Effect of Bridge Pier Shape on Depth of Scour

Halah Kais Jalal1, Waqed H. Hassan2

1 Graduate student, Civil Engineering Department, University of Kerbala, Kerbala, Iraq. E-mail: halakais94@gmail.com
2 Professor, University of Kerbala, Kerbala, Iraq. E-mail: Waaqidh@uokerbala.edu.iq

Abstract: Scour at bridge piers is a common problem in infrastructure engineering, as local scour around bridge piers is one of the most common causes of bridge failure. Prediction of the maximum depth of local scour thus plays an important role in ensuring safety and economy during design and maintenance. A computational fluid dynamics (CFD)-based simulation methodology for computing the depth of local scour around bridge piers using Flow-3D model is thus proposed in this study. The objective is to investigate the effects of the bridge pier shape on local scour in order to develop an optimal hydraulic design for minimum depth of scour. Local scouring around different shapes of pier (circular, rectangular, square, octagonal, elliptic, and lenticular) in non-cohesive bed sediment under clear water scour conditions was thus simulated for each pier shape at different flow intensities, fluid depths, and pier sizes. By comparing the numerical results for predicted scour depth around a circular pier with the laboratory experimental results produced by Melville in 1975, the model was found to have about a 10% error rate for prediction of scour depth, demonstrating good agreement with experimental models. The model results revealed that the rectangular pier shape recorded the maximum depth of scour, while the minimum depth of scour was measured for the lenticular shape, this being about 40% lower than for other shapes. All factors investigated were found to have direct effect on scour but the most significant factor was pier width; scour reached its maximum depth value at pier width ratios of 0.2 in rectangular pier shapes; the results also showed that scour depth was comparatively higher upstream and lower downstream. Overall, the proposed numerical simulation Flow-3D method is a reliable tool for predicting and generating discussion of spatial influences on the bridge scour process.

Keywords: Bridge Scour, Flow-3D, Pier Shapes, CFD

1. Introduction

Scouring at bridges is defined as the lowering of the stream bed at piers and abutments as a result of the erosive action of flowing water such that there is a tendency for the foundations of riverine structures such as bridges to become exposed [1]. Many contemporary researchers have studied the issue of scouring extensively from different points of view and under different conditions, as it is recognised as one of the major causes of bridge failures alongside collisions and overloading, leading to massive loss of life and economic impacts, as pointed out by Shepherd [2]. Scour can be caused by either base-flow or by flood events, when the bed shear stress at the pier foundation exceeds the critical bed shear stress. Usually the scouring rate is most robust during flood events [3].

The scour at a bridge site generally consists of three components: 1) general scour, 2) contraction scour, and 3) local scour [4]. Among the three scour components, local scour, the main object of the present study,
plays the most important role in scour-related risks to bridges. In such scouring, a downward flow is induced at the upstream end of the pier that leads to local erosion around the pier. The extent of scouring depends on the balance between streambed erosion and sediment deposition [5], and there are two types of sediment transport in the approaching flow, clear water scour and live bed scour. In clear water scour, no sediments are delivered by the river’s approaching flow, while an interaction exists between sediment transport and the live bed scour process [6].

Many contemporary studies have focused on local scour phenomena in vertical bridge piers either experimentally and/or theoretically by considering a number of parameters related to this phenomenon. However, such experimental scour tests are mostly conducted for large-scale and important bridges as they are expensive and labour-intensive, though some studies have investigated analytical solutions to bridge scour for practical and economic purposes. This study focuses on the influence of multiple variables on local scour depth around a bridge pier of different shapes to determine optimum hydraulic design of pier shape [7]. The main variables thus investigated are velocity, pier diameter, water depth, and pier shape. The shape of a pier is one of the important factors that plays an essential role in the creation and the strength of the vortex system, and as such, the effect of pier shape on local scour has been studied by several researchers ([8], [9], [10], [11] and [12]). By carefully examining recent studies, it becomes clear, however, that circular shapes are studied most extensively by many researchers ([9], [12], [13], [14], [15], [16] and [17]).

In recent years, the ever-increasing capabilities of computer hardware and software have allowed computational fluid dynamics (CFD) to be more widely used to determine fluid flow behaviour in industrial and environmental applications. The basis of numerical simulation is that rather than designing a big model and using expensive instruments such as a velocimeter to measure key variables, basic fluid behaviour including velocity distribution, turbulence kinetic energy, bed shear and stresses can be obtained by applying computational fluid dynamics (CFD) programs such as FLUENT, CFX, and PHOENIX. The application of simulation software is similar in many ways to the set-up of an experiment where Computational Fluid Dynamics (CFD) methods are always used for the simulation of flow process by discretisation and solving of Navier-Stokes and continuity equations for the computational cells.

The 3D scour simulation method has thus been adopted by many researchers in recent decades. For example, [18] developed a numerical model to simulate the field scour data, which was verified by the measurements obtained in the laboratory. Similarly, [19] developed a 3D model to simulate the complex local flow field around the scour area using a finite-volume method, while [20] used a numerical model to simulate of local scour in complex bridge piers under tidal flow, and [21] used a 3D CFD model to examine scale effects on turbulent flow and sediment scour. All of these studies assumed that running water under a clear-water environment predominantly induces most of the interactions between the flow and riverbed, however.

Flow-3D is a commercial CFD package with special modules for hydraulic engineering applications; it employs numerical techniques to solve fluid motion equations to obtain both transient and three-dimensional solutions to multi-scale multi-physics flow problems. Its array of physical and numerical options allows users to apply the Flow-3D program to a wide variety of fluid flows and heat transfer phenomena, and this software is thus widely used for solving various hydraulic problems [22].

In this paper, a comparison between the numerical Flow-3D results and measurements from the Melville experiment [9] is used to verify the former’s calculations. The main purpose is to evaluate the model's ability to predict and simulate scour-hole depth. The parametric analysis of a case study was conducted using the proposed simulation model to investigate the influence of certain parameters on bridge scour development, including pier shape, flow intensity, flow depth, and pier size.

2. Experimental Data

To validate the effectiveness of the selected method, numerical simulations of flow and bed deformation around a cylindrical pier were compared with laboratory experiment results from the Melville experimental [9]. As per that experiment, sand was initially placed in a flume with dimensions 19 m long and 45.6 cm
wide. A cylinder with diameter \( b \) of 5.08 cm was simulated as a bridge pier model. The bed material was relatively uniform sized sand with a median grain size \( d_{50} \) of 0.385 mm, a height of 12.7 cm, and a density of 2650 kg/m\(^3\). The approaching mean flow velocity was 0.25 m/s. Water was thus discharged at the rate of 0.01712 m\(^3\)/s, with water depth of 15 cm. Figure (1) shows a sketch of the experiment setup.

Figure (1). Plan view of Melville experimental setup [9].

3. Numerical Model

Flow-3D CFD code was used to simulate the scour process around a bridge pier. The program uses a structured orthogonal grid and illustrates the complex boundaries of the solution domain by the application of a fractional area/volume method (FAVOR), which allows rectangular computational cells to be partially blocked by an obstacle. The sharp free surfaces (e.g. hydraulic jumps, free jets in air) are modelled using the Volume-of-Fluid (VOF) method.

3.1 Governing equations

The equations involved the motion of a viscous fluid, namely the continuity equation and the momentum conservation equation, are known together as Navier- Stokes equations:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j}
\]

Continuity equation

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

In the Reynolds-averaged Navier-Stokes approach, the Navier-Stokes equations are averaged over a time interval or across a grouping of equivalent flows. The goal of this approach is to obtain the mean effect of turbulent quantities [23]. The Reynolds-averaged Navier-Stokes (RANS) equations for a viscous incompressible Newtonian fluid are

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} - \frac{\partial \left( \bar{u}_i \bar{u}_j \right)}{\partial x_j}
\]

where:

\[
\sigma_{ij} = 2\nu s_{ij}
\]

\[
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]
\[
\frac{\partial \mathbf{u}_i \mathbf{u}_j}{\partial t} = v_t \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k
\]

(6)

where \( u_i \) is the fluid velocity component in \( i \) direction, \( \mathbf{u}_i \) is the fluctuation of fluid velocity in \( i \) direction, \( P \) is the pressure, \( s_{ij} \) is the strain rate tensor, \( \mathbf{u}_i \mathbf{u}_j \) is the Reynolds stress tensor, \( \rho \) is the fluid density, \( v \) is the fluid kinetic viscosity, \( v_t \) is the turbulence viscosity, \( k \) is the turbulent kinetic energy, and \( \delta_{ij} \) is the Kronecker delta (\( \delta_{ij} = 1 \) if \( i = j \); \( \delta_{ij} = 0 \) if \( i \neq j \)).

3.2 Turbulence Model

In Flow-3D, six turbulence models are available: 1) the Prandtl mixing length model, 2) the one-equation, 3) the two-equation \( k - \varepsilon \) model, 4) RNG, 5) \( k - \omega \) models, and 6) a large eddy simulation LES model.

Work in [24] introduced the RNG-based \( k - \varepsilon \) model; the main characteristic of this model is the numerical constants in the \( k - \varepsilon \) model, which are directly obtained from renormalization group theory, and are in good agreement with the standard \( k - \varepsilon \) model. However, a constant equation has been empirically shown in the standard \( k - \varepsilon \) model that must be derived explicitly in the RNG model. Generally, however, the RNG model has wider applicability than the standard \( k - \varepsilon \) model. The governing equations are

\[
\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_i} = T_{ij} - \varepsilon
\]

(7)

\[
\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_i} = C_{1e} \frac{\varepsilon}{k} T_{ij} - \frac{1}{\rho} \left[ \frac{\partial}{\partial x_j} \left( \frac{\mu + \frac{\mu_t}{\sigma_k}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} + \frac{\varepsilon}{k} \right) - C_{2e} \rho \frac{\varepsilon^2}{k} \right]
\]

(8)

where \( k \) is the Reynolds-averaged kinetic energy, \( \varepsilon = \frac{1}{2} \mathbf{u}_i \mathbf{u}_j ; \varepsilon \) is the dissipation rate of turbulent kinetic energy; \( \varepsilon = v \left( \frac{u_i}{\partial x_k} \right) \left( \frac{u_i}{\partial x_k} \right); \mu_t \) is the turbulent eddy viscosity; \( \mu_t = \frac{c_{\mu} \rho k^2}{\varepsilon} \); and \( T_{ij} = \frac{\mu_t}{\rho} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \) where \( C_{\mu} , C_{1\varepsilon} , C_{2\varepsilon} , \sigma_k , \) and \( \sigma_\varepsilon \) are model coefficients, and have a values of 0.085, 1.42, 1.39, 0.7179, and 0.7179 respectively. In this study, the RNG model was used to simulate the bridge pier turbulent flow for two reasons: 1) the model has a better agreement with experimental observations, and 2) this model is considered to be the most accurate and strongest model available in the software for scour simulations. The RNG model was also found to perform better for scour simulations in [25].

3.3 Boundary Conditions

Determination of the appropriate boundary conditions is one of the most important phases of numerical flow analysis, as the boundary conditions should be well matched with the physical conditions of the problem. In the simulation performed in the present study, the boundary condition of the numerical model is the specific velocity at the inlet, with different velocity ranges used in simulation for the outflow at the outlet and symmetry on the top, right and left sides, and the wall on the bottom.
3.4 Mesh Generation

The mesh for Flow-3D software has a cell of cubic shape that must be considered as one of the affecting factors in the simulation process. The number and size of mesh geometry can alter the time of operation, and thus these parameters should be adjusted to appropriate values to obtain acceptable results. Several different cell sizes were thus selected (3, 2.5, 1.5, 1, and 0.4 cm) to identify the best cell size to satisfy the phenomenon conditions. To obtain better observations of scour development around a pier, two mesh planes with finer resolutions were defined for both sides of the pier in the x and y directions. Minimum cell size near the pier was about 0.4 cm and the largest cell size was limited to 1 cm. In total, 252,000 cells were generated for the field section, as seen in Figure (3).

Figure (2): Boundary conditions of the typical problem.

Figure (3). Meshing structure of the 3D model for simulation of scour around a pier.

4. Numerical Method Description

The computational domain in this study matched the Melville laboratory experiments [9]. For the computational domain in the numerical simulation, the dimensions of the flume were scaled in order to save computation time, however. Different shapes of pier (circular, rectangular, square, octagonal, elliptic and lenticular) were used in the simulations, as shown in Figure (4). The pier models with different shapes were located in the flume in such a manner as to have a constant length to width ratio of length/width=2, L=2b, and pier width 5.08cm.

Figure (4). Shapes of piers used in the simulations.
Figure (4). Different pier shape models and dimensions.

For the computational domain in the numerical simulation, the inlet was located at a distance of 6b, where b = vertical diameter of pier, upstream of the cylinder, whose diameter was the same as in the Melville experiment [9]. The outlet was located at a distance of 14b downstream of the cylinder; thus, the whole length of the numerical model was 20b, as seen in Figure (5). One solid component was settled at the outer sides of the geometry (start of the channel) to prepare an inflow bottom at the top edge of the sediment at elevation of 0.127 m in order to prevent upward movement of sediments at the initiation of simulation. The water depth was 0.15 m above the sediment level, and all other parameters in the model were the same as those used in the experimental model.
The present study then changed the parameters v/vc, y/b, pier diameter, and ks to examine which parameter affected scour depth most. In addition, running time was set to 30 minutes, as the sediment transport reached balanced state of scour developing at this time, and the simulation of 30 min takes about seven days for all different cases in simulation.

5. Sediment Scour Model

The sediment scour model was assumed as a multiple non-cohesive sediment species with different properties including grain size, mass density, critical shear stress, angle of repose, and parameters for entrainment and transport. The model used two concentration fields: the suspended sediment and the packed sediment. Sediment is entrained by picking up and re-suspension from shearing and small eddies in the packed sediment interface. There is no way to compute the flow dynamics for each individual grain of sediment, and thus an empirical model must be used. The model used was based on Mastbergen and Van den Berg [26] with the addition of the Soulsby-Whitehouse equation [27] to forecast the critical Shields parameter. The first step for computing the critical Shields parameter is calculating the dimensionless parameter $d_{s,i}$:

$$d_{s,i} = d_i \left[ \frac{\rho_f (\rho_i - \rho_f)}{\mu_f^2} \frac{\|g\|}{\tau} \right]^{1/3} \quad (9)$$

where $\rho_i$ is the density of the sediment species $i$, $\rho_f$ is the fluid density, $d_i$ is the diameter, $\mu_f$ is the dynamic viscosity of fluid and $\|g\|$ is the magnitude of the acceleration of gravity $g$. Thus, the dimensionless critical Shields parameter was computed using the Soulsby-Whitehouse equation [27].

$$\theta_{cr,i} = \frac{0.3}{1 + 1.2 d_{s,i}} + 0.055 \left[ 1 - \exp \exp (-0.02d_{s,i}) \right] \quad (10)$$

The local Shields parameter can be computed based on the local bed shear stress, $\tau$:

$$\theta_i = \frac{\tau}{\|g\| d_i (\rho_i - \rho_f)} \quad (11)$$

where $\tau$ is calculated using the law of the wall and the quadratic law of bottom shear stress for 3D turbulent flow and shallow water turbulent flow, respectively, with some consideration of bed surface roughness. It was thus assumed that the Nikuradse roughness of the bed surface, $K_s$, was proportional to the local median grain diameter in packed sediment $d_{50, packed}$:

$$K_s = C_{Rough} d_{50, packed} \quad (12)$$

where $C_{Rough}$ is a user-defined coefficient with default value 1.0. The entrainment lift velocity of sediment was then computed as in [26].

$$U_{lifft,i} = \alpha_i n_s d^{0.3} \left( \theta_i - \theta_{cr,i} \right)^{1.5} \sqrt{\frac{\|g\| d_i (\rho_i - \rho_f)}{\rho_f}} \quad (13)$$

where $\alpha_i$ is the entrainment parameter, whose recommended value is 0.018 [26], and $n_s$ is the outward pointing normal to the packed bed interface. The term $U_{lifft,i}$ is then used to compute the amount of packed sediment.
sediment converted into suspension, effectively acting as a mass source of suspended sediment at the packed bed interface. After that, the sediment is transported with fluid flow, and bed-load transport is the main mode of sediment transport due to it rolling or bouncing over the surface of the packed bed of sediment. The Meyer, Peter, and Müller [28] equation for bed load transport rate is

\[ \Phi_i = \beta_{MPM,i} (\theta_i - \theta_{cr,i})^{1.5} C_{b,i} \]  

(14)

where \( \beta_{MPM,i} \) is a coefficient typically equal to 8.0, \( C_{b,i} \) is the volume fraction of species \( i \) in the bed material, and \( \Phi_i \) is the dimensionless bed-load transport rate related to the volumetric bed-load transport rate \( q_{b,i} \), by

\[ q_{b,i} = \Phi_i \left[ \frac{\rho_i - \rho_f}{\rho_f} \right] d_i^2 \]  

(15)

Equation (15) thus computes the bed-load transport rate in units of volume per bed width per time unit. The Meyer-Peter and Mueller equation, subsequent researchers have suggested, produces values ranging from 5.0 for low transport to 13.0 for very high sand transport, with 5.7 being a typically-reported value for sand and gravel [29]. In this article, the parameter selection for sediment scour after calibration of many runs were critical Shields number = 0.05, entrainment coefficient = 0.018, bed load coefficient = 12, and a maximum packing fraction of 0.64.

6. Results and Discussion

6.1 Verification of Scour Depth around Cylindrical Pier

To verify the effectiveness of the numerical model, it was applied in similar conditions to the physical model. Identical boundary conditions and other important parameters were thus applied to the numerical model, which directly affected the results and allowed calibration; after verifying the conformity of the numerical results with the laboratory results, the calibrated values of those parameters were thus used to decrease simulation errors. Figure (6) shows the results of scour depth around a cylindrical bridge pier development over 1800 sec. (steady state) simulation time at a velocity equal to 0.25 m/sec using the numerical model in Flow-3D.

![Figure (6): Scour depth (in negative values, blue colour) and deposition height (in positive values, red colour) around a cylinder pier after 1800 sec. simulation time.](image)
Figure (7) shows scour depth against time and offers a comparison between the model and experimental values. The maximum scour depth was 3.6 cm in simulation, while the experimental value was equal to 4 cm [9]. These results show good agreement, with an error ratio of only 10%.

Based on these results, the proposed scour simulation using a Flow-3D numerical model is verified as preferable methodology to accurately predict bridge scour depth and flow field development around piers.

![Figure (7). Scour depth against time around cylindrical pier.](image)

### 6.2 Effect of Flow Intensity on Scour for Different Pier Shapes.

Flow intensity is defined as the ratio of the approach mean velocity (v) to the critical mean velocity (vc). Flow velocity regarded as an important factor affecting scour depth development because it has a direct influence on the kinetic energy of running water.

Simulations were performed for different pier shape models (circular, rectangular, square, octagonal, elliptical, and lenticular) under different flow intensity ratios (0.55, 0.76, 1.0). The boundary conditions, flow depth, and other parameters were held same with the exception of flow intensity, which was varied to investigate its influence on the scour. According to the results seen in Figure (8), scour depth around each pier increased linearly with flow intensity and reached a maximum value at a flow intensity equal to 1.0.

These results also revealed that a rectangular pier with a blunt nose upstream pier shape developed maximum scour depth due to the upstream pier shape and corners, where a strong horseshoe vortex system occurs. Both square and octagonal shapes have less scour depth compared with a rectangle shape, and both circular and elliptic shapes have round nose pier shapes, which generate less scour depth at the upstream face but increase scour depth at the middle due to the increase in velocity. The lenticular pier with a sharp nose pier shape developed less scour depth as compared with other shapes due to having an upstream shape that encourages only a very weak vortex system. The lenticular shape reduced the scour depth by about 40% compared to the rectangular shape at v/vc equal to 1.
6.3 Effect of Flow Depth on Scour for Different Pier Shapes

Flow depth is another important factor affecting the development of local scour depth; it is usually referred to as flow shallowness and is examined by relating flow depth \( y \) to pier width \( b \). Flow shallowness is classified relative to the local scour at bridge piers of three classes: 1) narrow pier; 2) intermediate width pier; and 3) wide pier. The ratios chosen in this study were based on this classification, and six piers with different shapes were simulated with different flow shallowness ratios \( 0.197, 0.984, 2.953 \) to examine the variation of the maximum scour depth against flow depth. According to the results shown in Figure (9), at a shallow flow depth (wide pier) ratio \( 0.197 \), scour depth reduced to a minimum value.

For deep flows compared to pier width, for a narrow pier at ratio \( 2.953 \), scour depth increased and reached a maximum value dependent on pier diameter \( b \). According to the results, scour depth increases with increasing flow depth, though rectangular piers develop the maximum scour depth compared with other shapes.

Figure (8): Influence of flow intensity on scour depth for different pier shapes.
6.4 Effect of Pier Width on Local Scour Depth at Different Pier Shapes

The width of a pier has a direct effect on the depth of scour: the wider the pier, the deeper the scour. The ratio of pier width to channel width is thus probably a better measure of scour potential than pier width alone. In this study, six pier shapes with varied diameter cross sections were simulated, with this variation represented as a ratio b/B, (0.111, 0.15, 0.20). Figure (10) shows the relationship between the simulated maximum scour depth at 30 minutes and the pier width ratio for different flow intensity ratios. It can be

Figure (9): Scour depth versus flow depth for different pier shapes for different flow intensity.

Figure (9): Scour depth versus flow depth for different pier shapes for different flow intensity.
observed that the scour depth steadily increases with the increase in pier width ratio, reaching a maximum scour depth at a ratio of 0.20; the lenticular pier still has the minimum scour depth, however.

![Scour depth versus pier width at different pier shapes](image)

Figure (10): Scour depth versus pier width at different pier shapes

7. Development of Scour Depth Formula

The scour depth around a bridge pier is a function of various different variables including flow, fluid, sediment characteristics, and pier geometry. The following relationship thus defines scour depth as a function of its independent variables:

$$ds = f(y, v, g, B, vc, b, ks, d_{50})$$  \hspace{1cm} (16)

where $ds$ is the scour depth, $v$ is the approach flow velocity, $y$ is the approach flow depth, $g$ is acceleration due to gravity, $d_{50}$ is the mean diameter of particles, $b$ is the width of the pier, $vc$ is the mean velocity at threshold motion of sediment for the approach flow, $ks$ is the pier shape factor, and $B$ is flume width. Dimensional analysis of the variables in Equation 16 reduces it to three non-dimensional parameters.

$$\frac{ds}{b} = f\left(y, v, \frac{b}{vc}, B, ks\right)$$  \hspace{1cm} (17)

These four parameters were thus used as independent variables in the development of a new scour depth
The confidence of the suggested relationship was evaluated according to the coefficient of determination \((R^2)\), and Eureqa-Formulize Statistics software used to analyse the equation using nonlinear regression analysis. The following formula is thus suggested to predict the scour depth around bridge piers depending on the numerical results from the resulting 126 item dataset in terms of pier width \((\frac{b}{B})\), flow depth \((\frac{\nu}{\nu_c})\), flow intensity \((\frac{\nu}{\nu_c})\), and pier shape factor \(k_s\):

\[
\frac{ds}{b} = 12.5 * \left(\frac{b}{B}\right) + 0.2 * k_s * \left(\frac{\nu}{\nu_c}\right) + \frac{0.0756 * \left(\frac{\nu}{\nu_c}\right) * \left(\frac{\nu}{\nu_c}\right)}{\left(\frac{b}{B}\right)} - 0.162 * \left(\frac{\nu}{\nu_c}\right)^2 - 1.484 \quad (18)
\]

The coefficient of determination \((R^2)\) and mean square error (MSE) for this formula are 0.96 and 0.00625, respectively Figure (11).

8. Conclusion

The validation of the numerical model in Flow-3D based on a comparison of scour depth estimation between the numerical and experimental models indicates about a 10% error rate in prediction of scour depth for the numerical model. According to this validation, the proposed numerical model in Flow-3D is thus an effective tool for analysis and prediction of scour development and flow fields around bridge piers.

Some of the constraints for the use of Flow-3D are that the running time for a simulation process sometimes exceeds 168 hours for a 30-min scour simulation with several different cases, as used during this study, however.

Nevertheless, Flow-3D is an acceptable model for the simulation of different flow features (velocity, depth) for different pier shapes and widths. The obtained results suggest that scour depth rate depends on pier shape, with rectangular piers developing maximum scour depth, much higher than for other shapes, while lenticular piers have minimum scour depth compared to other shapes due to their minimum exposed area without side corners. The results indicate that the scour depth also reaches a maximum value with the increase in pier width, and that rectangular piers have a maximum value of scour depth ratio of 1.7 at a maximum pier width ratio 0.2, while other cases, particularly the lenticular pier, have scour depth of about 1.01 at pier width ratio 0.2. Overall, the results suggest that the lenticular shape can be considered as the
optimum pier shapes for reducing the maximum scour depth, producing a reduction of about 40% as compared with the rectangular shape in most circumstances.

References

[1] P. N. Cheremisinoff, N. P. Cheremisinoff, S. L. Cheng, and C. W. Bert, Civil Engineering Practice: Surveying, vol. 4. Technomic Pub. Co., 1988.
[2] R. Shepherd and J. D. Frost, “Failures in civil engineering: Structural, foundation and geoenvironmental case studies,” 1995.
[3] Reda Mahmoud Abd Elaal Ali DIAB, “Experimental Investigation on Scouring around Piers of different Shape and Alignment in Gravel,” Institut für Wasserbau und Wasserwirtschaft Technische Universität Darmstadt, 2011.
[4] J. Kattell and M. Eriksson, “Bridge scour evaluation: Screening, analysis, & countermeasures,” 1998.
[5] M. Muzzammil, T. Gangadharaiah, and A. K. Gupta, “An experimental investigation of a horseshoe vortex induced by a bridge pier,” in Proceedings of the institution of civil engineers-water management, 2004, vol. 157, no. 2, pp. 109–119.
[6] Melville and S. E. Coleman, “Bridge scour.” 2000. Water Resources Publications, LLC, Colorado, U.S.A.
[7] W. H. Hassan, M. H. Jassem, and S. S. Mohammed, “A GA-HP Model for the Optimal Design of Sewer Networks,” Water Resour. Manag., vol. 32, no. 3, pp. 865–879, 2018.
[8] E. M. Laursen and A. Toch, “Scour Around Bridge Piers And Abutments by,” no. 4, 1956.
[9] B. W. Melville, “Local scour at bridge sites,” PhD Thesis, vol. 1994, p. 227, 1975.
[10] C. Fael, R. Lança, and A. Cardoso, “Effect of pier shape and pier alignment on the equilibrium scour depth at single piers,” Int. J. Sediment Res., no. 1992, pp. 1–7, 2016.
[11] Obeid Z. H., “3D numerical simulation of local scouring and velocity distributions around bridge piers with different shapes”, 2016.A Peer Rev. Int. J. Asian Acad. Res. Assoc., vol. 20, no. 16, p. 2801.
[12] Breusers, Nicollet and Shen, “Local scour around cylindrical piers”, 1977. J. Hydraul. Res.
[13] R. Ettema, “Scour at bridge piers,” 1980. PhD Thesis, Auckland University, Auckland, New Zealand.
[14] W. M. Chiew, “Local scour at bridge piers, School of Engineering. New Zealand.” PhD thesis, Department of Civil Engineering, report, 1984.
[15] A. Kumar and U. C. Kothyari, “Three-dimensional flow characteristics within the scour hole around circular uniform and compound piers,” J. Hydraul. Eng., vol. 138, no. 5, pp. 420–429, 2011.
[16] A. R. Deshmukh and R. V Raikar, “A CLEAR WATER SCOUR AROUND A CIRCULAR BRIDGE PIER UNDER STEADY FLOW FOR DIFFERENT OPENING RATIOS,” pp. 158–162, 2014.
[17] M. Alemi and R. Maia, “Numerical Simulation of the Flow and Local Scour Process around Single and Complex Bridge Piers,” Int. J. Civ. Eng., vol. 16, no. 5, pp. 475–487, 2018.

[18] A. Kassem, T. M. Salaheldin, J. Imran, and M. H. Chaudhry, “Numerical Modeling of Scour in Cohesive Soils Around Artificial Rock Island of Cooper River Bridge,” no. 03, pp. 45–50, 1851.

[19] H. Zhang, H. Nakagawa, K. Kawaike, and Y. Baba, “Experiment and simulation of turbulent flow in local scour around a spur dyke,” Int. J. Sediment Res., vol. 24, no. 1, pp. 33–45, 2009.

[20] J. Vasquez and B. Walsh, “CFD simulation of local scour in complex piers under tidal flow,” Proc. thirty-third IAHR Congr. Water Eng. a Sustain. Environ., no. 604, pp. 913–920, 2009.

[21] W. Huang, Q. Yang, and H. Xiao, “CFD modeling of scale effects on turbulence flow and scour around bridge piers,” Comput. Fluids, vol. 38, no. 5, pp. 1050–1058, May 2009.

[22] F. S. I. Flow-3D v.9.2, “No Title,” 2008.

[23] D. Drikakis, “Advances in turbulent flow computations using high-resolution methods,” Prog. Aerosp. Sci., vol. 39, no. 6–7, pp. 405–424, 2003.

[24] Yakhot and Orszag, “Renormalization Group Analysis of Turbulence. I. Basic Theory,” vol. 1, no. 1, pp. 3–51, 1986.

[25] FLOW-3D manual, “FLOW-3D user manual, version 11.” Flow Science Santa Fe, NM, 2014.

[26] D. R. Mastbergen and J. H. Van Den Berg, “Breaching in fine sands and the generation of sustained turbidity currents in submarine canyons,” Sedimentology, vol. 50, no. 4, pp. 625–637, 2003.

[27] R. Soulsby, Dynamics of marine sands: a manual for practical applications. Thomas Telford, 1997.

[28] E. Meyer-Peter and R. Müller, “Formulas for bed-load transport,” in IAHSR 2nd meeting, Stockholm, appendix 2, 1948.

[29] G. Wei, J. Brethour, M. Grünzner, and J. Burnham, “Sedimentation Scour Model,” no. August, 2014.