XVIII International Conference on Water Resources
CMWR 2010
J. Carrera (Ed)
CIMNE, Barcelona 2010

3D NUMERICAL MODELLING OF LOCAL SCOUR AROUND THE CYLINDRICAL BRIDGE PIERS

A.Azhari *, S.F.Saghravani † and B.A. Mohammadnezhad ††

* Phd.Condidate, Shahrood University of Technology (SUT), Shahrood, Iran.
E-mail: amir@azhari.ir

†Assistant Professor, Shahrood University of Technology (SUT), Shahrood, Iran.
E-mail: sfsaghravani@yahoo.com

†† Assistant Professor, Water Engineering Department, Urmia Univercity, Urmia, Iran
Email: b.mohammadnezhad@urmia.ac.ir

Key words: Local scour, Bridge pier, Turbulence, Multiphase, 3-D model.

Summary. *A 3-D numerical model and a physical model were employed to simulate and predict the local scour depth around a set of three vertical circular piers located in a river. Computations were performed using k-ε turbulence model. The flow was considered unsteady. The Eulerian granular multiphase model for solving related equations was applied. Based on the results, the maximum scour depth occurs around the first pier. The time course tests were showed that 80% of scour hole is extracted in the first hour. The results from the CFD tests showed a good agreement with the experimental data collected from the physical model which an indication of the precision of the chosen CFD procedure.*

1 INTRODUCTION

In the recent years, many researchers investigated the complex flow pattern and sediment transport around bridge piers. Dou et al.[1] (1998) and Dou and Jones[2] (2000) carried out numerical calculations to evaluate the rate of sediment transport for local scour around a bridge pier. They proposed an equation for effective sediment transport capacity (STC), which included the effects of dune flow, vortices and local turbulence, to simulate local scour depth in a 3-D numerical model. Sand movement by wind around cylindrical bodies was investigated by Kan and Kawamura[3] (2000). They calculated the 3-D flow field around a cylindrical body by the mean of the maker and cell (MAC) method. Conservation of sand was employed to estimate the bed profile. Olsen and Kjellesvig[4] (1998) suggested a combined 3-D hydraulic and sediment transport models for the suspended sediment concentration for scour modeling which was used to simulate the flow and scouring around a single cylindrical pier. The bed concentration formula (Van-Rijn 1987) was used for the equilibrium sediment concentration near the bed. Since the equation consists of some empirical coefficients, the scour depth may not be estimated precisely. Moreover, the numerical results did not provide enough information about unsteadiness in the flow. Liang Ge and Seung oh lee[5] (2005) developed a chimera overset grid flow solver for solving the URANS equations in arbitrarily complex, multi connected domains. Their method was validated and applied to investigate the physics of flow passed a real bridge foundation mounted on a fixed flat bed. It was shown that the numerical model can reproduce large scale unsteady vortices that contain a significant portion of the total turbulence kinetic energydeveloped a chimera overset grid flow solver for solving the unsteady Reynolds-averaged Navier-Stokes (RANS) equations in arbitrarily complex, multi connected domains. Their method was validated and applied to investigate the physics of flow passed a real-life bridge foundation mounted on a fixed flat bed. It was shown that the numerical model can reproduce large scale unsteady vortices that contain a significant portion of the total turbulence kinetic energy. Huang et al.[6] (2009), conducted a series of computational model simulations using 3-D CFD models to examine the scale effects in turbulent flows and sediment scour.

In the present research, the scour around a set of 3 round piers was studied experimentally and numerically. The results of experiments would be helpful in verifying the numerical procedure used.

2 GOVERNING EQUATIONS OF THE TWO-PHASE MODEL

The Navier-Stokes and continuity equations, for both the fluid (f) and solid (s) phases, were employed to simulate the flow through a 3-D CFD code. The Eulerian-Eulerian multiphase model was used in which the continuity and the momentum equations are solved for each phase and therefore, the determination of separate flow field solutions is allowed.
The continuity equations for the fluid (f) and solid (s) phases are in the form of:

$$\frac{\partial}{\partial t}(\alpha_q\rho_q)+\nabla(\alpha_q\rho_q v_q)=0 \qquad (1)$$

where subcript q is (s),and (f), $\alpha_s + \alpha_l = 1$; α_s and α_l are volume fraction for solid and water phase; ρ_s and ρ_l are mass density of solid and water respectively.
The momentum equation for the water phase is:

$$\frac{\partial}{\partial t}(\alpha_l\rho_l\vec{v}_l)+\nabla.(\alpha_l\rho_l\vec{v}_l\vec{v}_l)=-\alpha_l\nabla p+\nabla.\bar{\bar{\tau}}_l+\alpha_l\rho_l\vec{g}+K_{s,l}(\vec{v}_s-\vec{v}_l) \qquad (2)$$

The conservation of momentum for the s^{th} solid phase is:

$$\frac{\partial}{\partial t}(\alpha_s\rho_s\vec{v}_s)+\nabla.(\alpha_s\rho_s\vec{v}_s\vec{v}_s)=-\alpha_s\nabla p-\nabla p_s+\nabla.\bar{\bar{\tau}}_s+\alpha_s\rho_s\vec{g}+\alpha_s\rho_s(\vec{F}_s+\vec{F}_{lift,s}+\vec{F}_{vm,s})+$$
$$m\textstyle\sum_{l-1}^{N}(K_{l,s}(\vec{v}_l-\vec{v}_s)+\dot{m}_{ls}\vec{v}_{ls}) \qquad (3)$$

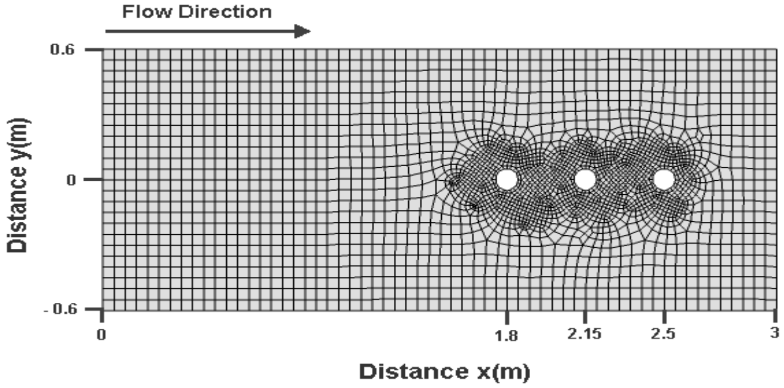
Where, p_s is the s^{th} solids pressure, $K_{ls} = K_{sl}$ is the momentum exchange coefficient between fluid phase l and solid phase s, N is the total number of phases. The lift force $F_{lift,s}$ and the virtual mass force $F_{vm,s}$ have been neglected in the calculations, because they give a minor contribution to the solution with respect to the other terms.

3 NUMERICAL SCHEME AND BOUNDARY CONDITIONS

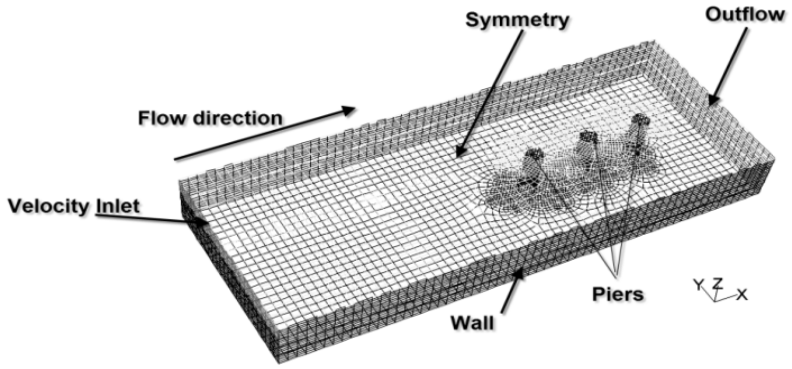
The finite volume scheme was used to solve the governing equations with an unstructured mesh. Typical boundary conditions used in this study are solid wall, outflow, specified velocity inlet, and symmetry. Fig.1 (a) and (b) show the mesh and boundary conditions of the model. The model consists of three cylindrical piers, 10 cm in diameter and apart 35 cm center to center that were placed in the central axis of the river channel. The inlet velocity was set to 0.35 m/sec. The model geometrical data is presented in table 1.

Length (m)	3
Width (m)	1.2
Water Depth (cm)	12
Sand Depth (cm)	25

Table 1: Properties of the model.



(a)



(b)

Figure 1: Numerical model description (a) Mesh and, (b) Boundary conditions

4 EXPERIMENTAL SETUP

The experiments were conducted at Water Research Institute (WRI), Tehran, Iran. Tests were carried out in a rectangular flume with concrete bed and walls. The length, width, and depth of the flume were 8.0, 1.2, 0.7 (all in meter) respectively. The flow was allowed to be developed in the first 5m, thus the piers were placed in the last 3m. Fig. 2 shows a snapshot of experimental model. The bottom of the flume was covered with 25 cm of sand with the size d_{50}=1.3 mm, the standard variation σ_g (= $\sqrt{\frac{d84}{d16}}$) = 1.4 and specific gravity ρ= 2.6. All experiments were carried out with clear water.

The bed scour profiles were measured using a NANO AGE, BPF-LMC-2 bed-profiler which sweeps the bottom using a green laser beam with 0.1 mm precision. The time variation of the scour depth in front of the each pier were measured using an ADV.

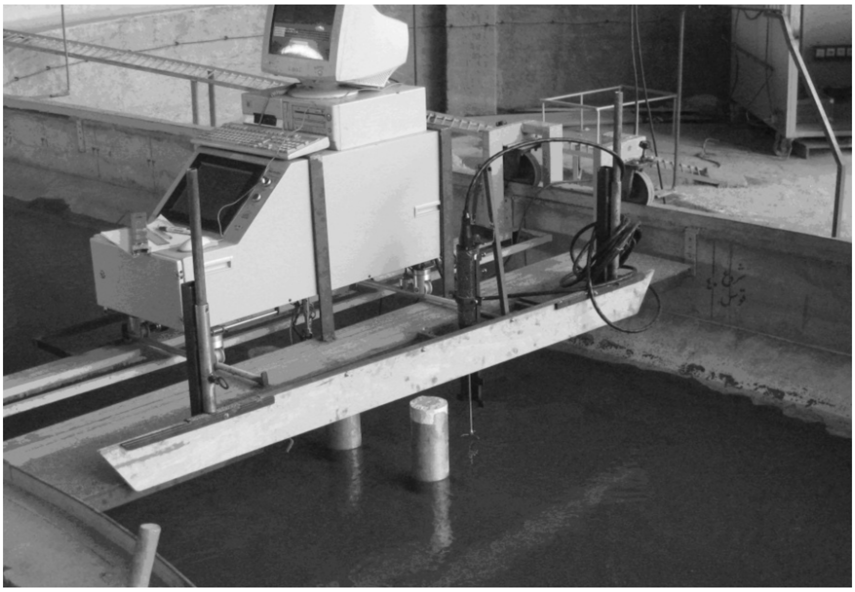
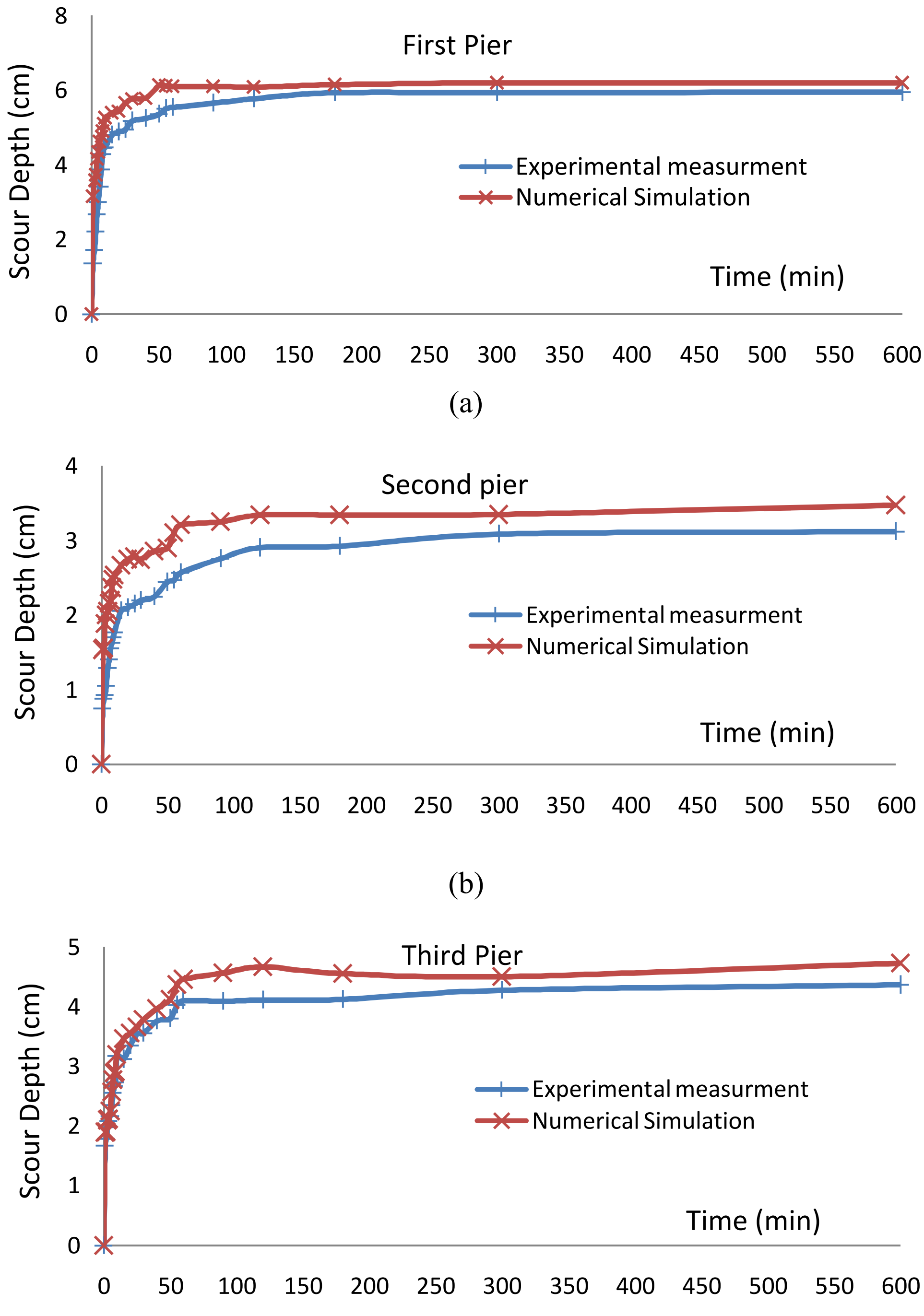


Fig. 2: A snapshot of experimental model

5 NUMERICAL AND EXPERIMENTAL RESULTS

In the numerical simulations, the volume fraction of sediment contour $\alpha_s \approx 0.5$ was chosen as the bed profiles[7]. Figures 3 (a), (b), and (c) show the results of experimental and numerical tests as time series of scour depth 3cm upstream of each pier. Figures 4 and 5 show the computed and experimental bed contours at T=10 and 600 min, respectively. Figure 6 shows a comparison between scour depth variations with time at 3 cm in front of piers obtained by experimental tests.



(a)

(b)

(c)

Figure 3: Scour depth variation with time at 3 cm in front of each pier

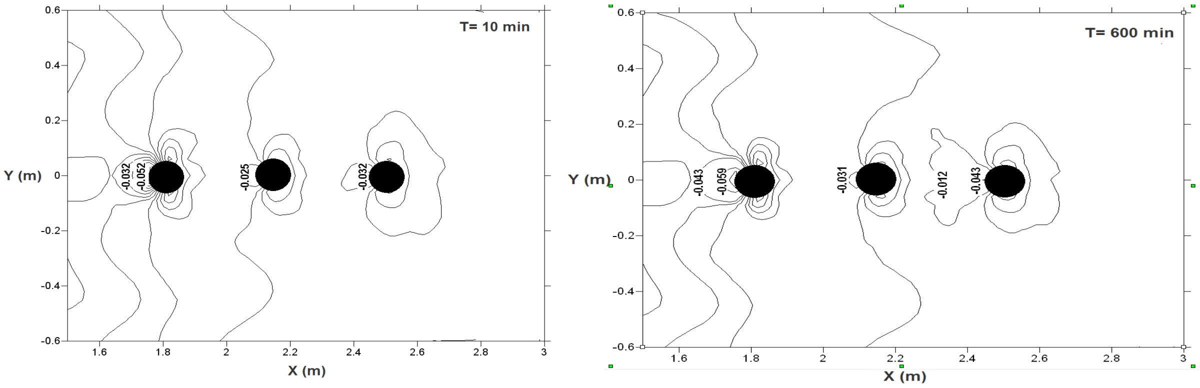


Figure 4: Bed profiles during the development of scouring resulted from Numerical model for t= 10 and 600 min.

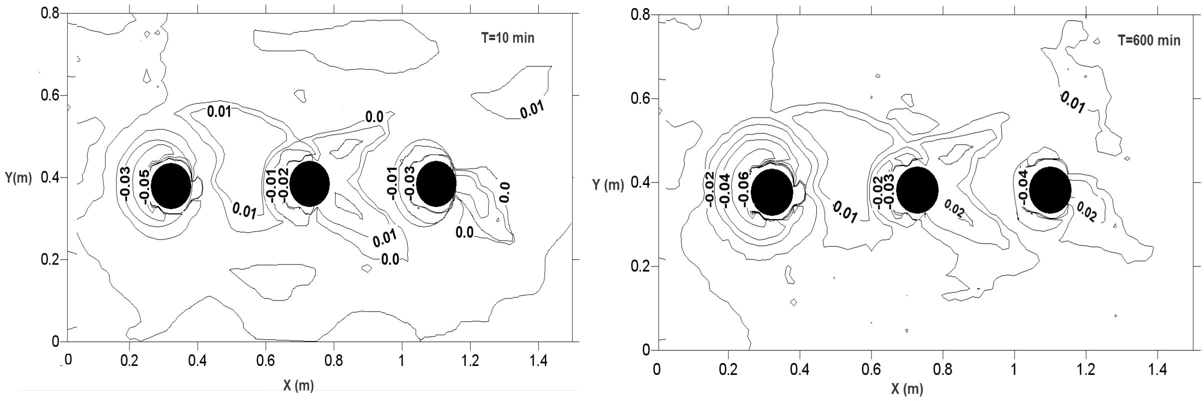


Figure 5: Bed profiles during the development of scouring resulted from experimental model for t= 10 and 600 min.

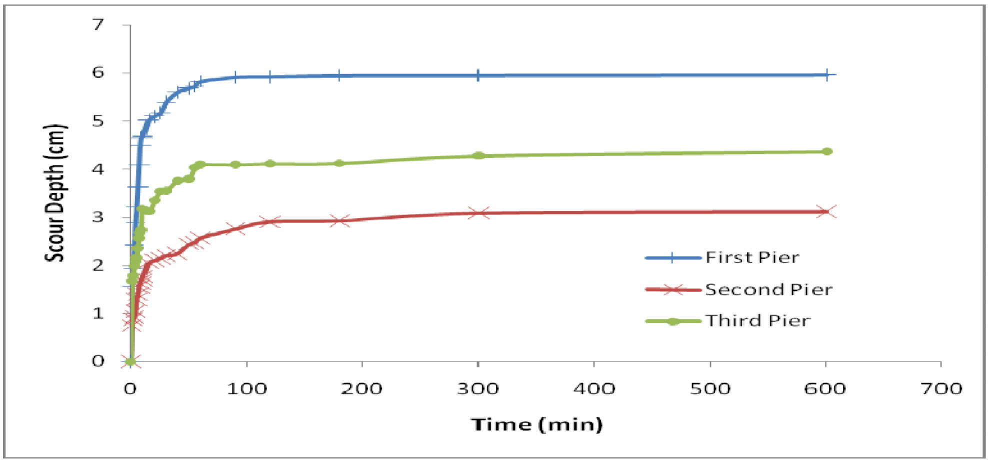


Figure 6: Comparision of scour depth vaiaion with time in the piers obtained by experimental tests .

(At 3 cm in front the pier)

6 CONCLUSIONS

A two-phase flow numerical model along with a physical model was employed in order to investigate the progressive development of the scour holes around a set of 3 cylindrical bridge piers. The results were in a good agreement which is an indication of accuracy of the numerical model and the solution procedure. The following conclusions could be made to the results of this research:

1. The numerical model predicts a bigger scour depth than the measured depth in the physical model.
2. The scour depths in front of the piers are in the order of the front pier, the back pier, and the middle pier.
3. Time variation of scour indicated that 80% of equilibrium scour depth occurs in the first hour.
4. The numerical models can be used a powerful and inexpensive tool by designers and engineers although the need for physical models cannot be ignored based on this investigation.

ACKNOWLEDGEMENT: This research was technically and financially supported by Water Research Institute, affiliated to the Iranian Ministry of Energy

REFERENCES

[1] Dou, X., Jones, J. S., Young, G. K., and Stein, S. M. (1998). "Using a 3-D model to predict local scour." International Water Resources Conference, Memphis, 198-203.

[2] Dou, X. S., and Jones, J. S. (2000). "A new sediment transport formula for local scour prediction." Joint Conference on Water Resources Engineering and Water Resources Planning & Management, Minneapolis.

[3] Kan, M., and Kawamura, T. (2000). "Numerical study of flow structure around a flat plate standing on the sand." Theoretical and Applied Mechanics, 49, 197-203.

[4] Olsen, N. R. B., and Kjellesvig, H. M. (1998). "Three-dimensional numerical flow modeling for estimation of maximum local scour depth." Journal of Hydraulic Research/De Recherches Hydrauliques, 36(4), 579-590.

[5] Ge, L., and Seoung, O, L. (2005). "3D unsteady RANS modeling of complex hydraulic engineering flows. I: Numerical model." Journal of Hydraulic Engineering, 131(9), 800-808.

[6] W. Huang et al. , Computers & Fluids 38 (2009) 1050–1058,ELSEVIER, CFD modeling of scale effects on turbulence flow and scour around bridge piers.

[7] Zhihe Zhao ., H. J. S. Fernando. Environ Fluid Mech,DOI 10.1007/s10652-007-9017-8, Springer (2007), Numerical simulation of scour around pipelines using an Euler–Euler coupled two-phase model.