Journal of Computational and Applied Mathematics 280 (2015) 68–79

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

journal homepage: www.elsevier.com/locate/cam

Numerical modeling of scour depth at side piers of the bridge

Mohammad Ehteram [a,*], Amin Mahdavi Meymand [b]

[a] *Semnan University, Tehran, Iran*
[b] *Chamran university, Iran*

ARTICLE INFO

Article history:
Received 6 August 2014
Received in revised form 16 November 2014

Keywords:
Numerical models
Sediment transmission
Scour

ABSTRACT

Computational power and development of numerical models are new windows for simulation about flow and sediment transmission. The aim of this article is related to Numerical modeling of three-dimensional flow and sediment at the narrowing location of bridge. We selected the SSIM2 model because this model has high ability for numerical simulation. We used this model in the erodible and rigid bed. We compare the results with experimental data. Comparison shows model with high ability for simulation about velocity and level of water but when the value of perturbation is high to have the high error for simulation. The numerical model is used for scour depth and bed variations. This model predicts the bed variation and scour depth and it shows the maximum of scour depth but the depth of scour hole is less about fifteen percent than experimental results. Based on all the results, this method suggests a good tool about river engineering problems.

1. Introduction and history

Knowledge of scour hole-dimensions can be used by bridge engineers to determine the extent of countermeasures needed to prevent scour at bridge piers. Studies of local scour were carried out until recently using numerical and physical models. Some of the studies emphasize scour around large obstacle. (Wang, 2006; [1,2]). The geometry of scour hold depends on many variables. Sediment size, inflow of sediments, size distribution and velocity in the river, geometry of the structure and geometry of the bed around the structure are all parameters that will affect the shape and magnitude of the scour hole (Kuipers and Vreugdenhil, 1973; [3–5]).

Most experimental studies of local scour have been carried out under idealized conditions, i.e. flat bed, uniform upstream water flow and so forth. Such condition can make adaption to prototype conditions difficult if the river bed has a complex geometry. A more satisfactory approach is to calculate the water and sediment flow with a three dimensional numerical model. The geometry of the bed and the obstacle can then be calculated specifically for each case [6,7], used averaged two dimensional and Turbulence model. The three dimensional models are used for the more complex when we secondary flows and vertical velocities in the depth. In general, various types of scouring around the piers can be divided in to three main categories: general scour, while the amount of sediment arrived to the river or a part of it, is less than the exit sediment, the erosion scours in the floor or walls of the river. In this effect, the floor of the river is dished gradually that it is called kafkani. Kafkani can enumerate as the effective factors to reduce the bed level around the bridge piers in long time period [8]. Scouring in effect of narrowing section: this type of scouring mainly occurs in the vicinity of the piers distances. In the scouring in the effect of narrowing flow section, the bed level and flow rate will increased and thus the potential for sediment transport will also increase from the river bed and near the bridge pier [9]. Local scouring: in general, the hydraulic structure and

* Corresponding author. Tel.: +09352272414.
E-mail address: Eh.mohammad@yahoo.com (M. Ehteram).

http://dx.doi.org/10.1016/j.cam.2014.11.039
0377-0427/

bridge piers may change the flow pattern and create turbulence flows near the relevant structures and finally create a hole in the place of structures. This kind of erosion is severely in the flooding conditions and is one of main factors to occur the damaged hydraulic structures and bridges.

Many researchers due to the laboratory or field data and wide studies on the mechanism of scouring have provided many relationships to estimate and predict scouring depth. Each of these relationships is based on a number of parameters affecting the scouring. Some of the researchers consider the effect on one or two parameters in their relations and others use more parameters in their relations.

There are different numerical models. The SSIIM model is three-dimensional software for simulation of water and sediment movements and it was developed by Nilz Oulsen in the hydraulic engineering department and environment of Norway science technology. The three dimensional CFD model is based on the Finite Volume method and solves Navier–Stokes turbulence model on the standard turbulence model k–e [10]. SSIM model is the numerical software with applications in the field of river engineering, environmental hydraulics and sediment that the primary purpose of this software structure is to simulate the movement of sediment in the river and channel geometry. The main advantage of SSIM model comparing CFD software is structure is an ability to analyze complex geometries to model sediment transport in live bed.

2. The governing equation

The three dimensional model is solved by equation reported below [10]. The Navier–Stokes equations for non-compressible and constant density flow can be modeled as:

$$\frac{\partial r}{\partial t} + \frac{\partial (ru)}{\partial x} = 0 \tag{1}$$

$$r\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_j}{\partial x}\right) = B_i - \frac{\partial P}{\partial x} + (m + m_t)\frac{\partial^2 u_i}{\partial x_j \partial x_j}. \tag{2}$$

In this equation, r: density, u_i: velocity in the direction of x, B_i: solid forces, m dynamic viscosity, m_t: turbulence viscosity.

The value of turbulence viscosity is unknown and it is computed by turbulence model. The SSIM software uses the standard k–e turbulence model and RNG model. The RNG model has high ability about complex phenomenon simulation. In the k–e model, the value of m_t is computed by the following equation:

$$m_t = C_m r \frac{k^2}{e}. \tag{3}$$

C_m is an experimental coefficient to be considered as 0.09. K is turbulence kinetic energy and e is mortality coefficient. The values of k and e are not same in the different models. In the RNG model, we used the following equation:

$$r\frac{\partial k}{\partial t} + ru_j k_j = \left(m + \frac{m_t}{s_k} k_j\right)_j + 2m_t S_{ij} S_{ij} - re \tag{4}$$

$$r\frac{\partial e}{st} + ru_j e_j = \left(m + \frac{m_t}{s_s} e_j\right)_j + C_1 \frac{e}{k} 2m S_{ij} S_{ij} - C_2 r \frac{e^2}{K} - \frac{C_m h^3 \left(1 - \frac{h}{h_0}\right)}{1 + bh^3} \frac{e^3}{k} \tag{5}$$

$$h = S\frac{k}{e} \qquad S = \sqrt{2S_{ij}S_{ij}} \qquad S_{ij} = \frac{1}{2}\left(m_{ij} + m_{ij}\right).$$

C_1 and C_2: Experimental coefficients.
S_{ij}: Stress tensor.
s_k: Prantel number
s_s: Turbulence eshmit

The RNG model simulates flow separation and rotary flows because it has many terms in the e equation. (Yakhot, 1992) divided the sediments to two groups to include bed load and suspended load. The density of suspended load is computed by diffusion-movement equation.

$$\frac{\partial c}{\partial t} + u_i \frac{\partial c}{\partial x} + w \frac{\partial c}{\partial z} = \frac{\partial}{\partial x_i}\left(\Gamma \frac{\partial c}{x_i}\right) \tag{6}$$

c: Density of sediment
w: The fall velocity of sediments
Γ: Diffusion coefficient