is assumed that in the $\vec{X}$ -value vector, x is equal to the product of a scalar quantity (such as density) and a vector quantity (such as velocity), that is, the relation $\vec{X} = \rho \vec{V}$, by inserting $\vec{X}$ in Eq. (3) and inserting the obtained integral in Eq. (2), Eq. (4) has been resulted as follow:

$$\frac{\partial}{\partial t} \left(\int_V \rho dV \right) + \int_V \text{div}(\rho \vec{V}_{\text{rel}}) dV = 0 \quad (4)$$

To use the Divergence Theorem, it must be assumed that the density and velocity functions are completely continuous functions. For conditions where changes in the shape of control volume are very little over time and the shape of the control volume is constant, Eq. (5) exists:

$$\frac{\partial}{\partial t} \left(\int_V \rho dV \right) = \int_V \frac{\partial \rho}{\partial t} dV \quad (5)$$

And Eq. (6) can be written as follows:

$$\int_V \left[\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{V}_{\text{rel}}) \right] dV = 0 \quad (6)$$

If the dimensions of the control volume are assumed to be close to zero and the location variations of the quantities within it are not significant, then, the following equation is obtained:

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{V}_{\text{rel}}) = 0 \quad (7)$$

In the case where the control volume has a constant dimension, its relative velocity will be similar to the fluid velocity and the continuity equation will be written as follows:

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{V}) = 0 \quad (8)$$

Fig. 1 Selected control volume and its surface boundaries

If this vector equation is generalized, the following equation is obtained:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (9)$$

$$\left(\frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + v \frac{\partial \rho}{\partial y} + w \frac{\partial \rho}{\partial z} \right) + \rho \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) = 0 \quad (10)$$

Equation 10 can be simplified as follows:

$$\frac{D\rho}{Dt} + \rho \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) = 0 \quad (11)$$

Finally, the incompressible flow continuity equation is written as follows:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (12)$$

Fluid motion related equations are solved by finite volume approximation using Flow3D software. In this software, the flow environment is divided into a network with fixed-size rectangular cells. In each cell, there are average values of dependent quantities. All variables are considered at the central point of the cell and their values have been obtained, and only the velocity variable is calculated at the center of each side surface of the cell [12].

Favor method is used to solve the problem, in which curved barriers, boundary walls and other geometric shapes, by specifying the fraction of the area of their faces and considering the open volume fraction to the volume flow of the cell, in the network are considered. If a grid that is used in the volume method is a non-uniform grid, the order of accuracy for a numerical solution should have the appropriate order and the accuracy of the second order is the best. In all states, at least the accuracy that can be used for boundary conditions is the order accuracy of one. For example, if a part of a cell is blocked by an obstacle, linear interpolation is used for boundary conditions. In this software, for modeling the free surface of the water, the VOF model is used in which the element volume of fluid F is obtained by using equation [13] (i.e., the displacement equation for the F fluid volume element) [12]:

$$\frac{\partial F}{\partial t} + \frac{1}{V_f} \left(\frac{\partial}{\partial x}(FUA_x) + \frac{\partial}{\partial y}(FVA_y) + \frac{\partial}{\partial z}(FWA_z) \right) = 0. \quad (13)$$

The parameter F in the above equation is a coefficient that always has a value between zero and one and its value depends on the amount of fluid in the control volume. If $F=1$, it means that the control volume (the cell on which the calculations are performed) is full of fluid and if $F=0$,

it means that there is no water in the cell and the control volume filled with air [12].

2.1 Erosion model for modelling the scouring phenomenon in bridge pier

The erosion model is used to simulate the transfer, erosion, sedimentation and displacement of sediments by fluid (can be defined for all materials with different physical properties). In this model, two concentration fields are used, one associated with suspended load, and another with the bed load. The displacement of suspended sediments occurs as a result of the localized pressure gradient change. The flow of input particles which contain suspended particles and particles that resulted from erosion of the river bed causes the formation of suspended sediments. The adjacent particles block the sediments in the river bed, that leads to fix these sediments and do not move simply, and they can be moved only if they become eroded at the fluid-bed interface and become suspended load. Also, the suspended load is converted to bed load if the deposition rate of the sediment is higher than the erosion rate of the bed. The part of the control volume in which solid particles are filled in it, represented by the symbol f_s and the part filled with fluid is represented by the symbol f_L and the following Equation is written for it [13]:

$$f_s + f_L = 1 \quad (14)$$

The suspended load increases the actual fluid viscosity, and this increase occurs as long as the volume fraction of the solid particle (f_s), reaches to the cohesive volume fraction (f_{ace}). From this step onwards, the increase in suspended load does not increase the viscosity of the fluid, but also triggers the solid behavior of the particles. Now, the fluid average viscosity is obtained from the following equation: [13]

$$\mu^* = \mu_f \left[1 - \frac{\min(f_s, f_{SCO})}{f_{SCR}} \right]^{1.5} \quad (15)$$

In Eq. 15, μ_f is the molecular viscosity of the fluid, and f_{SCR} is the critical solid fraction with a default value of 0.67. The parameter μ^* is the increased amount of viscosity due to the presence of suspended load in the fluid and the maximum value of which is obtained in the critical solid fraction and is equal to $18 \mu_f$. If the volume fraction of the sediment is greater than or equal to f_{SCR} , the density of the dense sediment is equal to the $f_{SCR} \times \rho_s$ product. This relationship is one of the initial conditions of the model. The apparent density represented by the parameter $\bar{\rho}$ is represented as a linear function of the sediment volume [13].

$$\bar{\rho} = \rho_L + f_s(\rho_s - \rho_L) \quad (16)$$

The parameters ρ_s and ρ_f are the apparent density of the sediment particles and the apparent fluid density, respectively. The drift phenomenon is defined as the deposition of sediment particles due to the inducing of buoyancy forces on them. In the simulated scouring model in Flow-3D software, sediment particles are considered as spherical shapes and are affected by fluid viscosity. Therefore, the sedimentation coefficient D_f is obtained from the following equation: [13]

$$D_f = \frac{d_{50}^2 \times (\rho_s - \rho_L)}{18\mu} \quad (17)$$

Therefore, the sedimentation rate is calculated by the following equation:

$$u_{drif} = D_f \times f_L \frac{\nabla P}{\bar{\rho}} = \frac{f_L \times d_{50}^2}{18\mu} \frac{\nabla P}{\bar{\rho}} (\rho_s - \rho_L) \quad (18)$$

In Eq. (18), the parameter d_{50} is equal to the average diameter of the sediment particles, the parameter μ equals the fluid (water) viscosity, the parameter $\nabla P/\bar{\rho}$ is equivalent to the mechanical potential gradient or the body acceleration or sediment particle acceleration. The value of $\nabla P/\bar{\rho}$ can increase up to ten times the acceleration of gravity that leads to eliminate the numerical effect of the pressure fluctuation. By closing to the free surface of the fluid, the particle body acceleration is replaced by gravity acceleration (g). Sedimentation is a process that occurs only in the presence of solid particles, and if the control volume is completely filled with sediment, the coefficient f_L will be zero and $u_{drif}=0$ relationship will be formed. Also, in the Flow-3D model, the sedimentation coefficient required to calculate the buoyancy force is computed from the following equation: [12]

$$D_f = \begin{cases} \text{Equation (17) and (18)} \\ \left[\frac{f_s - f_{SCR}}{f_{SCO} - f_{SCR}} \right]^{-2} \left[1 - \frac{f_s - f_{SCR}}{f_{SCO} - f_{SCR}} \right]^3 \\ 0 \end{cases} \quad (19)$$

At the surface of sediments in the river bed, there is a shear stress that causes erosion and movement of sediment particles in the bed. This erosion is a function of fluid shear stress at the bed surface, critical shear stress, and the density of the fluid and sediment. The minimum shear stress required to lift the sediment particles from the fluid-active bed surface using the critical shields parameter is shown in the following equation: [12]

$$\theta_{cr} = \frac{\tau_{cr}}{g(\rho_L - \rho_s)d} \quad (20)$$

In the above equation, θ_{cr} is the critical shields parameter, and τ_{cr} is the shear stress that is applied by the water

stream to the sedimentary particles at the beginning of motion. The purpose of this model is to estimate the amount of sediments that have been eroded from the bed surface. Hence, the shear velocity parameter $\sqrt{\frac{\tau}{\rho}}$ is defined for measuring the power of water flow that needs to lift the sediments from the bed surface, and the rate of removal of sediment from the bed (u_{lift}) can be obtained using the follow equation [12]:

$$u_{lift} = \alpha n_s \sqrt{\frac{\tau - \tau_{cr}}{\bar{\rho}}} \quad (21)$$

where n_s is the normal vector of the bed surface, α is a dimensionless parameter indicating the possibility of the lifting of the sediment from the bed, and is usually equal to or less than 1. Now, the Eq. (22) is used to obtain the natural angle of sediment placement in the river bed. In Eq. 22, n and g are the normal vector of the surface and gravity vector, respectively. If the flow is constant, the minimum slope where the scouring hole walls can be stable can be determined according to the internal friction angle of the sediment particles. If the internal friction angle of the sediments is high (such as clay), that is, the stability of the wall at a steep slope is high. In materials where the angle is low (like sand), the scouring hole wall has a lot of tendency to failure and move forward [13].

$$\theta = \frac{n}{|g|} \quad (22)$$

The critical shear stress that required to erosion the inclined surface in Flow-3D software, can be calculated with considering the effect of the internal friction angle of the sediments, which is one of the input variable of the model, and calculated using the following equation: [13]

$$\tau_{cr} = \tau_{cr}^0 \sqrt{1 - \frac{\sin^2 \varphi}{\sin^2 \xi}} \quad (23)$$

According to the above equation, if the natural slope of the sediments is equal to their internal friction angle ($\xi=\varphi$), the critical shear stress will be equal to zero ($\tau_{cr} = 0$). This condition means that the bed surface will be eroded due to any shear stress which applied on it. Also, if the relation $\varphi > \xi$ is established, the relation ($\tau_{cr} < 0$) will be established, that is, the sediments will erode even if shear stress does not exist. This relationship also indicates that as the internal friction angle of the sediment particles increases, if the wall slope (φ) is increased, the scouring hole wall will also be eroded without any shear stress ($\tau_{cr} = 0$) [10]. The motion of suspended sediments in the system is expressed by the equation called the convection–diffusion equation. By adding expressions that related to drifting and lifting of the sediments, Eq. (23) can be written as follows:

$$\left(\frac{\partial c_s}{\partial t}\right)_x + u \cdot \nabla c_s = D \nabla^2 c_s - u_{lift} \cdot \nabla c_s - u_{drift} \cdot \nabla c_s \quad (24)$$

In Eq. 24, U is a fluid velocity and the parameters u_{lift} and u_{drift} are equal to the sum of the lifting and sedimentation rates of the particles, respectively. In general, the Flow-3D erosion model uses a simple and direct method to simulate erosion and sedimentation in three-dimensional flows. The simulation results indicate that the models created in three-dimensional state are in good agreement with the experimental conditions. In previous versions of this software, there were limitations in creating the correct scouring model, which in the new version (v. 11.2), these limitations have been resolved. For example, in the previous version it was not possible to simulate the coarse aggregate particles, and if this type of particles were simulated, the hypothesis governing the sedimentation in the model was ignored. In general, the bed load should be simulated in a model in which sediment particles should slipped or rotated on dense bed (or suspended in fluid flow). Also, in the case of erodible beds, it was only possible to define a specific type of particle in the model [13]. According to the preliminary studies carried out in this field, it is possible to simulate the scouring model of the inclined bridge pier with the help of the erosion equations provided in this section of the paper.

3 Experimental data

Flow and scouring modeling around the inclined pier of the bridge is performed on the group of the pile with different geometric shapes. One of the most important parts of the numerical studies is to measure the precision of the

simulation model. In this study, to determine the precision of the model, the scouring depth around the inclined pier of the bridge, which is located on the cubic foundation, has been compared in two cases; one was the simulation by the Flow-3D software, and the other was the empirical tests conducted by Esmaili Varaki et al. [1] in the Hydraulic Lab of the University of Guilan.

Due to the fact that the laboratory model is at a scale of 1.190 compared to the actual sample of the 8th bridge of Ahvaz, this study simulates and investigates the flow in a channel with the length of 6 m, the width of 0.92 m and the height of 0.6 m. The channel profile simulated in this model is shown in Fig. 2. Also, the geometric characteristics of the models of the bridge piers that located on the pile group with different geometric shapes are represented in this figure. In Table 1, the hydraulic parameters of the flow used in this study are written.

The amount of scouring around the piers depends on many parameters and each of these factors has a special and different effect on the scouring mechanism that may increase or decrease the erosion of the river bed around the bridge piers. These parameters include the type of fluid and the amount of bed sediment, the pier geometry and the hydraulic specifications of the flow, and they can be expressed in the form of the following equation: [1]

$$f_1 = (y, d_s, D, D_p, T_p, d_p, l_m, l_n, D_{50}, Z, U, \rho, \mu, \alpha, t, t_e) \quad (25)$$

where y is the depth of flow, d_s is the scour depth, D is the bridges' pier width, D_p is the width of the pile cap, T_p is the thickness of the pile cap, d_p is the pile diameter, l_m is the distance of the piles in a row, l_n is the distance of the piles in a column, D_{50} is the diameter of 50 percent of particles which have a smaller diameter compared to it, Z is the leveling of the pile cap, U is the average velocity of the

Fig. 2 Geometric characteristics of the inclined bridge pier and the group of the pile placed beneath the pier

flow, ρ is the density of the water, μ is dynamic viscosity, g is gravity acceleration, α is the angle of piers inclination in the parallel plane with the flow, t is the time since the start of the scouring and t_e is time to reach the scouring equilibrium. Figure 3 shows the important geometric parameters of the inclined pier group.

3.1 Sensitivity analysis of the meshing in computational grid

In the model gridding process, in addition to the number of meshes, the size of the meshed in each direction also influences the modeling results. In the case where the

variable size meshes are used in the same direction, the software automatically applies the size of the meshes in that direction, but in the case where two orthogonal directions have been designed, the cell shape ratio in a solving grid must be considered carefully (which is the number obtained by dividing the largest cube dimension into its smallest dimension) and should not be greater than 4. In the model designed in this research, the model was created in 3 different modes with different mesh sizes and the accuracy of the results of these three modes was investigated using simulation data, and finally, an optimal design was selected for the model.

Table 1 Geometric characteristics and hydraulic flow parameters used in current research models

Model	Relative velocity (U/U_e)	Relative depth of flow (y/D^*)	Relative foundation leveling (Z/T_{pc})
Inclined bridge pier on the cubic foundation	0.95	1	At the same level with the bed
Inclined bridge pier on the cylindrical pile group	0.95	1	-2, -1, 1
Inclined bridge pier on the rectangular pile group	0.95	1	-2, -1, 1
Inclined bridge pier on the rounded rectangular pile group	0.95	1	-2, -1, 1
Inclined bridge pier on the aerofoil-shaped pile group	0.95	1	-2, -1, 1

Fig. 3 Geometric Parameters of the inclined pier group. **a** In direction of the flow, **b** against the flow

Table 2 Sensitivity analysis of the meshing in the computational grid in this research

Mode	No. of meshes	The ratio of the maximum dimensions (shape ratio)	Maximum relative scouring depth (d_s/D) [1]	Maximum relative scouring depth (d_s/D) (numerical simulations in FLOW-3D)	Mean absolute percentage error (MAPE %)	Root Mean Square Error (RMSE)
1	891,260	1.86	0.990	0.694	29.89	0.296
2	128,156	1.43	0.990	0.791	20.10	0.199
3	1,860,000	1.01	0.990	0.898	9.25	0.092
4	> 1,860,000	Requires high processing system and more time for simulation				

To compare the results, Table 2 has been used and the results obtained for the maximum scouring depth (d_s/D) (maximum scouring depth to width of foundation) in the current numerical model with experimental results that obtained by Esmaili Varaki et al. [1] have been compared. The mean absolute percentage error (MAPE %) and the root mean square error (RMSE) have been calculated for each group of the results. In Eqs. 26 and 27 the error estimation method is presented [1].

MAPE (%)

$$100 \times \frac{1}{n} \sum_{i=1}^n \left| \frac{X_{\text{exp}} - X_{\text{num}}}{X_{\text{exp}}} \right| \quad (26)$$

RMSE (m^3/s)

$$\sqrt{\frac{1}{n} \sum_{i=1}^n (X_{\text{exp}} - X_{\text{num}})^2} \quad (27)$$

where X_{exp} is the amount of experimental data, X_{num} is the quantity of numerical data, and n is the number of data.

Given the amount of errors in all of the meshing modes, mode 3 was selected as the best mode in terms of the number of meshes which have 1,860,000 computational cells. In this case, the meshes were selected uniformly in each channel direction with dimensions of 0.99, 0.1 and 0.1 in length, width and height, direction, respectively. This mode has the highest computational accuracy and the best simulation time. Also, the largest shape ratio in this model is 1.01 which is smaller than 3, and therefore, the selected gridding is also appropriate in this regard. For model results not to be affected by the number and size of the meshes, the meshing of all of the studied models in this research has been selected similar to each other [1].

3.2 Comparison of numerical results with experimental results

One of the methods to verify the accuracy of the simulated model and determine the error percentage of the data obtained from the FLOW-3D software and compare its results with the experimental data is to use Eq. (28)

$$E = \frac{((d_s/D)_N - (d_s/D)_E)}{(d_s/D)_E} \times 100 \quad (28)$$

where E represents the relative error percentage, P indicates the maximum relative scouring depth in numerical solution method and P equivalent to the maximum scouring depth in the experimental method. In this research, to validate the model of the development of scouring around the inclined bridge pier group with time, the

Fig. 4 Comparison of scouring development with time in numerical and experimental results for $Z/D=0$ and $U/U_c=0.95$ foundation levels

Fig. 5 Fitting of Maximum scouring depth (d_s/D) in numerical and experimental results

results obtained in simulation with the FLOW-3D software have been compared with similar charts in experiments conducted by Esmaili Varaki et al. [1] in foundation leveling of $Z/D=0$ (The ratio of distance between flow bed and foundation to the foundation width). Figures 4 and 5 show the comparisons of developing the scouring around the inclined pier group in the $Z/D=0$ and $U/U_c=0.95$ foundation levels which are obtained from numerical and experimental results and show a relative error percentage in each method [1].

As it is shown in Fig. 4, the development of scouring with time in numerical results is similar to the experimental results obtained by Esmaili-varaki et al. [9]. In Fig. 5, the mean value of the relative error of the maximum scouring depth and the RMSE error value in the numerical results are 7.43 and 0.49, respectively. Table 3 shows the relative error values and RMSE for maximum scouring depth obtained by simulation and experimental results [1].

Table 3 Relative error values and RMSE of maximum scouring depths obtained from simulation and experimental results

RMSE	Relative error percent-ages (%)	maximum scouring depths that obtained from experimental results	Maximum scouring depths that obtained from simulation results	Time (min)
008/0	20/7	123/0	115/0	17
026/0	34/5	500/0	473/0	37
035/0	49/6	548/0	513/0	89
058/0	11/9	637/0	579/0	175
048/0	26/7	672/0	623/0	312
074/0	22/9	803/0	729/0	715
080/0	33/8	969/0	888/0	1553
091/0	25/9	990/0	898/0	1784
090/0	12/9	988/0	898/0	2767

Fig. 6 Input the properties of sediment particle into the Flow-3D software

As it can be seen in Table 3, the highest relative error and RMSE at the time of 1784 min are 9.25% and 0.91%, respectively. It can be said that the numerical simulation results and experimental results are in good agreement; therefore, the simulation model can be used to evaluate other conditions and analyze the obtained results.

4 Model setup

In this part of the paper, different stages of creating a numerical model for simulating the scouring process at the inclined bridge pier and the method of entering the model and flow information in the Flow-3D software is described. One of the important tasks for creating this model, which makes the physical conditions and sediment transport during the scouring process well-modeled, is to enter the information of the bed sediment particles. This

is done using the sediment scour option in the software. In this research, the information of this section has been completed using the properties of sediment particles expressed by Esmaili Varaki et al. The river bed sediment particles have been selected as non-cohesive sand particles with a diameter of 0.7 mm and a relative density of 2.65. In Fig. 6, the information entered into the software has been indicated.

The most important stage of the modeling process in the Flow-3D software is using the meshing and geometry tab. In this tab, the geometric conditions of the simulation environment are designed in the software, and network meshing, boundary conditions, and initial conditions are applied to the model. The geometry of the model in the Flow-3D software is defined in three methods:

4.1 First method

In the software itself and in the Primitive Creation section, there are tools that can create simple geometries, and with the choice of each of these tools, dimensions and material of the geometry can be defined. Various independent parts of any designed geometry are called Component and each Component can have multiple Subcomponents. The sub-components associated with each Component have the same materials and properties that are defined for the entire Component. In the definition of the geometric components of a model, the smaller the number of the components, the simpler the modeling will be and the simulation time will be shorter.

4.2 Second method

In this method, software like 3D AutoCAD can be used and the geometry of the model is designed and after integrating the model, that model opens in the Flow-3D software. It should be noted that if this method is used, the system of measurement units in the geometry designing program

Fig. 7 The method of opening a geometry designed in programs like AutoCAD in the Flow-3D software

Fig. 8 Method of opening the 3D Model of the Channel Geometry with inclined bridge pier Using STL Option in Flow-3D Software

Fig. 9 The 3D geometry of the channel with an inclined bridge pier in Flow-3D software

and the Flow-3D software must be similar and the dimension unit should be selected the same. The geometry drawn in the AutoCAD program has the stl format and can be opened in the Flow-3D software using the icon shown in Fig. 7.

4.3 Third method

If the aim is to model the flow of a river, then the topography of the river should be saved as a rigid body into the Flow-3D program. River topographical information is typically provided by the mapping team and has an ASCII format. To use this information in the Flow-3D software, data must be prepared in a format that can be readable for this software. For this reason, the file containing the topographic information of the location is stored in the stl format and opened in the software.

In this research, the second method and the AutoCAD software have been used to design the geometry of the

inclined bridge pier models. At first, the 3D geometry of the channel model, with its inclined bridge piers is plotted in the AutoCAD software and stored as an stl file, and then opened with the STL icon in the Flow-3D software. Figures 8 and 9 show the steps of opening the 3D model in the Flow-3D software.

To determine the bed sediment particles in the Flow-3D software, a three-dimensional profile of the sediment is required. Therefore, for designing the sand gravel cross section in the software, the cross-section was designed according to the method specified in the following Fig. 10.

Once the geometry of the model is drawn, the solving area should mesh. The Flow-3D software only has the ability to create the ordered cubic solving grids. If the cubic grids are used, the ability to produce results has been increased and also the speed of the software to create and solve the problem has been increased. The gridding of the model is done using the Mesh-Cartesian section on the meshing and geometry tab. In this step, various

Fig. 11 Final gridding for the inclined bridge pier model

Fig. 12 Flow lines in x–y plane when the pile cap are above the bed ($Z/T_p=0$); **a** near the bed ($Z_w=1$ cm); **b** in the middle depth ($Z_w=6$ cm); **c** near the water surface ($Z_w=9$ cm)

methods are used to create the mesh, and either the same size meshes can be created in the whole model, or different sized meshes can be created for each dimension.

Even in one dimension, a non-uniform size for the meshes can be used, and in the sections of the model that have the more turbulence or the results of modeling are more important, a smaller size meshes can be used. In general, it can be said that by minimizing the size and increasing the number of meshes, the simulation results will be more accurate, but by minimizing the size of the meshes, the increasing trend of the model accuracy is stopped and below a certain limit, if smaller meshes are selected, the accuracy of the model results will not be increased, but

only the modeling time will increase. In such a condition, the size of the output files of the software will also increase and will be problematic. Therefore, there is an optimal mesh number for achieving the desired accuracy, which is different in each model.

In the gridding process, in addition to the number of meshes, the size of the meshed in each direction also influences the modeling results. In the case where the variable size meshes are used in the same direction, the software automatically applies the size of the meshes in that direction, but in the case where two orthogonal directions have been designed, the cell shape ratio in a solving grid must be considered carefully (which is the number

Fig. 13 Flow lines in x–y plane when the pile cap are in the same level with the bed ($Z/T_p = -1$); **a** near the bed ($Z_w = 1$ cm); **b** in the middle depth ($Z_w = 6$ cm); **c** near the water surface ($Z_w = 9$ cm)

obtained by dividing the largest cube dimension into its smallest dimension) and should not be greater than 4. In the model designed in this research, the model was created in 3 different modes with different mesh sizes and the accuracy of the results of these three modes was investigated using simulation data, and finally, an optimal design was selected for the model. The gridding information of the model such as the number and size of meshes in each direction is shown in Fig. 11.

In Fig. 11, 1,963,500 of meshes have been used in the flow and the inclined bridge pier model grid. The meshes

in each channel direction were uniformly created with dimensions of 0.02, 0.02 and 0.02 mm, respectively in three directions in order to reach the highest computational accuracy and at the same time the optimal simulation time. Also, according to Fig. 11, it is clear that the maximum shape ratio in this model is 1.01, which is smaller than 4, and therefore the designed network is also appropriate. For the results of the model are not affected by the number and size of the meshes, the same gridding was created for each of the 15 models studied in this paper, all of which are in accordance with Fig. 11.

Fig. 14 Flow lines in x-y plane when the pile cap are below the level of the bed ($Z/T_p = -2$); **a** near the bed ($Z_w = 1$ cm); **b** in the middle depth ($Z_w = 6$ cm); **c** Near the water surface ($Z_w = 9$ cm)

The final grid designed for the inclined bridge piers is shown in Fig. 11.

5 Result and discussions

5.1 The flow pattern around the inclined pier group

The flow pattern around the pier group is different from the flow pattern around the single pier. In the case where the flow is simulated near a pier group, the upstream pier operates like a generator of the turbulent flow and the downstream pier acts like a device that reduces the flow

resistance [14]. The results are related to the flow conditions on the xy plane around the pier group in the following three situations are shown in Figs. 12, 13 and 14, respectively. (1) Placement of pile cap above the bed ($Z/T_p = 0$), (2) At the same level with bed ($Z/T_p = -1$) and (3) Below the bed ($Z/T_p = -2$).

As it can be seen in Fig. 12, when the pile cap is located above the bed, the flow vortices formed between the first and second piers of the pier group and formed at the middle depths and also near the water surface and formed in the total distance between the two piers; At the downstream of the pier group, the flow vortices are located near the bed to the middle depth of the flow.

Fig. 15 Total shear stress in the flow bed at different levels. **a** Above the bed ($Z/T_p=0$); **b** same level with bed ($Z/T_p=-1$); **c** lower than the bed ($Z/T_p=-2$)

As shown in Fig. 13, when a pile cap is on the same level with the bed, from near the bed to the middle depths of flow and at the distance between the two piers, the rotational flows develop while near the surface on the upstream side of the second pier, the flow lines are parallel to the river bed.

As it can be seen in Fig. 14, when the pile cap is lower than the bed surface, due to the increased distance between the piers, the rotational flows in each of the three

depths of flow and in the downstream of the first pier are more intense and the flow lines in the upstream of the second pier are approximately parallel to the bed. It is worth noting that, at all levels of the pile cap, the longitudinal component of the flow velocity increases as it reaches the first pier. The increase in this region and its extension to the downstream has been occurred due to the narrowing of the flow which, in turn, is due to the placement of the pier and rotational lifting flow on the downstream side.

Fig. 16 Longitudinal profile of scouring around the inclined pier group with piles that have different geometric shapes. **a** C1, **b** R1, **c** P1, **d** A1

Fig. 17 Scouring hole around the inclined pier group on the piles which have different geometries

Fig. 18 Reduction percentage of the maximum scouring depth with the piles which have different geometries than a cylindrical pile

This condition causes the flow rate between the pier group to approach zero, which is due to the formation of the downward vortices and flows. Figure 15 shows the total shear stress created in the flow bed at different levels of the pile cap placement.

Comparison of stress variations at different levels of pile cap placement showed that the highest shear stress is created when the pile cap position is at the same level as the river bed ($Z/T_p = -1$), and by placing the pile cap at a lower level than the river bed ($Z/T_p = -2$), the maximum shear stress decreases. This may be due to the fact that in this case, the distance between the pier group increases and the presence of the second pier decreases the flow rate in the pier group and different pier in the one pier group act as the two Independent piers in the formation of flow pattern. Decreasing the shear stress occurs when

Fig. 19 Scouring hole around the inclined pier group with different leveling of pile caps

the pile cap is located at a level above the bed. The reason for decreasing shear stress in these conditions ($Z/T_p = 0$) is that the pile cap thickness acts as a ring and prevents the direct collision of the vortices with the sedimentary bed. Therefore, by examining shear stress variations in different levels of the pile cap, it is expected that the maximum scouring depth around the pier group will occur at the level of the bed surface. Figure 16 shows the amount of flow vortices in the x - z plane at each of the three levels of the pile cap.

As shown in Fig. 16, the aerofoil-shaped pile has a better effect on reducing scouring than other geometric shapes such that the maximum scouring depth around the inclined pier group using an aerofoil shaped pile is 36.3 cm, while the maximum scouring depth for cylindrical (circular), rectangular and rounded rectangular piles is equal to 18.4, 2.91, 3.59 cm, respectively. The greater effect of the aerofoil pile is related to its geometric shape, which reduces the lifting vortices. Figures 17 and 18 show the scouring hole around the inclined pier group and the column diagram of the reduction of the maximum scouring depth percent using piles with different geometric shapes than the cylindrical pile, respectively.

Figure 19 shows the scouring hole around the inclined pier group and at the lower part of the pile cap in different levels of the pile caps.

Figure 19 shows that the highest and lowest amount of scouring hole and erosion occurs in the conditions that the pile caps are in the same level as the bed and below the bed, respectively. Also, Fig. 20 compares the final longitudinal section of scouring at different leveling of pile caps.

By comparing the results of the maximum scouring depth at different leveling of the pile caps, it is concluded that the largest reduction in scouring depth occurs in aerofoil-shaped pile caps and pile caps with the sharper nose and aerodynamic shape are good options to control the horseshoe vortices and will reduce the scouring hole depth. Also, when the pile cap is at the same level with the bed, the penetration power of the flow into the space below the pile cap has been increased, which increases the power of the flow erosion and the maximum scouring depth occurs at this level.

In this research, the geometric shape of the piles installed under the inclined piers and pile cap leveling on the sedimentary bed has been studied and the effects of them on the properties of scouring around the inclined pier group have been investigated numerically by the FLOW-3D software.

Fig. 20 Final longitudinal sections of the scouring at different pile cap leveling. **a** Above the bed ($Z/T_p = 0$); **b** in the same level as the bed ($Z/T_p = -1$); **c** lower than the bed ($Z/T_p = -2$)

6 Conclusions

Due to the importance of identifying the mechanisms affecting the scouring on this types of bridge piers, the effect of the geometric shape of the piles that installed under the inclined piers and also the effect of the pile cap leveling in the sedimentary bed have been investigated and the properties of the scouring around the inclined pier group was studied numerically and using the FLOW-3D software. The study of total shear stress in the flow bed at different leveling of the pile caps shows that of the highest shear stress was created when the pile cap position is at the same level with the river bed and by installing the pile cap at a lower level than the river bed, the maximum shear stress decreases. This may be due to the fact that in this case the distance between the pier group increases and

the presence of the second pier decreases the flow rate in the pier group and different pier in the one pier group act as the two Independent piers in the formation of flow pattern. By comparing the final longitudinal sections of the scouring at different leveling of the pile cap, it is concluded that the largest reduction in scouring depth occurs in aerofoil-shaped pile caps and pile caps with the sharper nose and better aerodynamic shapes are good options to control the horseshoe vortices and will reduce the scouring depth around the inclined pier group.

Conclusion points of this research are:

1. The study of the longitudinal sections of the scouring around the inclined pier group shows that the aerofoil-shaped pile has a better effect on reducing scouring than other geometric shapes such that the maximum

scouring depth around the inclined pier group using an aerofoil shaped pile is 36.3 cm, while the maximum scouring depth for cylindrical (circular), rectangular and rounded rectangular piles is equal to 18.4, 91.3, 59.3 cm, respectively. The greater effect of the aerofoil pile is related to its geometric shape, which reduces the lifting vortices.

2. The greatest amount of scouring occurs when the pile cap is at the same level as the river bed ($Z/T_p = -1$). In this condition, the empty space below the pile cap gradually increases and some of the water flow penetrates it, causing erosion and scouring around the pile groups. One of the important points is that in this case, the geometric shape of the pile group is effective in reducing erosion and scouring that occurs under the pile cap. Under the same conditions of the pile cap leveling, the aerofoil-shaped pile caps have the lowest size of scouring hole and erosion amount.
3. When the pile cap is above the bed level ($Z/T_p = 0$), the thickness of the pile cap acts as a ring and prevents the direct collision of flow vortices to the bed, and the scouring hole depth is less than the conditions that the pile cap and the bed are in the same level ($Z/T_p = -1$). Also, if the pile cap is at the lower level than the bed ($Z/T_p = -2$), the amount of erosion and scouring in the space below the pile cap is less than the two other leveling conditions, which is due to the large distance of the space below the pile cap with the flow bed and leads to reduce the penetration power of the flow vortices into the space below the pile cap.

Acknowledgements Not applicable.

Compliance with ethical standards

Conflict of interest The authors declare that they have no conflict of interest.

References

1. Esmaili Varaki M, Saadati S (2015) The experimental study of the effect of the angle of pier group deployment on the foundation on the scouring around the bridge pier. *J Sci Water Soil* 25(4/2):27–39
2. Hannah CR (1978) Scour at pile groups. University of Canterbury, N. Z., Civil Eng. Research report no 78
3. Melville BW, Sutherland AJ (1988) Scour protection at bridge piers. *J Hyd Eng* 81 ASCE 111(11):1811–1881
4. Parola AC, Mahavadi SK, Brown BM, El-Khoury A (1996) Effects of rectangular foundation geometry on local pier scour. *J Hydraul Eng ASCE* 122(1):35–40
5. Melville BW, Raudwicky AJ (1996) Effects of foundation geometry on bridge pier scour. *J Hydrol Eng ASCE* 188(1):e813–e819
6. Melville BW, Chiew YM (1999) Time scale for local scour at bridge piers. *J Hydraul Eng ASCE* 125(1):59–65
7. Ataie-Ashtiani B, Baratian-Ghorghi Z, Beheshti AA (2010) Experimental investigation of clear-water local scour of compound piers. *J Hydraul Eng ASCE* 136(6):343–351
8. Amini A, Bruce W, Melville M, Thamer M, Abdul H, Ghazali A (2012) Clear-water local scour around pile groups in shallow-water flow. *J Hydraul Eng ASCE* 138(2):177–185
9. Esmailvaraki M, Hosseini S, Fazloli R (2012) Experimental study of the effect of makeup and diameter of the piles on the scour depth inclined pier group based on the pile group. In: Thirteenth Iranian hydraulic conference. University Tabriz
10. Wen and King (2012) CFD prediction of local scour hole around bridge piers. *J Cent South Univ* 19:237–281
11. Hedayatifar M, Pourlk M (2014) Flow dynamics simulation with flow-3D. Attihenger Publications, Tehran
12. Maroosi M, Roshan R, Sarkordeh H (2014) Analysis and design with flow-3D software. Fadak isatist Publication, First printing, Tehran
13. Ghasemzadeh F, Shinyzadeh P (2015) Simulation of hydraulic issues in flow-3D. Innovative Publishing, Tehran
14. Sadat Jafari M, Ayoubzadeh A, Esmaili Varaki M, Rostami M (2016) Simulation of the flow pattern around the inclined pier group using the flow-3D numerical model. *J Water Soil* 30(6):1860–1873

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.