18th International Conference on
TRANSPORT AND SEDIMENTATION OF SOLID PARTICLES
11-15 September 2017, Prague, Czech Republic
ISSN 0867-7964 ISBN 978-83-7717-269-8

HYDRO-MORPHODYNAMIC MODELLING OF LOCAL BRIDGE PIER SCOUR IN ALLUVIAL BEDS

JK Vonkeman[1], GR Basson[2] & GJF Smit[3]

1 PhD student, Dept. of Civil Eng., Stellenbosch University, Private Bag X1 Matieland 7602 South Africa, Email: jkvonkeman@gmail.com
2 Prof., Head Water Division, Dept. of Civil Eng., Stellenbosch University, Private Bag X1 Matieland 7602 South Africa, Email: grbasson@sun.ac.za
3 Prof., Dept. of Applied Mathematics, Stellenbosch University, Private Bag X1 Matieland 7602 South Africa, Email: fsmit@sun.ac.za

The key aim of the study was to determine whether a hydro-morphodynamic model, recently developed by Sawadogo (2015), could be applied to simulate local scour around a cylindrical bridge pier. ANSYS Fluent solved flow equations for the Eulerian multiphase and k-ε turbulent models which were coupled with sediment transport and bed deformation sub-models. Despite the extensive studies on bridge pier scour that have been done since the 1950's, comparatively few of these addressed numerical modelling. Unlike existing methods, the proposed model is coupled, fully 3D and employs more than one bed load function. It was established that the hydrodynamic model performs reasonably well in resolving the complex flow field and the crucial horseshoe vortex. The model's ability to simulate the flow field is constrained by the 1st order k-ε model. Nevertheless, the results are less sensitive to the choice of turbulence model than the computational grid. The numerical simulations are highly sensitive to the mesh resolution and aspect ratio to resolve the vortices and to obtain stability. Sawadogo (2015) also discovered that the proposed model is sensitive to a fine mesh which could yield an unstable irregularly shaped scour hole. The proposed model has demonstrated that it has the potential to simulate pier scour but the instabilities encountered by the morphodynamic sub-models require further research to master the scour simulation.

KEY WORDS: CFD, Numerical Modelling, Local Bridge Pier Scour

1. INTRODUCTION

Local bridge pier scour, or the washing away of riverbed material in the vicinity of a pier during floods, has been attributed as the main cause of failure of bridges founded in alluvial beds (Deshmukh & Raikar, 2014). In fact, nearly two thirds of structural bridge failures in the United States are due to scouring and not to overloading (Sumer, 2007). In spite of the extensive studies that have been conducted on bridge pier scour for the last six decades, there is still no universally agreed upon design procedure to accurately predict bridge pier scour (Rooseboom, 2013).

The scouring process is complex owing to the formation of three-dimensional (3D) boundary layers and the intricate horseshoe vortex, which is the mechanism responsible for scouring. A downflow, driven by a strong pressure gradient and vertical velocity component upstream of the pier, rolls up when it comes into contact with the riverbed. The resulting circulation and flow separation forms the vortex, which acts as an impinging jet, digging up sediment.

The local scour process is affected by several different yet interrelated flow, pier and sediment parameters that limit the extent to which a mathematical analysis can be made (Chiew, 1984). Traditional methods of predicting the equilibrium scour depth near bridge piers rely on overly simplified formulas that have been calibrated by small scale laboratory experiments. These methods often yield a range of varying unreliable results.

Alternatively, advanced Computational Fluid Dynamics (CFD) models are becoming increasingly popular as technology advances and the cost of computational time decreases. Although extensive research has been conducted on pier scour for more than six decades, comparatively few studies have been presented on numerical modelling (Baykal et al., 2015). The accuracy of their studies depends on the model's ability to resolve the vortices as well as the selected sediment transport model (Salaheldin et al., 2004). Consequently, the majority of the studies have attempted to resolve the flow for a flat rigid bed and not to model sediment transport.

Commercial software exists to model sediment transport though they are not fully coupled (Afzul, 2013). In other words, the same time step is used for the turbulent flow simulation as for the scouring component despite their discrepancy in temporal scales, which is in the order of seconds and of hours, respectively (Lui & Garcia, 2008). The interaction between fluid and sediment is described as coupled because the sediment transport modifies the flow but also the bed in terms of elevation, slope and roughness, which in turn modifies the local flow field over different time scales. Furthermore, the existing numerical models are not fully 3D as they use a layer-averaged approach in conjunction with Saint-Venant equations (Sawadogo, 2015).

2. NUMERICAL MODELLING

2.1. OUTLINE OF THE PROPOSED MODEL

Sawadogo (2015) employed a coupled and fully 3D numerical model to investigate the scour pattern caused by bottom outlet sediment flushing. He appropriated the approach by Schneiderbauer & Pirker (2014) to model aeolian snow-transport. The key aim of the present study was to determine whether this model could also be applied to other fields of sediment transport, in particular to simulate bridge pier scour in an alluvial bed. In addition to being coupled and fully 3D, the proposed model differentiates between saltating and rolling modes of particle entrainment by employing more than one bed load function.

ANSYS Fluent was used to solve the Navier-Stokes flow equations coupled with the user-defined sediment transport and bed deformation sub-models. A brief outline of the proposed model is presented; however, the reader is recommended to consult the dissertation by Sawadogo (2015) for further calculation details.

The hydrodynamic model follows an Eulerian multiphase approach which accounts for three phases, namely water, rolling and saltating particles, respectively. The 3D governing equations used to model the flow field include continuity and the conservation of momentum. These equations are closed by the standard k-ε turbulent model, which is widely used for its simplicity and low computational demands.

The morphodynamic model considers the particulate phase as a continuum and the packed bed surface as a rough wall, whereby the quantity of entrained particles is determined from the wall shear forces of the fluid. The mass source term from the continuity equation accounts for the erosion and deposition of the two particulate phases, which were modelled by considering the following modes of alluvial particle transport:

- Particles rolling due to hydrodynamic shearing;
- Particle entrainment due to shear stresses;
- The ejection of saltating particles due to impacting particles;
- Particles rebound or trapped after impact;
- Particle deposition and accumulation; and
- Shearing slides of particles on slopes.

Finally, the Immersion Boundary (IB) method was employed to model the packed bed deformation whereby the volume fraction of a transport diffusion equation tracked the bed surface inside the computational domain. In addition, the IB method applies a forcing scheme to the momentum and turbulence equations to suppress velocities and the turbulent quantities within the packed bed.

2.2. NUMERICAL SOLUTION TECHNIQUE

The flow field and scour process associated with bridge pier scour is distinctly different form that of bottom outlet sediment flushing owing to the formation of complex vortices. Thus, the model setup was an imperative step to resolve the vortex structures, particularly for a k-ε turbulence model (Richardson & Panchang, 1998).

The results from the numerical model were validated against experimental data for a cylindrical pier. The flow was characterized by a velocity range of 0.14 to 0.37 *m/s* for the clear-water scour condition $V/Vc < 1$. The sediment bed was defined by a particle size of 0.74 *mm*, a relative density of 1.28 and a 45° saturated angle of repose.

The computational domain was modelled for a 1 *m* width and 0.2 *m* water depth with a 0.11 *m* pier diameter *D*. The IB method required that the 0.2 *m* deep sediment bed was included in the computational domain which extended 0.4 *m* upstream and 0.8 *m* downstream of the pier. A distance of approximately 12*D* was modelled downstream of the pier to ensure the outflow remained undisturbed (Salahaldin et al., 2004).

A combination of a mass-flow inlet boundary and a pressure outlet boundary was used for its robustness. The free surface of the fluid was approximated as a rigid lid by a symmetry boundary, an approach which is widely adopted for its computational simplicity. Such an assumption is justified in instances where significant changes in the water level and bow wave are negligible and have a small Froude number ($Fr < 0.2$ according to Roulund et al., 2005). The sidewalls and the pier surface were set as wall boundaries whereby a no-slip condition was defined, which is imperative for the boundary layer to form the horseshoe vortex (Richardson & Panchang, 1998). Furthermore, the proposed model defines the surface of the sediment bed as a rough wall by applying the law-of-the-wall.

The law-of-the-wall was also used as a guideline to establish that a mesh resolution of 1.8 *mm* and unit aspect ratio was required to capture the boundary layer recirculation. However, the objective was to limit the computational grid to 1.5 million cells and thus a coarser cell resolution of 25 *mm* was permitted at regions further away from the area of interest at the pier, while maintaining a maximum aspect ratio of 10. It was found that larger aspect ratios, even in regions of 1*D* flow, would cause divergence because the sediment transport equations depend on the area-to-volume ratio of the grid cells.

A multi-block grid was used to accommodate the round pier in a hex-dominant fluid domain and several different grids were constructed to determine which meshing approach (whether unstructured, nonorthogonal curvilinear or orthogonal) offered the best mesh quality and the least skewed cells. The final approach adopted by the computational grid and domain are shown in Fig.1.

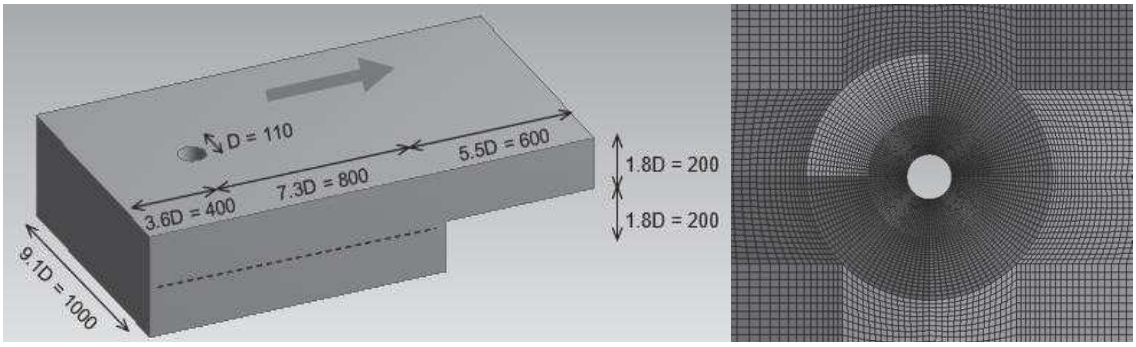

Fig.1 (a) Computational domain and (b) multi-block orthogonal meshing approach.

The default solution methods and under-relaxation factors recommended by Fluent were selected to ensure computational stability for the transient pressure based solver for incompressible flows. This includes the Phase Coupled SIMPLE scheme for pressure-velocity coupling alongside the First Order Upwind spatial discretization schemes and the First Order Implicit transient scheme.

A time step size of 0.001 seconds was implemented to satisfy the conditions for the Courant number < 1 and for the 0.002*D/V* recommendation. The simulations were processed by a Dell Precision T5600 Dual Xeon 3.3 GHz with 16 GB RAM, which required ±48 hours processing time to simulate the 2 hours of physical time required to reach the equilibrium scour depth in the laboratory.

3. RESULTS DISCUSSION

3.1. HYDRODYNAMIC MODELLING

Before the proposed numerical model's ability to simulate bridge pier scour could be evaluated, it had to be established that the flow field could be resolved. Flow visualization was done by means of velocity contour plots, such as those in Fig.2, whereby instantaneous velocities were measured in the laboratory by an Acoustic Doppler Velocimeter (ADV) with a 0.1 *m* grid resolution.

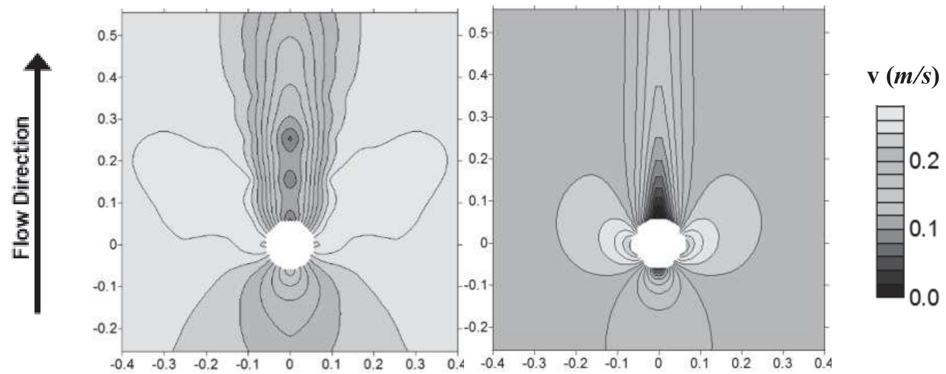

Fig.2 (a) Experimental and (b) numerical velocity profiles for a 0.2 *m/s* approach velocity.

The key discrepancy between the experimental work and numerical model is that the velocities are slightly underestimated by the numerical model, which is characteristic of the k-ε turbulence model (Salaheldin et al., 2004). Nevertheless, the turbulence model performs satisfactorily in reproducing the velocity field by capturing the dominant flow elements in a similar manner, namely the separated flow and the wake downstream of the pier. Fig.3 validates that the primary circulation of the horseshoe vortex was resolved. It was found that the numerical model's ability to resolve the horseshoe vortex is highly dependent on the mesh resolution and that the boundary layer recirculation would only be apparent for a wall unit distance < 1.8 *mm* in the vicinity of the pier.

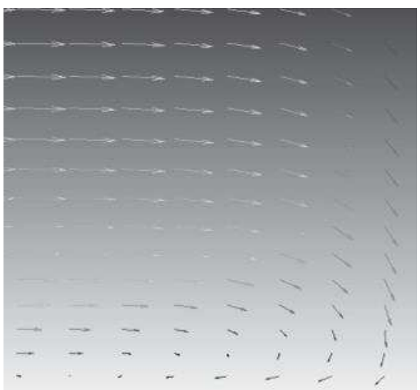

Fig.3 Velocity vectors directly upstream of the pier for an approach velocity of 0.14 *m/s*

The feasibility of a different turbulence model was briefly investigated by evaluating the vortex resolution from the proposed model in relation to those of the standard commercial Fluent code closed with different turbulence models. The k-ε and SST turbulence models yielded similar results, while the computationally intensive RSM model best captured the horseshoe vortex (Salaheldin et al., 2004). Contrary to expectation, the 1st order and 2nd order solutions did not yield discernibly different results. Given that the mesh is adequately fine, the 1st order k-ε turbulence model is sufficiently capable of resolving the primary horseshoe vortex circulation and potentially the subsequent scour calculations (Richardson & Panchang, 1998).

3.2. MORPHODYNAMIC MODELLING

With the final model setup optimized, the proposed morphodynamic model could be evaluated for its ability to simulate bridge pier scour. Fig.4 illustrates the resulting scour hole and the unusual numerical instability encountered by the model. The characteristic horseshoe U-shape has been scoured in the correct region even though the boundary of the conical scour hole is difficult to establish. The model predicted a maximum scour depth of 0.125 *m* comparable to the 0.116 *m* from the experimental work.

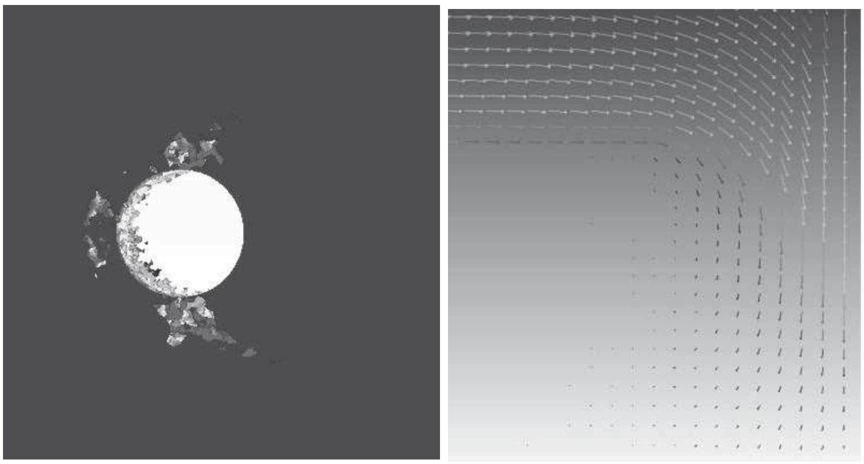

Fig.4 Scour hole simulation instability for an approach velocity of 0.17 *m/s*
(a) Scour hole elevations and (b) velocity vectors directly upstream of the pier.

Fig.5 demonstrates the significance of resolving the horseshoe vortex to accurately simulate the scour hole. The hole initiates at the sides of the pier and emulates the shear stress distribution, whereby the largest shear stress values coincide with regions of separated flow at polar angles > 30° (Roulund et al., 2005). However, if the horseshoe vortex is absent, the scour hole does not propagate towards the front of the pier where the deepest section of the hole should transpire. Of the few existing numerical models that have attempted to simulate bridge pier scour, some produced scour holes, such as those in Fig.5, and did not manage to resolve the crucial horseshoe vortex.

This instability is only present for model setups with the fine grid resolution prescribed by the horseshoe vortex. Sawadogo (2015) stated that “a finer mesh could result in instability at the surface of the bed causing an irregular shape of the scour hole” which is in agreement with the current study. However, the fine grid size required to resolve the horseshoe vortex cannot be sacrificed for the sake of stability because an incorrect scour hole would be predicted. Thus, the cause of the instability must be addressed to yield the proposed numerical model useful to bridge pier scour problems.

It is suspected that the IB method may not be applicable to instances where vortices are present because the only other model to implement this method was that of Khosronejad et al. (2011) who did not manage to resolve the vortices. The inconclusive boundary of the scour hole may be attributed to the velocity vectors which have not been suppressed inside the packed bed. According to Lui & Garcia (2008), the discrepancy in the temporal scales makes hydro-morphodynamic simulations stiff and unstable, implying that the time scaling factor may require further investigation for stabilization.

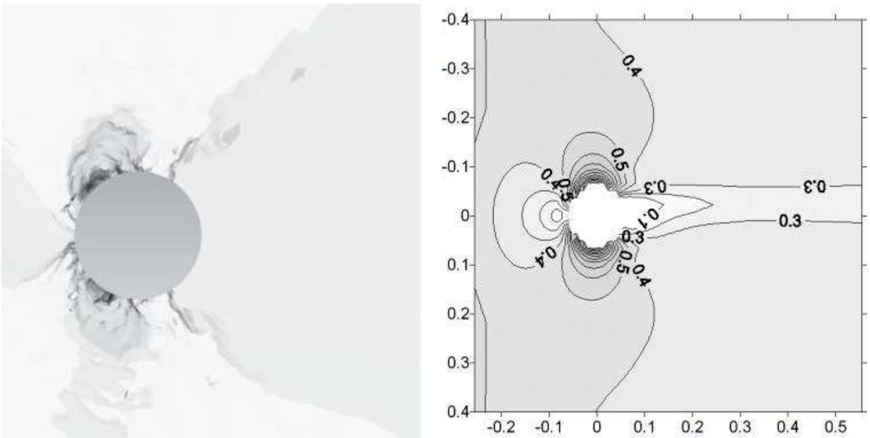

Fig.5 (a) Scour hole in absence of the horseshoe vortex (b) comparable to shear stress distributions.

4. CONCLUSIONS

In short, the hydrodynamic model performed reasonably well in reproducing the velocity profile from the laboratory. However, the model's ability to resolve the complex flow field was constrained by the 1st order k-ε turbulence model because the velocities were underestimated (Salaheldin et al., 2004). Nevertheless, the results are less sensitive to the choice of turbulence model than the geometric mesh representation (Richardson & Panchang, 1998). The numerical simulations are highly sensitive to the mesh resolution and aspect ratio to resolve the vortices and to obtain stability. The sensitivity analysis by Sawadogo (2015) also revealed that the proposed numerical model is sensitive to the mesh resolution, particularly a fine mesh which could cause an unstable irregularly shaped scour hole to be formed.

The proposed model has shown evidence that it has the potential to accurately simulate bridge pier scour as the scour hole position and depth have been verified. However, the cause of the instability in the scour hole shape must be addressed to yield the proposed numerical model useful to bridge pier scour problems. It is recommended that the applicability of the IB method should be investigated and that an improved turbulence model such as the RSM method should be implemented.

ACKNOWLEDGEMENTS

The financial assistance of the National Research Foundation (NRF) towards this research is hereby acknowledged. Opinions expressed and conclusions arrived at, are those of the author and not necessarily to be attributed to the NRF.

REFERENCES

25. Afzul, M.S. 2013. *3D Numerical modelling of sediment transport under current and waves*. MSc Thesis, Department of Civil and Transport Engineering, Norwegian University of Science and Technology.
26. Andreotti, B. 2004. A two-species model of aeolian sand transport. *Journal of Fluid Mechanics*, Volume 510, pp. 47-70.
27. Baykal, C., Sumer, B.M., Fuhrman, D.R., Jacobsen, N.G. and Fredsøe, J. 2015. Numerical investigation of flow and scour around a vertical circular cylindrical pier. *Philosophical Transactions of the Royal Society A,* 373(2033).
28. Chiew, Y.M. 1984. *Local scour at bridge piers*. PhD Thesis, Department of Civil Engineering, University of Auckland.
29. Deshmukh, A.R. and Raikar, R.V. 2014. A clear-water scour around a circular bridge pier under steady flow for different opening ratios. *International Journal of Research in Engineering and Technology*, 3(1), pp. 158-162, January 2014.
30. Khosronejad A, Kang S, Borazjani I, Sotiropoulos F. 2011. Curvilinear Immersed Boundary Method for simulating coupled flow and bed morphodynamic interactions due to sediment transport phenomena. *Advances in Water Resources*, 34(7), pp. 829–843.
31. Liu, X. and García, M.H. 2008. Three-dimensional numerical model with free water surface and mesh deformation for local sediment scour. *Journal of Waterway, Port, Coastal and Ocean Engineering*, 134, pp. 203-217.
32. Richardson, J.E. and Panchang, V.G. 1998. Three-dimensional simulation of scour-inducing flow at bridge piers. *Journal of Hydraulic Engineering*, ASCE, 124(5), pp. 530-540.
33. Rooseboom, A. 2013. SANRAL *Drainage Manual*, 6th Edition, Chapter 8.
34. Roulund, A., Sumer, B.M., Fredsøe, J. and Michelsen, J. 2005. Numerical and experimental investigation of flow and scour around a circular pile. *Journal of Fluid Mechanics*, 524, pp. 351-401.
35. Salaheldin, T.M., Irman, J. and Chaudhry, M.H. 2004. Numerical modelling of three-dimensional flow field around circular piers. *Journal of Hydraulic Engineering*, ASCE, 130(2), pp. 91-100.
36. Sawadogo, O. 2015. *Coupled Fully Three-Dimensional Mathematical Modelling of Sediment Deposition and Erosion in Reservoirs*. PhD dissertaion, Department of Civil Engineering, Stellenbosch University.
37. Schneiderbauer, S. & Pirker, S. 2014. *IB and ALE methods for modelling erosion and sedimentation of a packed bed in the vicinity of an obstacle*. Johannes Kepler University, Austria.
38. Sumer, B.M. 2007. Mathematical modelling of scour: A review. *Journal of Hydraulic Research*, 45(6), pp. 723-735.