Three-dimensional numerical simulation of local scour around circular bridge pier using Flow-3D software

Halah Kais Jalal¹, Waqed H. Hassan²

¹ Graduate student, Civil Engineering Department, University of Kerbala, Kerbala, Iraq.
² Professor, University of Kerbala, Kerbala, Iraq.
E-mail: halah.q@s.uokerbala.edu.iq, Waaqidh@uokerbala.edu.iq

Abstract. The problem of local scouring around circular bridge pier has been studied numerically by Computational Fluid Dynamics (CFD) using Flow-3D model to represent the evolution of local scour and the maximum depth of the scour hole which is important in the bridge pier design. The aim of this study is to verify the ability of the numerical simulation model Flow-3D to accurately simulate and predict the scour depth around the bridge pier. This verification is conducted by comparison the numerical results with Melville laboratory experimental model. The maximum scour depth around the circular pier obtained from numerical results after 30 min is 3.6 cm, while the scouring depth obtained from Melville model is 4 cm. According to these results, the error rate ratio between the numerical and experimental models is close to 10%. The results showed a good validation with experimental results. Finally, the proposed Flow-3D model considered an effective tool in predicting and simulating the scour depth around bridge pier and considered an economic method to predict potential results.

Keywords: Local scour, Flow-3D, CFD, Verification

1. Background

Scouring can be defined as erosion action of flowing water, which removes and erodes the bed material from the near bridge piers and abutments [1]. Scouring around bridges piers considered as one of the main reasons that cause of the bridge failed alongside collisions and overloading, leading to massive loss of life and economic impacts, as explained [2], the local scour prediction especially the maximum scour depth is, therefore, essential for bridge design, maintenance and evaluation. Many researchers from all over the world have studied the problem of scouring extensively from different points of view and under different conditions. The scour made at a bridge site generally contained three different types, which are; general scour, contraction scour and local scour [3], of the three types of scour, local scouring considered as the important part of this study, because it plays the most important part in the risks associated with scouring bridges.

Many prior studies have aimed on the techniques and methodologies for analysing local scouring of bridge using empirical tests [4], [5], [6], [7],[8],[9],[10], [11]. Most of these empirical tests of scour are often performed on large and important bridges, due they are expensive and labour-intensive. However, for the most popular highway bridges, empirical tests do not apply, although the scour is frequently occurring in
such regular bridges, although some studies have examined an analytical solution to bridge scour for economic and practical purposes.

In the last years, the three-dimensional simulation method of scouring has become more prevalence due to the increasing capabilities of computers and software that allow the wider use of computational fluid dynamics (CFD) to determine fluid flow behaviour in industrial and environmental applications. The using of CFD software such as FLUENT, CFX, and PHOENIX are in many ways similar to setting up an experiment, so the original concept of this numerical simulation is that instead of design and construction a physical model using expansive appurtenance instrument like velocimeter. The complicated models that are difficult to model in laboratory conditions can be simply modeled by using numerical simulation. It is widely accepted that a good numerical model can certainly complement model tests and can help design engineers identify the most important cases in which model tests may be conducted. This is an attractive idea to solve complicated problems and large model studies for which there is no need for extra workers or existing large setup for determining the actual results. 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.

In the present study, the commercial code Flow-3D is used for modelling the scour depth around the bridges pier. The Flow-3D model is a CFD package with special units for hydraulic engineering applications; numerical techniques are used to solve fluid motion equations for transient and three-dimensional solutions to obtain multi-scale multi-physics flow problems. The combination of physical and numerical options allows users to apply Flow-3D to a wide range of fluid flows and heat transfer phenomena and is widely used to solve various hydraulic problems [12].

The numerical simulation of scour by Flow-3D has been proposed by many researchers, for example [13] used Flow-3D to simulate the flow that occurs at the base of a circular bridge pier within a scour hole, while [14] used a numerical model to simulate of local scouring in complex bridge piers under tidal flow,[15] used Flow-3D to study the effect of pier shapes on local scour depth under different conditions and [16] used CFD code to simulate the 3D flow and local scouring around bridges pier of different shapes. All these studies hypothesized that water running under clear-water conditions predominantly leads to most interactions between flow and river bed.

In this paper, the validation of the numerical simulation by the laboratory models of [4] to compare results of the laboratory experiments of local scour around the bridge with numerical simulation results of CFD code Flow-3D for the verification purpose. The main purpose of this verification is to test the effectiveness of the numerical model Flow-3D in predicting the scour depth around the bridges pier.

2. Numerical and Experimental Model

To validate the accuracy of FLOW-3D numerical model to simulate the flow and prediction of local scour depth and velocity profile around the bridge pier, it is results were compared with laboratory experiment results from the Melville experimental [4]. The numerical simulation condition is about the same as that conducted in Melville experimental. For the experimental model, the flume used was 19 m long, 45.6 cm wide and 44 cm deep with a circular bridge pier have a diameter (b) of 5.08 cm. The sand was used as bed material with a median grain size d50 of 0.385 mm, height 12.7 cm and density of 2650 kg/m3 with the angle of repose of the sand was 32°. The approach flow velocity was during the test set equal to 0.25 m/s with a water depth of 15 cm. The running time was set equal to 30 min when the required scour holes reaching to an intermediate condition. Figure (1) shows the sketch of the experiment model.
For the numerical model, an inlet located at a distance of 6b, (where b: is the pier diameter) upstream of the pier with a diameter equal to 5.08 cm. The outlet located at a distance of 14b downstream of the pier. One sold component was settled at the inlet of the flume, to prepare an inflow bottom at the top edge of the sediment at an elevation of 12.7 cm in order to prevent against upward movement of sediments in the beginning of simulations. Packed sediment component placed in the flume at a depth equal to 12.7 cm and the water depth was 15 cm above the sediment level. The other parameters used in this numerical simulation were set equal to those used in Melville model. The geometric representation of the numerical model is shown in Figure (2).

3. Numerical Method

FLOW-3D CFD package was made by Flow Science Inc. and it uses two methods the VOF and FAVOR to determine the location of the free surface and the location of the obstacles respectively. FLOW-3D uses specially developed numerical techniques to solve the equations of motion of liquids to obtain transient, three-dimensional transient solutions to multi-scale flow and physics problems [12]. FLOW-3D used to simulate the scour process around the bridge piers and has a powerful capacity to investigate the behavior of liquids and gases specializing in the solution of transient, free-surface problems and sediment transport. A non-hydrostatic finite difference model is used to solve the 3D equations of Navier-Stokes.
3.1 Governing equations
The equations governing the motion of a viscous fluid namely the continuity equation and the momentum conservation equations. These equations are known as Navier-Stokes equations. The Navier-Stokes equations for uncompressed viscous liquids take the following form:

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

Continuity equation

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{2}
\]

In the Reynolds-averaged Navier-Stokes method, the average Navier-Stokes equations are calculated over a period of time or across a set of equivalent flows. The aim of this approach is to obtain the average effect of turbulent quantities [17]. The Reynolds-averaged Navier-Stokes (RANS) for non-compressive Newtonian fluid equations are:

\[
\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} - \frac{\partial \langle \bar{u}_i \bar{u}_j \rangle}{\partial x_j} \tag{3}
\]

where:

\[
\sigma_{ij} = 2 \nu s_{ij} \tag{4}
\]

\[
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{5}
\]

\[
\bar{u}_i \bar{u}_j = \nu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k \tag{6}
\]

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

3.2 Turbulence model
There are six different types of turbulence models contained in Flow-3D are: 1) the Prandtl mixing length model, 2) the one-equation, 3) the two-equation \( k-\epsilon \), 4) RNG, 5) \( k-\omega \) models, and 6) a large eddy simulation LES model. Work in [18] introduced the RNG-based \( k-\epsilon \) model; the main feature of this model is the numerical constants in the \( k-\epsilon \) model, which are obtained directly from the renormalization group theory, and correspond well to the standard \( k-\epsilon \) model. However, a constant equation has been empirically shown in the standard \( k-\epsilon \) model which must be explicitly derived in the RNG model. Generally, however, the RNG model applies more broadly than the standard \( k-\epsilon \) model. The RNG model used in this paper for these reasons: 1) this model is well suited for the modeling of turbulent flow over bridge pier, 2) this model is considered as the most accurate and strong model available in the software for scouring simulations, 3) RNG model may perform better for scour simulations due their
suitability in cases where a large amount of turbulence is created which, in this case, is caused by the flow of the fluid through the control structure [12]. The governing equations are:

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

(7)

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

(8)

where \( k \) is the Reynolds-averaged kinetic energy, \( \varepsilon \) 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 \( \sigma_e \) are model coefficients, and have values of 0.085, 1.42, 1.39, 0.7179, and 0.7179 respectively.

### 3.3 Sediment scour model

The sediment scour model has assumed a type of incoherent sediment that has different characteristics including grain size, mass density, critical shear stress, angle of repose, and parameters for entrainment and transport. There is no way to calculate the flow dynamics for each individual grain of sediment, and therefore an experimental model should be used. The model used was based on Mastbergen and Van den Berg [19] with the addition of the Soulsby-Whitehouse equation [20] to forecast the critical Shields parameter. The first step to calculate the critical Shields parameter is to calculate the dimensionless parameter \( d_{*i} \):

\[
d_{*i} = d_i \left[ \frac{\rho_f (\rho_i - \rho_f)}{\mu^2_f} \right]^{\frac{1}{3}} 
\]

(9)

where \( \rho_i \) is the density of the sediment types \( i \), \( \rho_f \) is the density of the fluid, \( d_i \) is the diameter, \( \mu_f \) is the dynamic viscosity of fluid and \( \| g \| \) is the amount of acceleration of gravity \( g \). Thus, the dimensionless critical Shields parameter was calculated using the Soulsby-Whitehouse equation [20].

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

(10)

The local Shields parameter can be calculated depended on the local bed shear stress, \( \tau \):

\[
\theta_l = \frac{\tau}{\| g \| d_i (\rho_i - \rho_f)} 
\]

(11)

where \( \tau \) is computed by using the law of the wall and the quadratic law of the bottom shear stress for three-dimensional turbulent flow and shallow water turbulent flow, respectively, taking into account some
roughness of the bed surface. Thus it was assumed that the Nikuradse roughness of the bed surface, $K_s$, was proportional to the diameter of the local median grain in packed sediment $d_{50 \text{ packed}}$.

$$K_s = C_{\text{rough}} d_{50 \text{ packed}}$$

(12)

where $C_{\text{rough}}$ is a user-defined coefficient with default value 1.0. The lifting speed of the sediment was then calculated as in [19].

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

(13)

where $\alpha_i$ is the entrainment parameter, whose recommended value is 0.018 [19], and the outward $n_s$ is referring naturally to the packed bed interface. The term $U_{\text{lift},i}$ is then used to calculate the amount of packed sediment converted into suspension, and effectively acts as a mass source of sediment suspended in the packed bed interface. After that, the sediment is transported with fluid flow, and bed-load transport is the main method of the sediment transport since it rolls or bounces on the surface of the packed bed of sediment. The Meyer, Peter, and Müller [21] equation for the bed load transport rate are:

$$\Phi_i = \beta_{\text{MPM},i} \left( \theta_i - \theta_{cr,i} \right) 1.5 C_{b,i}$$

(14)

where $\beta_{\text{MPM},i}$ coefficient is usually 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

$$q_{b,i} = \Phi_i \left( \|g\| \frac{(\rho_i - \rho_f)}{\rho_f} \right)^{1.2} d_i^{1.7}$$

(15)

Equation (15) calculates the bed-load transport rate in units of volume per bed width per unit of time. The Meyer-Peter and Mueller equation, subsequent researchers have suggested, produces values ranging from 5.0 for low to 13.0 for very high sand transport, with 5.7 being a typically-reported value for sand and gravel [22]. In this article, the parameter selection for sediment scour after calibration of many runs were critical Shields number = 0.05, entrainment coefficient = 0.018, bedload coefficient = 12, and a maximum packing fraction of 0.64.

3.4 Meshing

In FLOW-3D, the mesh is the most important matter for an accurate solution. The mesh considering as one of the affecting factors on the simulation process and can alter the time of operation depending on the number of cells. Various types of mesh accuracy have been examined, providing the best precision/computation time results. Therefore, different cell sizes are selected as (30, 25, 20, 15, 10, 5 and 1) mm to identify the optimum cell size that satisfies the phenomenon conditions, so that number of test that carried out as shown in Figure (3) to determine the best cell size that the depth of scour doesn't change significantly.
Figure 3. Effect of Cell Size on Scour Depth

According to Figure (3) when the cell size reaches 5 mm, the scour depth becomes constant and shows independence from cell size. The larger cell size 30 mm caused the problem that cell size doesn't give an accurate result. The cell size of 5 mm considered the optimum cell size depending on accuracy of results and high clearness scour depth around pier. For better observation of scour development around a pier two mesh planes have been identified with finer resolutions for both sides of the pier in x and y directions. Minimum cell size near the pier is about 5 mm and the largest cell size is limited as 10 mm to reduce in computation time. The total numbers of cells are 252,000 cells are generated for the working section, which is shown in Figure (4).

Figure 4. Meshing Plane Structure Around a Circular Pier

3.5 Boundary conditions

The important thing of the numerical simulation is the determination of the representative boundary conditions for the hydraulic analysis. The boundary conditions were applied to the numerical model and some important parameters that directly affect the results were calibrated and after verifying the conformity percentage of numerical results with laboratory results, the calibrated values of parameters were used in the numerical simulation. The boundary conditions used in this study shown in Figure (5). The type of boundary set at the inlet was specific velocity (V), the velocity of water was 0.25 m/sec and the elevation of water was 0.15 m. At the downstream, the boundary was set as outflow (O), while symmetric boundaries (S) were set at the top and right sides and wall boundary (W) at the bottom of the numerical model. These boundaries conditions of the numerical model should be corresponded to the physical conditions of the problem.
4. Verification of Scour Depth Prediction Results

To test the effectiveness of the numerical model, this numerical model was simulated in similar conditions to the physical model. Figures (6) show the three-dimensional output of the numerical model for the maximum scour depth prediction over 30 min simulation times.

Figure (7) shows the scour depth results around a circular bridge pier for the physical model [4], while Figure (8) presents the scour depth of the numerical model. These results represented by contours of the evolution of scour depth around the circular bridge pier.
Figure 7. Contour Lines Represented the Depth of Scour Around Circular Bridge Pier for Melville Model

Figure 8. Contour Lines Represented the Depth of Scour Around the bridge Pier for the Numerical model.

Figure (9) presents the development of scour depth with time and compared final results with the experimental value. The maximum scours depth obtained from the numerical model is 3.6 cm, while the maximum depth of scouring for the experimental model is 4 cm. The results illustrated a good corresponding with experimental results with an error ratio close to 10%.
The comparisons of the flow velocity distribution on the $y = 0$ slice between the experiment measurements \cite{4} and the numerical simulations at 30 min simulation time are illustrating in Figures (10) and (11). The experiment-like velocity is normalized by the average flow velocity approach that was 0.25 m/sec. In the experiment and numerical simulation, it can be noticed that a strong downward flow developed along the pier face produced fairly large speed components near the bed, resulting in the deformation of the profiles in the circular pier. The flow separates at the nose of the scour hole and reinserts it at the front of the pier, forming a horseshoe vortex, clearly recognizable from both the simulation and the experiment. From these results, the proposed scour simulation by using Flow-3D numerical model is verified as the preferred methodology for accurate prediction the depth of the bridge scouring and developing the flow field around the piers.

**Figure 9.** Scour depth against time around cylindrical pier.

**Figure 10.** Contour map of flow velocity around a pier at 30 min resulted by Melville \cite{4}.
5. Conclusion

This study aimed to verify the effectiveness of this numerical simulation in predicting the development of the scour depth at the bridge pier. The verification is concluding by comparing the numerical results of Flow-3D with Melville experimental model after 30 min of scour depth formulation. The comparison of the result indicates that the rate of error equal to 10% for the maximum depth of scour hole, this observation shows a good validation between the numerical and experimental work, so the numerical simulation successfully reproduces the scour depth. According to these results, the proposed numerical model Flow-3D considered an effective tool for the simulation and predicting of scour depth and flow fields around the bridges pier.

References

[1] Breusers Niclot and Shen 1977 Local scour around cylindrical piers Journal of Hydraulic Research, IAHR,15 (3): 211-252.
[2] Shepherd R. and Frost J D 1995 Failures in civil engineering: Structural, foundation and geoenvironmental case studies Journal of Hydraulic Engineering, Publisher ASCE.
[3] Cheremisinoff N P and Cheng S L 1987 Hydraulic mechanics 2 Civil Engineering Practice, Technomic Published Company, Lancaster, Pennsylvania, U.S.A. 780 p.
[4] Melville B W 1975 Local scour at bridge sites University of Auckland, New Zealand, phd. Thesis, Dept. of Civil eng., Rep. No. 117.
[5] Abdul-Nour M 1990 Scouring depth around multiple M.Sc. Thesis , Department of Irrigation and Drainage , University of Baghdad.
[6] Hosny M M 1995 Experimental study of local scour around circular bridge piers in cohesive soils Colorado State University, Fort Collins.
[7] Ansari S A Kothyari U C and Ranga Raju K G 2002 Influence of cohesion on scour around bridge piers Journal of Hydraulic Research, IAHR, pp. 40(6): 717-729.
[8] Khsaf S I 2010 A study of scour around Al-Kufa bridge piers Kufa Engineering.
Journal. Vol. 1 No. 1, 2010, University of Kufa / College Engineering / Civil Department.

[9] Hassan W H Jassem M H and Mohammed S S 2018 A GA-HP Model for the Optimal Design of Sewer Networks Water Resour. Manag., vol. 32, no. 3, pp. 865–879.

[10] Hassan W H 2017 Application of a genetic algorithm for the optimization of a cutoff wall under hydraulic structures J. Appl. Water Eng. Res., vol. 5, no. 1, pp. 22–30, Jan.

[11] Ataie-Ashtiani B 2013 Flow field around single and tandem piers Flow Turbulence and Combustion Journal of Hydraulic Engineering, volume 9429.

[12] Flow -3D manual 2014 Flow-3D user manual version 11, Flow Science Santa Fe, NM.

[13] Richardson J E and Panchang V G 1998 Three-Dimensional Simulation of Scour Inducing Flow at Bridge Piers Journal of Hydraulic Engineering, 124(5), pp. 530–540. doi: 10.1061/(asce)0733-9429(1998)124:5(530).

[14] Vasquez J and Walsh B 2009 CFD simulation of local scour in complex piers under tidal flow Proceedings of the thirty-third IAHR Congress: Water Engineering for a Sustainable Environment, (604), pp. 913–920.

[15] W H H and Halak Jalal 2019 Effect of Bridge Pier Shape on Depth of Scour Iop, Conf. Ser., (under publication).

[16] Obeid Z H 2016 3D numerical simulation of local scouring and velocity distributions around bridge piers with different shapes A Peer Reviewed International Journal of Asian Academic Research Associates, 20(16), p. 2801. doi: 10.1186/1757-7241-20-67.

[17] Drikakis D 2003 Advances in turbulent flow computations using high-resolution methods Progress in Aerospace Sciences, 39(6–7), pp. 405–424. doi: 10.1016/S0376-0421(03)00075-7.

[18] Yakhot and Orszag 1986 Renormalization Group Analysis of Turbulence, Basic Theory Journal of Scientific Computing, pp. 3–51. 1, pp. 3–51.

[19] Mastbergen D R and Van Den Berg J H 2003 Breaching in fine sands and the generation of sustained turbidity currents in submarine canyons Sedimentology, 50(4), pp. 625–637. doi: 10.1046/j.1365-3091.2003.00554.x.

[20] Soulsby R L and Whitehouse R J S W 1997 Threshold of sediment motion in Coastal Environments Proc. Combined Australian Coastal Engineering and Port Conference, EA, pp. 149–154.

[21] Meyer-Peter E and Müller R 1948 Formulas for bed-load transport Proceedings of the 2nd Meeting of the International Association for Hydraulic Structures Research, 39–64.

[22] Wei G Brethour J Grünzner M and Burnham J 2014 Sedimentation Scour Model Flow Science Report 03-14.