

Simulation of Local Scour around Circular and Round Nosed Bridge Pier using REEF3D

Sreedhara B M¹, Sanooj A², Manu³, S Mandal³

Research Scholar, Dept. of Applied Mechanics & Hydraulics, National Institute of Technology Karnataka, Surathkal, India¹

P.G. Student, Dept. of Applied Mechanics & Hydraulics, National Institute of Technology Karnataka, Surathkal, India²

Professor, Dept. of Applied Mechanics & Hydraulics, National Institute of Technology Karnataka, Surathkal India³

ABSTRACT: Sediment transport and resulting local scour around the bridge pier can leads to the failure of structure. The knowledge and understanding the erosion and prediction of scour depth is difficult for the structural design. In the current study the physics of local scour under current condition are modelled using CFD tool. The results obtained from numerical model are compared with experimental observations, showing good agreement. The study also concludes the effect of shape of pier on, velocity profile, mechanism and magnitude of local scour. The 3D CFD model REEF3D is used to calculate the detailed flow field and the resulting sediment transport pattern around the pier.

KEYWORDS: Local Scour, Numerical Modelling, REEF3D,

I. INTRODUCTION

Bridges are very important civil engineering structures of our transportation system, which provides the passage over a gap or a barrier across a river or a valley. But, while designing bridge structures the main problem encountered is the estimation of scour depth near piers and abutments. Scour occurs due to the erosion of streambed or bank material from bridge foundations due to flowing water which causes bridge failures. The bed level set lowered and the bridge foundations will get exposed as a result of erosion. The bed and banks materials of streams and from the vicinity of the piers and abutments of bridges get carried away and excavated due to the erosive action of flowing water [9]. This situation leads to structural collapse which causes loss of life and property. The correct prediction of local sediment transport in the vicinity of structures is an important research field due to its significant practical value as prediction is necessary for calculation of the scouring risk of the structures and the changes in bed form. An accurate prediction of scour depth and its evolution in time is crucial. The need of extensive experience and physical insight into the hydrodynamics, sediment transport and morphological processes are hence necessary to tackle these issues of high importance.

The total scour at a bridge pier may be as a result of one or more types of scouring. All these types of scour generates from different causes, and hence we need no estimate each of these scour individually and then should be added linearly to estimate the maximum scour [10]. The different types of scour are general scour, localised scour, constriction scour, local scour, clear-water scour and live bed scour. The cause of generation of local scour is due to the interference of natural flowing water with the river structures. This interaction causes formation of vortices at the base of the structure which will finally result in the erosion of bed material from its base and a hole is formed, which is termed as local scour. The scour at the surrounding of bridge pier, abutments spur dykes and river training works are few examples of local scour.

For the estimation of local scour, basically there are three approaches, namely, experimental, numerical and soft-computing techniques. The recent investigations done in this domain is through numerical modelling. The numerical model being developed for the present study is achieved with the help of REEF3D, an open-source computational fluid dynamics program. The use of the level-set method enables this CFD tool to calculate complex free surface of

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

calculating sediment transport [6]. It uses conventional formulae for bedload and suspended load. Then the Exner equation[8] can be solved and the resulting erosion or deposition is used to change the bed topography. The bed is geometrically represented with a level set function (separate from the free surface). This has an advantage, that re-meshing is not necessary, and thus improving the accuracy and the stability of the sediment transport algorithm[7]. Hence, the open source REEF3D program will be used in the present study for the prediction of the local scour depth around a bridge pier, using the experimental data collected.

II. NUMERICAL MODEL

To define the velocity and pressure field of the fluid domain to be stimulated, the CFD code incorporates the continuity as well as the incompressible RANS equations which are governing equations for mass and momentum conservation[4].

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t) \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + g_i \quad (2)$$

Where u is the velocity, ρ is the density of the fluid, p is the pressure, ν_t is the eddy viscosity, ν is the kinematic viscosity, and g is the acceleration due to gravity. The turbulence is being modelled by use of RANS equations. The term ν_t denotes eddy viscosity which is determined with the use of $k-\omega$ model[5]. For pressure gradient modelling of the RANS equation, the projection method is being used. For this projection, the time is treated explicitly in the incompressible flow Navier-Stokes equation.

The RANS equation's convective terms are discretised within a conservative finite difference framework through the 5th order (WENO) weighted essentially non oscillatory scheme[2].

In the present study, for modelling the boundary between the three phases, i.e., water-air and water-sediments, the LSM[11] is used. The location of interface will be denoted indirectly by the use of zero level-set of the smooth signed distance function. The change of sign distinguishes the phases.

III. SEDIMENT TRANSPORT MODEL

Bed shear stress formulations based on turbulent viscosity[12] is used for calculating the shear stress of bed

$$\tau = -\rho(\nu_t + \nu) \frac{\partial U}{\partial z} \quad (3)$$

Bed load formula by Engelund and Fredsoe[1] was used to calculate sediment transport rate in the bed cells

$$q_{b,i}^* = 0; \tau^* < \tau_{c,i}^* \\ q_{b,i}^* = 18.74(\tau^* - \tau_{c,i}^*)(\tau^{*0.5} - 0.7\tau_{c,i}^{*0.5}); \tau > \tau_{c,i}^* \quad (4)$$

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

$$\tau_{c,i}^* = \frac{\tau_{c,i}}{(\rho_s - \rho) g d_i}$$

Where ρ_s is the sediment density, ρ is the density of the flowing water, g is the acceleration due to gravity, and d_i is the particle diameter of the sediment. $q_{b,i}^*$ is the rate of bed load transport and $\tau_{c,i}^*$ is the critical shear stress[3].

$$\frac{\partial c}{\partial t} + U_j \frac{\partial c}{\partial x_j} + w_s \frac{\partial c}{\partial z} = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial c}{\partial x_j} \right) \quad (5)$$

$$D = w_s c_b \quad (6)$$

Where w_s is the falling velocity of the sediment particles, Γ is the diffusion coefficient. D is the deposition rate and c_b is the concentration of sediment nearby the bed. Also, rate of entrainment E is estimated with the suspended load formula by van Rijn[11].

$$c_{bed,susp,load,i} = \frac{E}{w_s}$$

$$c_{bed,susp,load,i} = 0.015 \frac{d_i}{\alpha} \frac{\left(\frac{\tau - \tau_{c,i}}{\tau_{c,i}} \right)^{1.5}}{\left(d_i \left(\frac{\rho_s (\rho_w - 1)}{v^2} \right)^{1/3} \right)^{0.8}} \quad (7)$$

Tracking of the movable sediment surface, is being done through the land slide model (LSM). The changing bed is implicitly represented by means of the zero level-set of the smooth signed distance function $\phi(x,t) = 0$. Corresponding to the motion of the interface, the change of $\phi(x,t)$ is attained by shifting the level-set, up and down in dependence with the erosion and deposition of bed cells[13].

IV. EXPERIMENTAL SETUP AND NUMERICAL TANK

The experimental tests were carried out in a tilting wave flume at the Department of Applied Mechanics and Hydraulics, National Institute of Technology Surathkal by Yathish (2010). The hydrodynamic flume used was a tilting and re-circulating flume type. It is 0.6m wide, 0.58m deep and 16.85m long from inlet to outlet. The whole section was filled with sediment layer of 10cm thickness. The tests were conducted for two model, a circular pier model and for a round nosed pier. The circular pier model was fabricated from PVC with an external diameter of 47mm. The round nosed pier was fabricated from wood, with a width of 47 mm and length as 3times the width of pier which is 141mm. The sand used was cohesion less and has median particle size (d_{50}) = 0.6 mm. The angle of repose of the sediment used was 20 degree. All the equilibrium scour experiments conducted were run for 24 hours.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

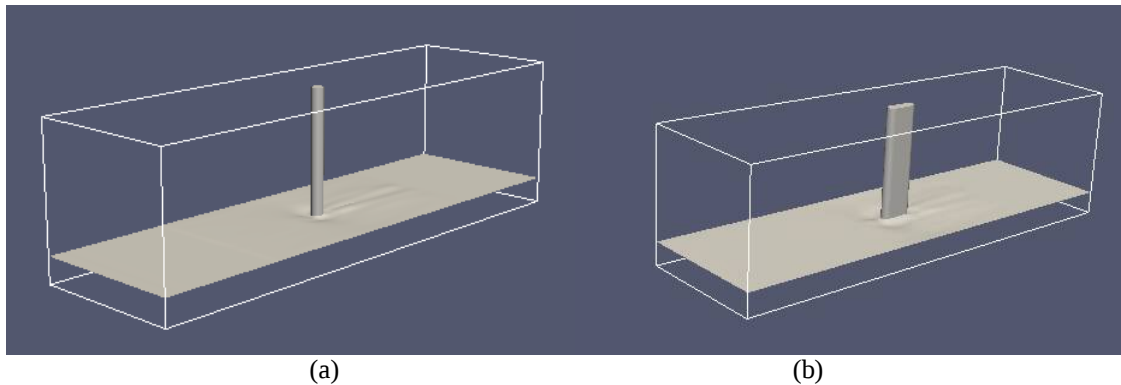


Figure 1: Numerical wave tank (a) with Circular pier and (b) with Round nosed pier

In this present study, the importance is on the local scour around the pile. The numerical setup used consists of a 3D channel which is 2m long, 0.6m wide and 0.58m depth with a water depth of 0.94m. The diameter of pier is 47 mm. The thickness of sediment layer was limited to 0.10m in the numerical flume. The cell size chosen for the study was 0.020m. The chosen parameters and test condition for experimental and numerical simulation are shown in below table.

V. NUMERICAL RESULTS AND DISCUSSION

A Circular pier and Round nosed pier are simulated. The other parameters for the flow and sediment properties are same for both tests. The numerical simulation is run for the current action. The input parameters with simulated scour depth are listed in table 1. The results, velocity profile and scour depth near the pier are similar to the experimental observation, still the simulation process is running to get still more better results for different input parameters.

Table 1: Details of parameters used in Simulation

Parameters	Circular Pier	Round Nosed Pier
Cylinder Dimension (mm)	47	47 X 141
Sediment size d50 (mm)	0.6	0.6
Sediment Depth (mm)	100	100
Flow Depth (mm)	94	94
Discharge Q (m3/S)	0.0125	0.0125
REEF3D Scour Depth (mm)	23.10	14.20

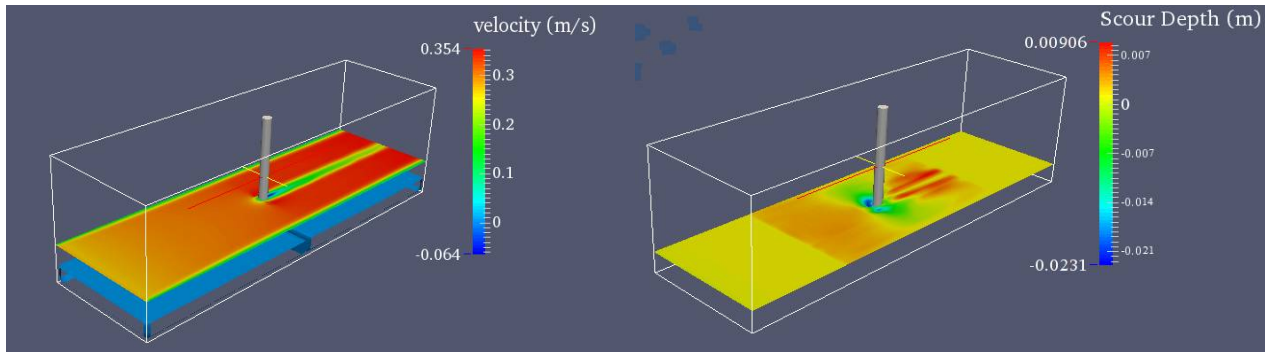
CIRCULAR PIER

At first a simulation is run for the circular pier exposed to current action. The circular pier creates sudden obstruction in the flow, which gives change in velocity in the flow as shown in fig. 2 (a) and leads to the formation of scour hole near pier as shown in fig. 2(b).

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

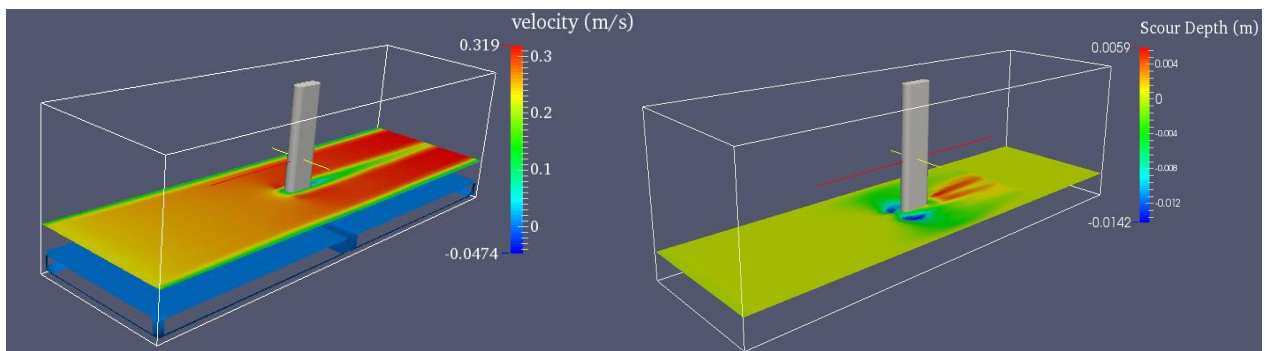
Vol. 5, Special Issue 9, May 2016



(a) (b)
Figure 2: (a) velocity profile and (b) local scour around Circular pier

ROUND NOSED PIER

In the second simulation the shape of the pier changed to round nosed type and keeping the other parameters same as in the previous section. The length of the pier is three times the diameter of the pier. The simulation is run for the current condition, it results in velocity profile changes as shown in fig. 3 (a) and causes scouring in upstream nose of the pier as shown in fig. 3 (b).



(a) (b)
Figure 3: (a) velocity profile and (b) local scour around Round nosed pier

VI. CONCLUSIONS

Numerical simulations of local scour around circular and round nosed pier are performed using the REEF3D tool. The flow for current condition is generated in numerical tank and free surface is captured using level set method. The local scour simulations are run to study the velocity profile and scour pattern around circular and round nosed pier. Finally, the simulated results are compared with circular and round nosed shape pier and concluded. The change in velocity of flow due obstruction is high in the circular pier as compared with round nosed pier. The depth of scour hole is higher near circular pier.

From the results, we found that the velocity profile and scour formation mechanism are similar to the experimental observations and still the numerical investigations are being continued for different flow and sediment condition to get much more good results.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

REFERENCES

- [1] Englund, F., and Fredsøe, J., 1976, "A Sediment Transport Model for Straight Alluvial Channels," Nord. Hydrol., 7(5), pp. 293–306.
- [2] Jiang, G. S., and Shu, C. W., 1996, "Efficient Implementation of Weighted Eno Schemes," J. Comput. Phys., 126(1), pp. 202–228.
- [3] L. C. v. Rijn, Sediment transport, part i: Bed load transport, Journal of Hydraulic Engineering 110 (1984) 1431–1457.
- [4] Martin Diken , B. Mutlu Sumer , Jørgen Fredsøe(2013) "Numerical and experimental investigation of flow and scour around a half-buried sphere" Coastal Engineering 73, 84–105
- [5] Mohammad Ehterama, Amin Mahdavi Meymand(2015) "Numerical modeling of scour depth at side piers of the bridge" Journal of Computational and Applied Mathematics 280 , 68–79
- [6] Mohammad Saud Afzal, Hans Bihs, Arun Kamath, Øivind A. Arntsen(2015) "Three-Dimensional Numerical Modelling of Pier Scour Under Current and Waves Using Level-Set Method" Journal of Offshore Mechanics and Arctic Engineering Vol. 137, 032001-7
- [7] Nadeem Ahmad, Hans Bihs, Arun Kamath, Øivind A. Arntsen(2015) "Three-dimensional CFD modeling of wave scour around side-by-side and triangular arrangement of piles with REEF3D" Procedia Engineering 116 (2015) 683 – 690
- [8] Osher, S., and Sethian, J. A., 1988, "Fronts Propagating With Curvature-Dependent Speed: Algorithms Based on Hamilton-Jacobi Formulations," J. Comput. Phys., 79(1), pp. 12–49.
- [9] Richardson, E.V. and Davies, S.R. 2001. "Evaluating scour at bridge pier" 4th Edition FHWA NHI 01-001 (HEC-18), Federal Administration, U.S. Department of Transportation, Washington, D.C.
- [10] Sreedhara B M, Manu, Pruthviraj U (2015) "Comparative Study on different Bridge Scour Monitoring Techniques: A Review" HYDRO 2015 International Conference, DOI: 10.13140/RG.2.1.3197.1925.
- [11] van Rijn, L. C., 1984, "Sediment Transport—Part II: Suspended Load Transport," J. Hydraul. Eng., 110(11), pp. 1613–1641.
- [12] Wilcox, D. C., 1994, "Turbulence Modeling for CFD", DCW Industries, La Canada, CA.
- [13] Xiaofeng Liu and Marcelo H. García "Three-Dimensional Numerical Model with Free Water Surface and Mesh Deformation for Local Sediment Scour" JOURNAL OF WATERWAY, PORT, COASTAL, AND OCEAN ENGINEERING 2008.134:203-217.