Numerical study of local scour around bridge piers

H Omara¹ and A Tawfik¹,²

¹ Egypt - Japan University of Science and Technology (E-Just), Environmental Engineering Department, P.O. Box 179 – New Borg El Arab City – Postal Code 21934 - Alexandria – Egypt Hewida.omara@ejust.edu.eg
² National Research Centre, Water Pollution Research Department, 12622, Dokki, Giza, Egypt E-mail: hewida.omara@ejust.edu.eg; ahmed.tawfik@ejust.edu.eg

Abstract. Numerical models have more flexibility avoiding the critical limitation in physical ones. The predictive capability of the hydro-morphological model around bridge piers in clear-water conditions have been investigated using mathematical modelling flow 3D 11.2 as there is a lack in using that model. The objective of the study to investigate the effect of cross section of the pier on the predictive capability of the model. Computations has been performed and the results are compared with several sets of experimental data available in the literature. The Van Rijn sediment transport model and (RNG) k-ε model are used. However, this study strongly demonstrates that a 3D hydro-morphological model can effectively predict the scour around piers with different cross section, there is significantly under predicts for the scour depth at the nose of the pier with different pier shapes. It is found that for the circular pier the capability of the model depending on the flow properties not only the geometry of the pier.

1. Introduction
Three-dimensional flows around vertical obstructions in natural rivers or streams are inherently complex because of generation of flow separation and turbulence structures in a wide range of scales [1]. Various analytical, numerical, experimental and data processing methods have been used to estimate the local pier scour. several three-dimensional numerical studies around a single vertical circular cylinder have been conducted to study the scour process and estimate the scour depth. A fully 3D hydrodynamic model flow occurring at the base of a cylindrical bridge pier within a scour hole was used by [2,3] investigated mean flow and turbulence kinetic energy distribution inside the equilibrium scour hole around cylindrical piers. Computational studies by focusing on flow patterns around piers on a flat rigid bed have been reported by [4–6]. [5] used FLUENT to predict the 3-d flow field around a circular cylinder for rigid and erodible beds which gives satisfactory agreement in the bed shear stresses with those calculated from the experimental velocities near the bed. Also, [7] used different turbulence models and compared with several sets of experimental data of [8], Dargahi (1987) and [9] to investigate the flow around cylindrical pier. they declared that the standard and the RNG k-ε models can be used to calculate the flow field around the pier although they slightly overestimate the velocity near the bed. [10] applied the same DES approach to study the dynamics of the flow past the cylindrical pier mounted on a flat rigid bed studied experimentally by Dargahi (1987) and obtained results in good agreement with the experimental observations. [11] investigate the effect of pier bluntness on the dynamics of scour and the predictive capabilities of the model for three bridge piers (cylindrical, square, and diamond cross-sectional shape) with movable-bed channels developed hydro-morpho dynamic CURVIB method. They observed discrepancies between experiments and
simulations. Modelling of the free-surface flow around a pier in a fixed scour hole was assessed by [12] to determine shear stress distributions at the channel-bed using mesh-based numerical models. The bed shear stress from FEM agrees well with the measurements. Some discrepancies exist for locations just before water enters the scour hole. Prediction of local scour around a single square pier using numerical FSUM and physical models for suspended sediment concentration and bed morphology was investigated by [13]. They found that the model predicts a bigger scour depth in comparison to the measured scour depth in the physical model. Both numerical model and experiment results show that the maximum scour depth occurs at two front edges of the pier. [14] carried out the 3D numerical analysis of pier scour using the level set method. They simulated the scour evolution and free surface under the currents and wave. Results showed good agreement with experimental observation made by [15]. A numerical scour model was used by [16] to get approximation approach of the equilibrium scour depth around a mono-pile by FLUENT.

A few CFD models have been developed for computing flow patterns and bed-profile changes around hydraulic structures, e.g., SSIIM, Fluent and Flow-3D models. There is a lack for using flow 3d to simulate scour around bridges piers. The objective of the present study is to investigate the capabilities in simulating scour around circular piers for piers with different cross section using 3-D computational fluid dynamics (CFD) solver Flow 3D 11.2.

2. Numerical Model

FLOW 3D can simulate bed and suspended load rate for steady or unsteady flow conditions. It is developed by Flow Science Inc., uses a non-hydrostatic finite difference model to solve the 3D Navier-Stokes equations, and has a powerful capability to deal with free surface flow and sediment transport issues [17]. The model can simulate the deposition and entrainment of sand, silt and other non-cohesive sediment, furthermore, it enables specifying multiple sediment species, and includes a bed load transport model, a nonlinear drift-flux model, and empirical equations to predict the entrainment and erosion of sediment. The continuity and momentum Navier-Stokes equations are applied:

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

\[
\frac{\partial u_i}{\partial x_i} = -\frac{1}{\rho} \frac{\partial}{\partial x_i} (p\delta_{ij} + \rho u_i u_j)
\]

(1)

(2)

Where, \( P \) is the dynamic pressure, \( \rho \) the flow density, \( u_i \) the i component of the local time-averaged flow velocity, \( \delta \) the Kronecker Delta function and \( \rho u_i u_j \) is the turbulence stresses. The numerical solution to complement, a turbulence model and renormalization-group (RNG) k-\( \varepsilon \) model has been used. Sediment scour requires accurate estimates of the near-wall shear stresses, so a good turbulence model should be selected for turbulent flows. The RNG model is known to describe more accurately low intensity turbulence flows and flows having strong shear regions. For computational domain, it was found that the upstream inlet should place at sufficient distance from the pier to ensure that the flow becomes fully developed 40D. Sufficient distance is provided between the pier and the downstream outflow to ensure that the flow returned to the undisturbed pattern approximately 12 times the pier diameter downstream of the pier [18]. So, the domain length used is, \( l=52D \), with fixed bed at the inlet to prevent scour to prevent scour at the inlet, Figure 1. The pile is located at \( (x=0, y=0 \) and \( z= -0.2) \) of the domain. The bed surface is at \( z=0 \).

![Figure 1. boundary condition of present study](image)
Finer mesh is required near solid boundaries including the pier and packed sediment bed to resolve the flow details near the solid boundaries. A considerable effort has been put into optimizing the computational mesh for the best grid and time convergence. The size of the cells adjacent to solid boundaries is chosen to satisfy the limits of the wall unit distance $11.225 < Z^+ < 30$ based on law of wall to avoid calculation in the laminar sub-layer as viscosity dampens turbulence near a wall and the value of $Z^+$ should be smaller than 30 to ensure that the law of the wall is applicable. The wall unit distance $Z^+$ is defined by the following equation:

$$Z^+ = \frac{\rho u_p}{\mu}$$

where $\rho$ fluid density; $u$ shear velocity; $z_p$ distance from Point $p$ to the wall; and $\mu$ dynamic viscosity of the fluid. Finer grid is also provided near the air-water interface to capture the small variation of the free surface [7]. It was used a mesh measuring 0.003 m near the pier and sediment bed surface and gradually increased to 0.025 m in the rest of the domain. The law-of-the-wall for the mean velocity modified for roughness is expressed as follows:

$$u_p = u \left[ \frac{1}{k} \ln \left( \frac{\rho c_k}{\mu} k^2 z_p \right) \right] - \Delta B$$

where $u_p$ mean flow velocity at point $p$; $k$ von Karman’s constant (0.418); $u$ shear velocity related to the bottom shear stress $\tau$ ($\tau = \rho u^2$); $\rho$ fluid density; $c_k$ constant; $k_p$ turbulent kinetic energy at point $p$; and $\Delta B$ roughness function. Free surface can be included in the one-fluid incompressible mode. In FLOW-3D, a proper definition of the boundary conditions is important for an accurate capture of the free-surface dynamics. The volume of fluid (VOF) method is employed. The top surface is defined by stagnation pressure with the atmospheric pressure value. At the bottom, there is an interface layer with thickness 0.2 m. At the exit of the channel, the outflow condition is used so the water level at the outlet is calculated from inside the domain, Figure 2. At a stagnation pressure source in the inlet, fluid is assumed to enter the domain at constant velocity. The roughness height of the bottom is (2.5 D50).

### 3. Sediment Scour Model

This model assumes non-cohesive sediment species and estimates the motion of sediment by predicting the erosion, advection and deposition process. Soulsby-Whitehouse equation [Sou97] used to predict the critical shields parameter $\theta_{cr,i}$:

$$\theta_{cr,i} = \frac{0.3}{1 + 1.2d_{i,i}} + 0.055 \left[ 1 - \exp \left( -0.02d_{i,i} \right) \right] \text{ where } d_{i,i} = d_i \left[ \frac{\rho_f (\rho_i - \rho_f)}{\rho_f} \right]^{\frac{1}{2}}$$

$\rho_i$ 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. The local shields parameter is:

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

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 consideration of bed surface roughness. Nikuradse roughness of the bed surface $k_s$ is:

$$k_s = C \text{rough} D_{50}, \text{ packed}$$

where $C$ is a roughness coefficient and $k_s$ is the physical roughness height taken here and equal to the 2.5. The entrainment coefficient can be used to scale scour rates or fit experimental data where it found that 0.005 give good result. The volumetric sediment transport rate per width of bed Van Rijn [vanRijn84] which has been described as follows is used,

$$\Phi_i = \beta_{VR,i} d_i^{-0.3} \left( \frac{\theta_i}{\theta_{cr,i}} - 1 \right)^{2.1} c_{bi,i}$$

Where $\beta_{VR,i}$ is coefficients typically equal to 0.053, $c_{bi,i}$ is the volume fraction of species $i$ in the bed material, $\Phi_i$ is the dimensionless bed-load transport rate which is taken 0.05, and is related to the volumetric bed-load transport rate, $q_{bi,i}$, by:

$$q_{bi,i} = \Phi_i \left( \frac{\rho_i - \rho_f}{\rho_f} \right) d_i^{1/2}$$
4. Case Study
The laboratory experiments reported by [9] which also used by [7,19] to validate their numerical model are simulated in the present study (flatbed to calibrate the model and equilibrium scour bed) for cylindrical pier, [20,21] for rectangular cross section pier and [22] for square cross section pier. All required information for physical experiment are presented in Table 1.

Table 1. Relevant Information on Various Experimental Conditions Modeled in the Present Study

| Experiment data               | Ahmed & Rajaratna m, (1998) | Vaghefi & Ghodsian, (2015) | Abdul-Hassan & Hadi, (2016)[20] | Fael et al., (2016)[21] | Ghiassi & Abbasnia, (2013)[22] | Khosronejad et al. (2012)[11] |
|------------------------------|-----------------------------|-----------------------------|---------------------------------|-------------------------|--------------------------------|------------------------------|
| Pier cross section           | cylindrical                  | Rectangular, square with around nose | Rectangular square nose | Rectangular square nose | square                         |                              |
| Flume dimension              | 20/1.22 m                   | 12/0.5m                     | 14/2m                          | 13/0.6m                 | 10/0.45 m                      |                              |
| Y (m)                        | 0.182                       | 0.18                        | 0.12                           | 0.2                     | 0.088                          | 0.139                        |
| U(m/sec)                     | 0.2927                      | 0.32                        | 0.18                           | 0.3                     | 0.15                           | 0.22                         |
| D(m)/pier length             | 0.089                       | 0.05                        | 0.045/0.2                      | 0.05/0.2                | 0.05                           | 0.165                        |
| D50 (mm)                     | 1.8                         | 0.6                         | 0.71                           | 0.86                    | 0.53                           | 0.85                         |
| Bed slope %So                | 0.02                        | 0.002                       | 0                               | 0                       | 0                              | 0                            |
| ds Scour depth (m)           | 0.040                       | 0.08                        | 0.038                          | 0.014                   | 0.06                           |                              |
| U* /U*c                      | 0.62                        | 0.86                        | 0.56                           | 0.96                    | 0.6                            | 0.73                         |
| Fr=U/√gh                     | 0.21                        | 0.24                        | 0.17                           | 0.22                    | 0.16                           | 0.13                         |
| R=uY/v (×103)                | 53.2                        | 57.6                        | 14.7                           | 15.5                    | 13                             | 30                           |

Note: Fr is Froud number; and R is Reynolds number.

5. Results and Discussion
Numerical simulations for experiment data were carried out in order to validate the applicability of the flow 3D model for evolutions of flow and bed elevation around a cylindrical bridge pier as shown in Figure 2. Moreover, the effect of different pier shapes on the predictive capabilities of the model was studied. The erosion process can be impeded by particles that are large relative to the size of the groove that forms at the base of the scour hole in front of the pier. The groove has been identified as the principle erosion zone for bridge pier scour [8,24,25] and forms under the action of the downflow that occurs in front of the pier.

Figure 2. The scour hole shape using flow 3D
Figure 3 shows scour contours at (t=30 min) observed by Ahmed and Rajaratnam 1998 and simulated using the computational fluid dynamics (CFD) versus the model results. The computed scour contours by Fael et al. [26] used coupling of a stochastic model for sediment pickup and deposition using a momentum equation of sediment particles in order to account the temporal change in bed topography for the effect of nonequilibrium sediment transport. It is notice that the maximum scour depth is 0.040,
0.041, 0.0397 m for the experiment, [26]) and present model. From that, flow 3d gives the most agreement result with the experiment (0.040m) however, there is some disagreement observed for the scour at the area close to the pier nose; while the maximum scour depth locates at pier sides. [26] mention that this was because a double-sided sharp slope immediately close to pier nose in numerical results, which may collapse in local avalanches of sediment into the erosion zone under the action of turbulent flow in the test. Hence, the presented maximum scour depth agrees well with the test. However, the scour hole shape and location between the numerical simulation and the test are quite different the model gives scour depth with sufficient accuracy. Comparisons between the bed elevation simulated using flow 3D and the experiment of [27] at the plane of symmetry are shown in figure 4. It gives better result for the scour shape. It confirms that for cylindrical pier the model gives under estimation to the scour at the nose however it gives the maximum scour at the pier sides with good agreement with the test. Also, [11] with hydrodynamic model declared that the numerical model they used gives overpredicted the growth rate of scour and failed to capture the region of deep scour at the pier nose For the cylindrical and square shapes.

Figure 3. Scour hole contours at (t=30 min) (cm): (a) observed by [9]; (b) simulated by [26] (m); (c) model results(m).

Figure 4. Bed contours observed by Vaghefi et al. [27] experimental (cm) and numerical (m) For square shapes, Comparisons between the bed elevation observed and simulated by [22] and present results at the plane of symmetry are shown in Figure 5. From the figure, it is declared that the observed maximum scour depth locates at the front corners of the pier which is agree well with the results of the model. However, there is still disturbance at the nose but it can be neglected.
Figure 5. Scour hole shape (a) observed and (b) simulated [22] (cm); (c) model results (m) of the experiment of Ghiassi & Abbasnia [22].

For rectangular shape with both square or round, Figures 6 and 7 show comparison between the observed scour contours of [20,21] and simulated one. Figure 6 shows that the maximum scour depth observed is 0.038 for rectangular with circular nose and 0.042 m for square nose versus the maximum scour depth simulated is 0.0375 for circular nose and 0.043 for square one. Figure 6 also present the scour hole shape observed by [21] and present results, it give the same agreement with the maximum scour depth observed 0.06 and simulated one 0.0609 m. The model gave good agreement with the observed for the maximum scour depth and outer perimeter of the scour hole.

Figure 6. Scour hole shape for experiment of [20,21] (cm) and model results (m) (a) Rectangular circular nose (a) rounded nose (Oblong)

Figure 7. Scour hole shape of [21] experiment and present result
6. Conclusions
It should also be expected that ‘perfect’ results are difficult to obtain in view of the many factors involved, which cannot be modelled directly using a numerical simulation.[16] The present investigation shows that the Flow 3D can be effectively used to estimate the maximum scour depth. The model simulates the maximum scour hole depth with sufficient accuracy on contrary with the prediction of the scour hole shape as it is failed to simulate the observed scour depth at the nose and the capability of the model to simulate the scour shape well depends on the cross sections shape and for the same cross section flow properties.

For cylindrical case, the maximum scour depth observed in experimental data located at the area close to the pier nose; while the maximum scour depth from the model situated at pier sides. The presented maximum scour depth agrees well with the experiments. However, the scour hole shape and location between the numerical simulation and the test are quite different. [11] also failed to capture the region of deep scour at the pier nose and declared that the predictive capabilities of the URANS hydro-morpho dynamic depend on the details of the geometry of the studied pier cross-section only. Although, it is found that for the circular pier the capability of the model depending on the flow properties not only the geometry of the pier.

7. References
[1] I.K. and Y.S. Hyung Suk Kim1, Mohamed Nabi2, Computational Modeling of Scours, Hydraul. Eng. (2014) 79–84.
[2] S. Dey, I.S.K. Bose, and G.L.N. Sastry, CLEAR WATER SCOUR AT CIRCULAR PIERS : A MODEL, J. Hydraul. Eng. (1995) 869–876.
[3] J.F.A.J.M. ANDREAS ROULUND1, B. MUTLU SUMER1, Numerical and experimental investigation of flow and scour around a circular pile, J. Fluid Mech. 534 (2005) 351–401. doi:10.1017/S0022112005004507.
[4] S., Richardson, E.V. and Davis, Evaluating Scour at Bridges, HEC 18, Third Ed. Novemb. 1995 Metr. Report No. (1995).
[5] K.H.M. Ali, O. Karim, Simulation of flow around piers, J. Hydraul. Res. 40 (2002) 161–174. doi:10.1080/0022168020949859.
[6] L. Ge, S.O. Lee, F. Sotiropoulos, M. Asce, T. Sturm, M. Asce, 3D Unsteady RANS Modeling of Complex Hydraulic Engineering Flows . II : Model Validation and Flow Physics, J. Hydraul. Eng. (2005) 809–820.
[7] T.M. Salaheldin, J. Imran, M.H. Chaudhry, Numerical Modeling of Three-Dimensional Flow Field Around Circular Piers, J. Hydraul. Eng. Vol. 130, (2004) 91–100.
[8] B.W. Melville, LOCAL SCOUR AT BRIDGE SITES, phD. 1994 (1975).
[9] F. Ahmed, N. Rajaratnam, Flow around Bridge Piers_Ahmed.pdf, J. Hydraul. Eng. 124 (1998) 288–300.
[10] C. Escauriaga, F. Sotiropoulos, Initial stages of erosion and bed form development in a turbulent flow around a cylindrical pier, J. Geophys. Res. Earth Surf. 116 (2011) 1–24. doi:10.1029/2010JF001749.
[11] A. Khosronejad, S. Kang, F. Sotiropoulos, Experimental and computational investigation of local scour around bridge piers, Adv. Water Resour. 37 (2012) 73–85. doi:10.1016/j.advwatres.2011.09.013.
[12] M.N. Sakib, phD-CFD Techniques for Simulation of Flow in a Scour Hole around a bridge pier, Ph.D. (2013).
[13] N.V. Thanh, D.H. Chung, T.D. Nghien, Prediction of the Local Scour at the Bridge Square Pier Using a 3D Numerical Model, Open J. Appl. Sci. 4 (2014) 34–42. doi:10.4236/ojapps.2014.42005.
[14] H.S. Kim, M. Nabi, I. Kimura, Y. Shimizu, Numerical investigation of local scour at two adjacent cylinders, Adv. Water Resour. 70 (2014) 131–147. doi:10.1016/j.advwatres.2014.04.018.
[15] B.M. Sumer1, and J. Fredsøe, SCOUR AROUND PILE IN COMBINED WAVES AND CURRENT, J. Hydraul. Eng. (2001) 403–411.
[16] A.L.J. Pang, M. Skote, S.Y. Lim, A. Gullman-strand, J. N. Morgan, A numerical approach for determining equilibrium scour depth around a mono-pile due to steady currents, Phys. Procedia. 57 (2016) 114–124. doi:10.1016/j.physprocedia.2016.02.010.
[17] Z. Xie, phD Theoretical and numerical research on sediment transport in pressurised flow conditions, Ph.D. (2011) 217.
[18] M. Akhtaruzzaman Sarker, Flow measurement around scoured bridge piers using Acoustic-Doppler Velocimeter (ADV), Flow Meas. Instrum. 9 (1998) 217–227. doi:10.1016/S0955-5986(98)00028-4.
[19] D.A. Lyn, Turbulence Models for Sediment Transport Engineering, Sediment. Eng. (2013) 763–825. doi:10.1061/9780784408148.ch16.

[20] Abdul-Hassan, Zaid Hadi, EXPERIMENTAL STUDY OF BRIDGE PIER, Int. J. Civ. Eng. Technol. 7 (2016) 162–171.

[21] C. Fael, R. Lancã, A. Cardoso, Effect of pier shape and pier alignment on the equilibrium scour depth at single piers, Int. J. Sediment Res. (2016) 1–7. doi:10.1016/j.ijsrc.2016.04.001.

[22] R. Ghiassi, A.H. Abbasnia, Investigation of Vorticity Effects on Local Scouring, Arab. J. Sci. Eng. 38 (2013) 537–548. doi:10.1007/s13369-012-0337-8.

[23] M. Vaghefi, G. and Masoud, The effect of circular bridge piers with different inclination angles toward, Sadhana. (2015). doi:10.1007/s12046-015-0443-x.

[24] R. Ettema, Scour at bridge piers, Rep. No. 216, Sch. Eng. Univ. Auckland, Auckland, New Zealand, 527pp, 1980. (1980).

[25] S.S. Shen, H. W., Schneider, V. R., and Karaki, Mechanics of local scour., U.S. Dep. Commer. Natl. Bur. Stan-Dards, Inst. Appl. Technol. Washington, D.C. (1966).

[26] Z.H.U. Zhi-wen, L.I.U. Zhen-qing, CFD prediction of local scour hole around bridge piers, J. Cent. South Univ. (2012) 273–281. doi:10.1007/s11771.

[27] M. Vaghefi, M. Ghodsian, S. Salimi, Scour Formation Due to Laterally Inclined Circular Pier, Arab. J. Sci. Eng. 41 (2016) 1311–1318. doi:10.1007/s13369-015-1920-6.

8. Acknowledgments
The first researcher is supported by a scholarship from the Mission Department for the Government of Egypt which is gratefully acknowledged. Second I am thankful to Egypt-Japan University of Science and Technology (E-JUST) and Japan International Cooperation Agency (JICA) for offering the tools and equipment needed for the research work.

9. Notation
D  pier diameter/width;
ds  equilibrium scour depth;
D50  mean sediment size;
gi  gravitational acceleration in i th direction
k  von Karman constant;
So  longitudinal slope of flume;
t  time;
U  average flow velocity;
u*  shear velocity;
Z+  wall unit distance;
zp  distance from point p to wall;
q  volume fraction of qth phase in control volume;
ΔB  roughness function;