International Journal of Environmental Science and Technology (2019) 16:5305–5310
https://doi.org/10.1007/s13762-019-02461-w

ORIGINAL PAPER

Simulation of blockage effects on scouring downstream of box culverts under unsteady flow conditions

M. Günal[1] · **A. Y. Günal**[1] · **K. Osman**[2]

Received: 23 December 2017 / Revised: 19 March 2019 / Accepted: 1 July 2019 / Published online: 8 July 2019

Abstract

Scouring at the downstream of culvert is considered as one of the main reasons of culvert failure. Failure of the culvert structure is a catastrophic phenomenon. Studying the causes of scouring will give an idea about the solution probabilities; the blockage and geometry of culvert have a great influence on the local scour. In this study, simulated results that are obtained from Flow-3D software of box culvert model and formation of scour at the outlet are compared with laboratory results of Sorourian to evaluate the accuracy of the numerical model. Laboratory box culvert was used at two conditions: the non-blocked situation and the partially blocked condition with the blockage ratio of 40%; for this destination, finer sediment was used in Sorourian's study with the median grain size of 0.85 mm, and the flow conditions in the laboratory were unsteady. It means that the flow rates changed over a time period. The performance of the numerical model based on renormalization group is compared with experimental results. The comparison shows that the numerical results have a good agreement with experimental results.

Keywords Box culvert · Scouring · Unsteady flow · Culvert · Scour depth

Introduction

The simple hydraulic structure used to determine flow control and flow rate in any free surface flow is called culvert. Generally, the culvert is used to transport tributary drainage via highway embankments and similar forms of drain crossing structures. We see many different culverts every day but rarely noticed them especially when they work properly. However, everyone will notice them when they fail. Culverts are a short conduit that usually carries storm-water runoff under a railway, roadway or an embankment. In urban and countryside, there are a large number of culverts and culvert size bridges. On the other hand, the culverts are a constricted part of the waterway, and during flood events, it is more likely to be fully or partially blocked by debris. For example, in the city of Wollongong at August 1998, a severe storm occurred, causing extensive flooding. Many real estates broke down due to the loss of human life and service was interrupted. Examining the waterway where the flood occurred at the following day, it showed that most of the waterways, bridges and culvert were blocked by debris, causing damage due to the bridges and culvert blockage. Because of this clogging, the flood level increased upstream of roads and railways, and diverted the flow out of its normal stream into overland flow paths, increasing the degree of flood damage (Rigby et al. 2002).

Scour is the erosion of bank of channel bed due to the bed erosion or water flow at the downstream of hydraulic structures (Ebraim and Heydarenjad 2014). It has been examined that for severe scouring, one of the main causes for the culvert failure is occurred during the time of great flood events, the formation of scour pit that undermines culvert and foundations appeared at the downstream channel bed. Because scouring is recognized as one of the main causes of collapse of the hydraulic structure, prediction of downstream scouring development has a great significant, not only to ensure safety but also to ensure effective long-term maintenance. Therefore, while designing such a structure scouring should be considered. Moody et al. (2005) showed that the erosion occurs when flows exert shear stresses on solid boundaries

---

Editorial responsibiility: Iskender Akkurt.

✉ M. Günal
gunal@gantep.edu.tr

[1] Civil Engineering Department, Gaziantep University, Gaziantep, Turkey

[2] Tishk International University, Erbil, Iraq

over a certain threshold, which depends on temperature, soil type, etc.

In the design of culvert, it is an important consideration to evaluate the degree and extent of local scouring at the downstream of the structure. However, because of the complexity of the problem at high flow rates, it is difficult to perform field measurements in structures that tend to hide the potential significance of the problem, as holes of scouring can be refilled when the flood peaks are gone. A considerable laboratory survey was carried out on scouring in a certain sort of culvert structures, but there is a considerable gap in the data, with an overall understanding. As a result, field data are inconsistent, making it difficult to ascertain the potential scour depth estimates obtained from small laboratory tests. Therefore, most of the design estimates of the scour pit depth and width are based on data obtained from onsite investigations and experiments using physical models (Chen 1970; Smith 1957; Simons and Stevens 1972).

Materials and methods

Case study: physical model

In this chapter, a 3D numerical model of sediment scour was simulated based on an experimental study by Sorourian (2015). The experiment was done in UTS Hydraulics Laboratory of the University of Technology, Sydney. The tests were done in the recirculating concrete flume, and the flume has the following dimensions: 19 m long with a depth and width of 600 mm and 605 mm, respectively. The flume had a constant slope of 0.001. The location of test section was 9 m from the inlet of the flume to ensure a fully developed flow reached the test area. The test section was a 5-m-long, 150-mm-deep sand basin with the same width as the flume. To make an unsteady flow in the culvert, a hydrograph was designed for input discharge of each set of experiment.

Numerical model

Computational fluid dynamics (CFD) is the science of predicting fluid flow and related phenomena by solving the mathematical equations which govern these processes using a numerical process. Also, CFD is a method of simulating a flow process in which standard flow equations such as the Navier–Stokes and continuity equations are capable of displaying turbulent solutions for each computational cell, these equations are the basis for essentially all CFD codes. The Flow-3D program is used in this paper. Flow-3D is a commercial CFD package with special modules intended for hydraulic engineering applications (Williams and Watford 1997; Mehnifard et al. 2015). Despite using a structured orthogonal grid, by the application of the fractional area/volume method (FAVOR), it can model complex geometries, which allows a rectangular computational cell to be partially blocked by an obstacle (Sorourian et al. 2013; Weeks et al. 2013). By the volume-of-fluid (VOF) method, sharp free surface is modeled (Hirt and Nichols 1981). CFD is quickly becoming the dominant technique of flow analysis, especially when used for comparison with experimental work.

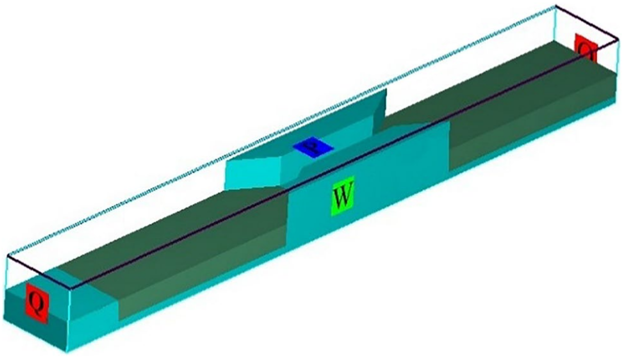

Fig. 1 Configuration of boundary conditions

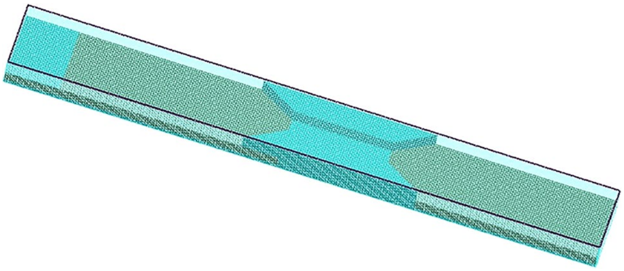

Fig. 2 Representation of the geometry meshing

Flow-3D has many types of boundary condition; each type uses a specific condition of models. Figure 1 shows the *Y* max and *Y* min directions were labeled (*W*) as a wall. The *Z* min boundary of the bottom is designated as the wall boundary, while the *Z* max boundary is designated as the pressure boundary and the zero fluid fraction. The boundary condition in *X* max direction was designated outflow, while in *X* min boundary, it was specified as the volume flow rate. Figure 2 shows the 3D computational domain for the mesh size in *x*, *y* and *z* directions as 0.02 m.

Results and analysis

In this section, the method of the CFD model is based on RNG and is compared with experimental results. It shows that two tests, US1B0 and US1B40, were done with the fine sediment ($d_{50} = 0.85$ mm) with the different blockage ratios. Each set was run in partially blocked and non-blocked conditions. The stepwise hydrograph of input discharge was introduced to Flow-3D as shown in Fig. 3.

The comparison between the results attained from numerical work and experimental results is presented in Table 1. The results show that the difference between CFD and physical model is within the acceptable range (Figs. 4, 5, 6).

Conclusion

Most important results and conclusions from the study are present below:

1. For first set ($d_{50}=0.85$ mm), centerline profile is along horizontal axis; in the flow rate 6 L/s, the observed measurement of scour depth at the first point of non-blocked condition was around 5 mm, and for partially blocked condition, it was about 21 mm, and the percentage of difference between both situations is about 76%, while numerical result for non-blocked situation

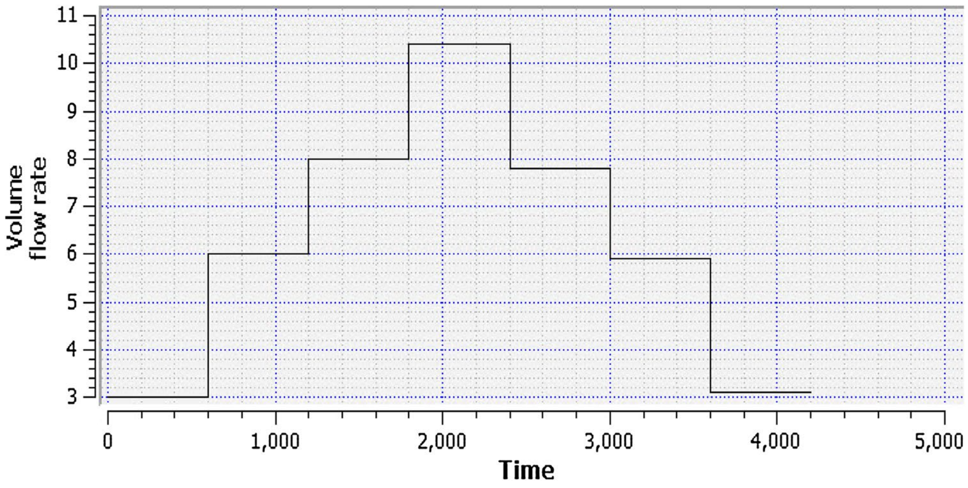

Fig. 3 Stepwise hydrograph for tests in Flow-3D

Table 1 Comparison between results obtained from numerical and physical models

Test	d_{50} (mm)	B (%)	Q (L/s)	t (min)	Experimental result		CFD result (RNG)
					X_{sm} (mm)	d_{sm} (mm)	d_{sm} (mm)
US1B0	0.85	0	3	10	300	11.2	11.3
		0	10.4	40	250	49.3	58.4
		0	3.1	70	300	56	70
US1B40	0.85	40	3	10	150	38.7	59
		40	10.4	40	300	56.4	79
		40	3.1	70	300	56.5	85

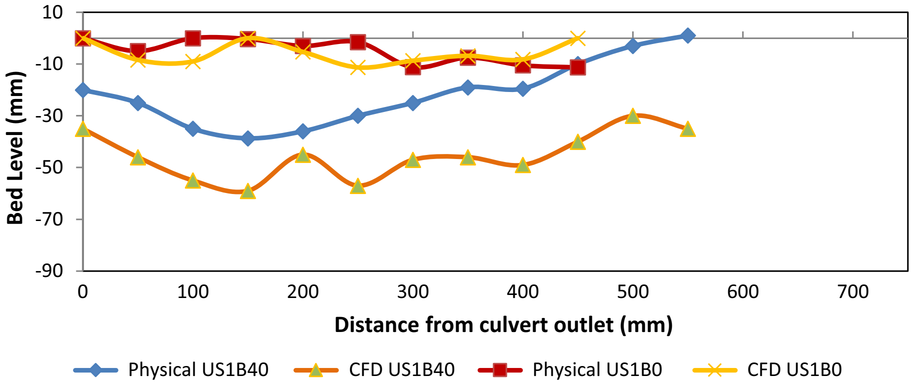

Fig. 4 Flow rate is 3 L/s at 10 min

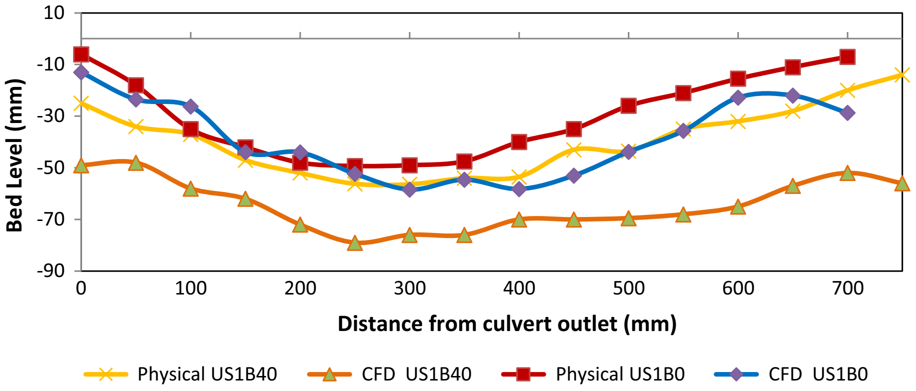

Fig. 5 Flow rate is 10.4 L/s at 40 min

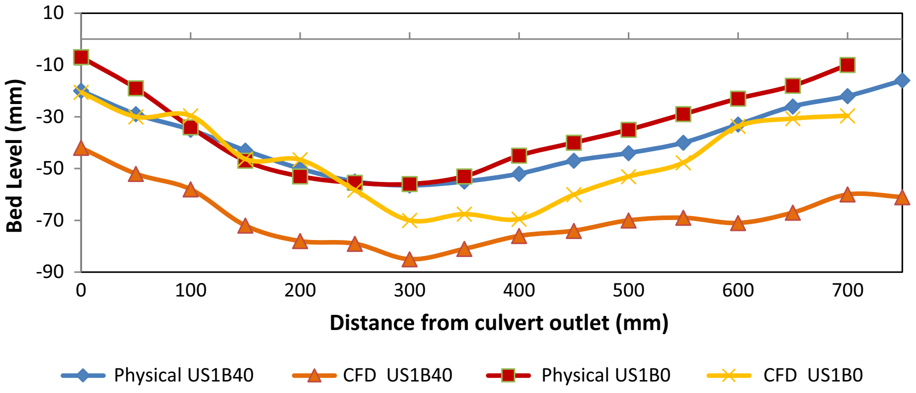

Fig. 6 Flow rate is 3.1 L/s at 70 min

was around 10 mm, and for partially blocked condition, it was around 35 mm, and the percentage of difference between both conditions was about 71%. It is revealed that by increasing the rising limb, in other words by increasing the blockage ratio at culvert inlet, the initiation of scouring happens in the initial steps of the hydrograph, while for the non-blocked condition it still takes a few more steps.

2. For first set ($d_{50}=0.85$ mm), the observed investigation at discharge 6 L/s shows that the maximum scour hole had formed in non-blocked condition around 36%, and for partially blocked situation, it was around 63%; when compared with the Flow-3D model, the maximum scour hole had formed in non-blocked condition around 29%, while the maximum scour hole had formed in partially blocked situation around 74%.
3. For first set ($d_{50}=0.85$ mm), location of maximum scour depth in experimental work for peak flow rate at 10.4 L/s in non-blocked condition is 0.83, and for partially blocked condition, it is 2, while in numerical work for non-blocked condition it is 1.2, and for partially blocked condition, it is 1.7. It is indicating the results of Flow-3D in all tests are close to the experimental results, and proportional change occurs between them.
4. By using empirical equation (RNG) for centerline profile along horizontal axis, the noticeable difference of scour was seen in the third step of the hydrograph, around 50 mm from culvert outlet. For third flow rate 10.7 L/s with fine sediment ($d_{50}=2.0$ mm) at 50 mm after outlet of culvert, the scour depth at this point for non-blocked condition was around 4 mm, and for partially blocked situation, it was about 25 mm, while the result achieved by Flow-3D for non-blocked condition was around 2 mm, and for partially blocked condition, it was about 29 mm.
5. For second set ($d_{50}=2.0$ mm), in experimental study for this test the maximum scour depth of partially blocked condition was around 48.9 mm, while the numerical value for same test by selecting RNG was about 69 mm,

and also for k-ε, it was around 83 mm. This variability indicated by using RNG. The overall results tell us that by using RNG, the value of this empirical equation is closer to k-ε when compared with experimental work.

6. The ability of the Flow-3D program has been confirmed by simulating the existing experimental study and comparing the numerical results with experimental outcomes.
7. In an observed laboratory with numerical results, the maximum scour depth was more in the partially blocked condition in all steps of the hydrograph. Flow-3D is a powerful tool to complement experimental investigation. However, in order to accurately verify the capabilities of the model, long-term simulation is necessary. Special attention is required for mesh resolution and the characteristics of the turbulence model that may affect the results.

Acknowledgements The authors wish to thank all who assisted in conducting this work.

References

Chen YH (1970) Scour at outlets of box culverts. Doctoral dissertation, Colorado State University

Ebraim N, Heydarenjad M (2014) Experimental investigation of the effect of flow angle of attack on the rate of scour around the slotted bridge pier at different level of river bend. IJRASET Int Res Appl Sci Eng Technol 2(12):276–282

Hirt CW, Nichols BD (1981) Volume of fluid (VOF) method for the dynamics of free boundaries. J Comput Phys 39(1):201–225

Mehnifard M, Dalfardi S, Baghdadi H, Seirfar Z (2015) Simulation of local scour caused by submerged horizontal jets with Flow-3D numerical model. Desert 20(1):47–55

Moody JA, Smith D, Ragan W (2005) Critical shear stress for erosion of cohesive soils subjected to temperatures typical of wildfires. J Geophys Res Earth Surf 110(F01004):1–13

Rigby EH, Boyd MJ, Roso S, Silveri P, Davis A (2002) Causes and effects of culvert blockage during large storms. In: Strecker EW, Huber WC (eds) 9th international conference on urban drainage. Portland, Oregon, USA, September 8–13, pp 1–16

Simons DG, Stevens MA (1972) Scour control in rock basins at culvert outlets. In: Shen HW (ed) River mechanics, vol II, Chap 24

Smith GL (1957) An analysis of scour below culvert outlets. Doctoral dissertation, Colorado State University

Sorourian S (2015) Study of blockage effects on scouring pattern downstream of box culverts. Doctoral dissertation

Sorourian S, Keshavarzi A, Ball J, Samali B (2013) Study of blockage effect on scouring pattern downstream of a box culvert under unsteady flow. Aust J Water Resour. https://doi.org/10.7158/w13-031.2014.18.2

Weeks B, Barthelmess A, Rigby E, Witheridge G, O'Loughlin G (2013) Project 11: blockage of hydraulic structures. Stage 2 report. Engineers Australia, Barton ACT, Australia

Williams RJ, Watford FA (1997) Identification of structures restricting tidal flow in New South Wales, Australia. Wetl Ecol Manag 5(1):87–97