CFD Simulation of Open Channel Flooding Flows and Scouring Around Bridge Structures

B. D. ADHIKARY, P. MAJUMDAR, and M. KOSTIC
Department of Mechanical Engineering
Northern Illinois University
DeKalb, IL 60115
U.S.A.
Email: kostic@niu.edu; Web: http://www.kostic.niu.edu

Abstract: - Simulation of scour-pit formations under the bridge decks and around the bridge piers, due to sediment suspension and transport caused by flooding and pressure flow conditions, is of significant interest nowadays to computational fluid dynamics (CFD) and hydraulic researchers. From CFD perspective, analysis of bridge under various flooding conditions and impact of the drag and lift forces on it, are necessary to determine bridge stability. This study is focused on the simulation of open channel turbulent flows over inundated bridge deck and evolution of scour pit under various flooding conditions. Solutions for flow field and turbulence, using a representative bed roughness value, are based on Reynolds Averaged Navier-Stokes (RANS) equations, and turbulence closure model using a commercial CFD code. An iterative computational methodology is developed for the evolution of scour-pit profile using the single-phase slip-flat-top surface with (re)moving boundary formulation, based on an empirical correlation for critical shear stress to characterize initiation of sediment removal. The computational model has demonstrated stable and converged solution for the evolution of the scour-pit shape and size using a constant critical shear stress value. The success of the simulation model can be substantially improved with development of a functional critical shear stress by taking into account solid-fluid interactions, scour bed profile and slope, as well as more extensive validation with experimental data.

Key-Words: - Bridge structures, CFD simulation, Computational fluid dynamics, Flooding flows, Flow scouring, Turbulence modeling.

1. Introduction

Design and construction of bridges needs reevaluation as it has raised considerable safety concern in recent times. Some recent bridge failures increase this apprehension to such a level that extensive research work is going on nation-wide to prevent these kinds of damages. Due to the flood or increased water flow, the downward movements of the bed or an increased drag and lift forces on the bridge decks and piers are some very common reasons for bridge failures. Under the flooded condition, formation of scour-pit at the water channel bottom weakens the strength of the bridge piers, which eventually damages the bridge. The accurate estimation of this kind of scour depends on the flow field and turbulence conditions, sediment bed forms, sediment transport and suspension phenomena, along with the interaction between the sediments and the water. Analysis of scour-pit development under the bridge decks and around the bridge piers due to sediment transport caused by flooding and pressure flow conditions, is important for the design, construction and maintenance of bridges.

Smith [1] presented the modeling technique of a contraction scour around a horizontal stationary cylinder by using two-phase volume-of-fraction (VOF) methodology available in FLOW-3D commercial CFD code. Guo [2] adapted the Shields-Rouse equation for the calculation of critical shear stress based on the experimental results and its application to sediment transport. This equation provides the way to correlate the critical shear stress for the initiation of sediment movement to the dimensionless sediment diameter. Singh [3] developed an integrated 2-D numerical sediment transport model to simulate the sediment transport in water medium. Both the suspended load and bed load have been taken into account in this study. Camenen *et al.* [4] proposed a new empirical correlation between the roughness height and some of the important hydrodynamic and sediment parameters for plane beds, under steady flow conditions. Zhao and Fernando [5] performed CFD simulation using *Fluent* commercial code for the evolution of scour around pipelines, using *Eulerian-Eulerian* coupled two-phase model for fluid and solid phases. They investigated the effect of different sediment transport modes, such as bed-load, suspended-load, and laminated-load, on the development of scour.

The objective of this study is to develop a simulation model to predict the shape and size of a scour-pit under the inundated bridge deck. The scour evolution under a flooded bridge deck due to the accelerated turbulent flow has been studied and a computational methodology has been developed using the single-phase (re)moving boundary formulation for open channel flows over flooded bridge deck. This is done based on the computational fluid dynamics analysis of the flow fields around the bridge deck and computation of shear stress over the stream scour bed contour.

2. Description of the Physical Problem

In this section, a description of the physical problem involving open channel flow over an inundated bridge deck, along with all physical parameters of importance and range of operating conditions under consideration, are presented. A schematic of the flow channel configuration for flow over a six-girder bridge and the bridge dimensions are shown in Fig. 1.

The characteristic dimensions, shown in Fig. 1 are as follows: h_u = water height from free-surface to the bottom of the channel or channel bed, h_b = water depth from bridge-bottom to channel-bottom and s = bridge height.

This study is primarily concerned with a two-dimensional (2-D) model and neglecting the presence of openings in the bridge-deck side walls.

Dimensionless parameters, used to characterize open channel flow over an inundated bridge deck, are described below.

Froude Number (Fr): The Froude number is an important parameter that governs characteristics of the open channel flow. Froude number is defined as the ratio of flow velocity to the velocity of free-surface wave at a particular location. It could be considered as a ratio of inertial forces to the gravity forces, and is expressed as:

$$Fr = \frac{V_u}{\sqrt{gL_c}} \qquad (1)$$

Where, V_u = upstream velocity and L_c = characteristics length for the channel flow.

Reynolds Number (Re): The Reynolds number is also considered as a very important parameter for open channel flow analysis which defines the level of flow inertia and turbulence, and is the ratio of inertia forces to the viscous forces, given as:

$$\mathrm{Re} = \frac{V_u D_h}{\nu} \qquad (2)$$

Where, V_u = upstream velocity; D_h = hydraulic diameter; ν = kinematic viscosity of the water.

In calculating the Reynolds number based on the upstream flow, water height (h_u) is considered as the hydraulic radius, resulting in hydraulic diameter D_h = $4h_u$. For open channel flows the Re numbers are usually very high and flows are turbulent.

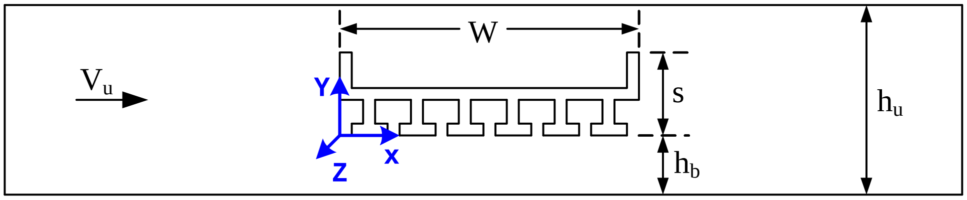


Fig. 1: Schematic and characteristic dimensions of the computational domain: water channel with bridge deck model

Inundation Ratio (h^*): The dimensionless inundation ratio, $h*$, is an indication of how much bridge is flooded, and it is defined as:

$$h* = \frac{h_u - h_b}{s} \qquad (3)$$

The configuration of $h* = 1$ denotes the critically- or just-immersed bridge deck, whereas $h* > 1$ denotes a fully-submerged bridge deck.

3. Simulation Model for Scour Analysis

Scour failures are primarily the result of the grounds, on which the foundation sits, being washed away. Under flooding conditions, the flow under the bridge gets accelerated and induces excessively high stresses at the river bed under the bridge. Such conditions enhance scour rates which may lead to catastrophic failure of the bridge. The stress force around the bridge, which is exerted by the accelerated flow under the bridge, is also enhancing the suspension and transport of the bed sediment and downward movement of the channel bed material.

Simulating the scour formation process in the sediment bottom deck due to accelerated turbulent flow under the bridge, is very complex due to the presence of multiple physical processes at the fluid-solid particle interface at the bottom. This includes the turbulent flow structure and wake formations, sediment particle suspension and deposition transport, fluid particle interactions, particle-particle interactions, and the coupling between the evolving scour profile and the turbulent flow field.

A number of modeling options are possible and have been investigated by different researchers as discussed above. While two-phase modeling options, such as, the VOF and Eulerian two-phase models, seem more accurate in terms of the physics involved, they are computationally more complex and impose significant challenges for modeling interfacial interactions and constitutive properties of the scoured materials.

In this study, a simplified model is proposed considering the fact that the scouring is mainly due to induced local shearing stresses at the sediment surface. The computational methodology is based on the CFD analysis of the flow field around the bridge deck and computation of shear stress over the scouring bed contour. The water computational domain is extended, by (re)moving layer-by-layer of the solid-material at the scouring boundary, where local shearing stresses are higher than the critical shearing stress, the latter representing the solid material resistance to destruction/scouring. Appropriate iterative procedure, one layer at a time, may lead to a realistic scour profile assuming that the critical shearing stress can be properly assessed in terms of experimental correlations. Use of such a correlation significantly simplifies the mathematical and computational difficulties associated with such a coupled, two-phase fluid-solid transport process.

3.1. Computational Fluid Dynamic Model:

The CFD analysis of open channel flow is based on using single-phase turbulent flow model with slip-flat-top methodology and using STAR-CD commercial CFD code. Options available for analyzing turbulent open channel flows are either a time-averaged approach using Reynolds Averaged Navier-Stokes (RANS) equations in turbulence closure models, or a space-averaged approach using Large Eddy Simulations (LES), the latter requires larger amount of computational time. In this study, the simulation is restricted to the RANS turbulence models. An accurate prediction of pressure scour formation, among others, depends on the appropriate selection of turbulence model. The turbulence closure models available in STAR-CD code include six different two-equation, $k-\varepsilon$ models, four two-equation, $k-\omega$ turbulence models, and three Reynolds Stress models. In an initial evaluation study of turbulent open channel flow with flat channel bottom surface, a number of turbulence models including the two-equation $k-\varepsilon$ turbulence models and $k-\omega$ turbulence models, and Reynolds Stress Model (RSM) were evaluated in terms of stable converging solutions and comparing the results with the experimental data for resulting lift and drag forces over the bridge deck. The high Reynolds number $k-\varepsilon$ model was identified as quite appropriate in terms of accuracy and computational requirements by comparing simulation results with the laboratory data obtained from the force balance measurements of lift and drag coefficients. Initial development of scour formation methodology is, therefore, carried out using high Reynolds number turbulence model along with wall-function treatment. However, for refined simulation model for scour evolution other turbulence models and Large Eddy Simulation (LES) model will be explored in the future.

Inlet velocity, thus, the mass flow-rate, is assigned as a boundary condition, whereas the so called standard outlet boundary condition is assumed. He AMG solver is used here in an iterative way to calculate the minimum mass, velocity and pressure residuals. As only 2-D simulations have been performed so far in this research, requiring only one cell thickness along the third Z-direction, the front and rear wall of the computational domain have been taken as the symmetry boundary walls, since the STAR-CD code is based on the 3-D simulation concept and does not have special 2-D mode. For all the simulations presented in this study, the inlet is defined as the velocity inlet; the outlet condition has been set as the standard outlet which indicates the constant pressure gradient in flow direction at the outlet. The top wall is defined as the slip-wall to simulate the free-surface open-channel flow, whereas the bottom wall is the no-slip boundary.

In this study an iterative procedure is used to establish the appropriate fully-developed inlet velocity profile and turbulence conditions. The iterative procedure starts with an average uniform velocity and specified turbulence conditions at the inlet and ends with determination of the (fully) established exit velocity profile and turbulence conditions. The process is repeated until the solution is converged. These exit profiles are then used as inlet conditions in the subsequent iterations.

3.2. Computational Methodology for the Evolution of Scour Profile

As stated above, the scouring computational methodology is based on the fact that the scouring is due to induced local shearing stresses at the sediment surface, thus requiring the CFD analysis of the flow field around the bridge deck and computation of shear stress over the bed contour. An iterative computational methodology is developed for evolution of scour-pit profile using the single-phase turbulent flow modeling (based on a representative bed roughness value) with slip-flat-top surface, by (re)moving boundary formulation and use of an empirical correlation for critical shear stress to characterize initiation of sediment removal. Use of such a correlation significantly simplifies the mathematical and computational difficulties associated with such a two-phase fluid-solid coupled transport process. One of the main challenges is to develop the apparent critical-scouring shearing stress correlations to account for all the complexities of solid material resistance to flow scouring. Additional consideration must me given to the characterization of the scour-bed surface roughness, which depends considerably on the type of bed forms and flow conditions.

3.3. Empirical Correlation for Critical Shear Stress at Channel Bottom:

In general, the apparent critical shearing stress is not constant, but a local function of other flow-and-material properties, like fluid-solid surface interactions, downgrade and upgrade slope scouring, where gravity has opposite contribution to grain removal and deposition, for example. Additional consideration must be given to the characterization of the scour-bed surface roughness, as stated above. However, since such a correlation depends on number of key parameters, such as, fluid and sediment particle’s type and size, type of obstacles, and flow condition, it has to be determined by experiments for a specific application [6].

A comprehensive review of various approaches to find out the critical shear stress at the bottom wall of an open channel flow, is given by Singh [3]. One of the critical shear stress correlations for sediment transport used in this simulation study is given by Guo [2], and is based on the Shields-Rouse equation with fitted parameters to match existing experimental data, i.e.:

$$\frac{\tau_c}{(\rho_s-\rho)gd_{50}}=\frac{0.23}{d_*}+0.054\left[1-\exp\left(-\frac{d_*^{0.85}}{23}\right)\right] \tag{4}$$

Where, ρ_s = density of sediment; g = acceleration due to gravity; d_{50} = median size of the bed sediment; $u_c^*=\sqrt{\frac{\tau_c}{\rho}}$ critical velocity; and;

$$d_*=d_{50}\left[\frac{(\rho_s/\rho-1)g}{\nu^2}\right]^{1/3} = \text{dimensionless diameter} \tag{5}$$

Our computational methodology is developed and implemented based on a constant critical shear stress value which is estimated using the above correlation for shear stress value over a bottom boundary surface.

3.4. Representative bed Roughness

Computational solution of the flow field, the local shear stresses, as well as the computed scour pit at the bed surface, are interrelated and strongly influenced by the selection of the representative surface roughness value, the latter used as an input to the CFD simulation. The bed roughness value can vary considerably depending on the type and form of the bed material. Sediment size and type as well as the sediment bed grain, influence the sediment transport rate and the induced stresses at the bed surface. A list of such functional relations and detailed discussion on the development of such relations are given by Camenen et al. [4]. A number of correlation for roughness is evaluated for sand using different formula considering $d_{50} = 1\ mm$. This evaluation study shows that the roughness value can vary by a factor of 1 to 5 depending on the empirical formula. In order to improve the accuracy of the single-phase simulation model, it is necessary to refine the selection of the apparent bed roughness value, by comparison of the computed shear stress distribution at bed scour surface with relevant experimental data. Further numerical parametric studies have to be performed with varying representative bed roughness values.

3.5. Implementation of Scour Methodology in STAR-CD software

The present methodology for the evolution of scouring has been presented elsewhere [7]. The methodology for the evolution of scour-pit shape has been implemented in STAR-CD commercial CFD code. A C-computer code is written to generate a modified script file by comparing wall shear stress values with the critical shear stress. A bottom wall layer is incrementally re-moved at locations where the local shear stress values exceeded the critical shear stress, to create new scour bottom geometry. After each iteration step, the scour profile depth at any location is increased or decreased accordingly.

Flow calculation is repeated with this new, modified geometry that results in a new shear stress distribution at the modified bottom surface. The iterative process starts with the flat-bottom geometry, extracting the results, comparing the shear stress at bottom with critical shear stress, and moving the bottom further down where shear stresses exceed the critical value, thus generating a new geometry of the computational domain to be used in the next iteration. After the bed contour is modified, the grid is re-meshed, and the simulation repeated to determine a new bed shear stress distribution. This iteration process of flow calculation and creation of the new scour bottom by removing the bottom surface layer is continued until sub-critical shear stresses are achieved everywhere. The iterative single-phase-flow scour-simulation methodology, in this initial stage, has involved some human intervention between iterations for full implementation in the STAR-CD commercial code, thus substantially prolonging over-all computation time. A full automation process is in final stage of development in order to fully automate the iteration process.

4. Results and Discussions

The geometry and simulation parameters used for initial scour evolution are shown in Fig. 2. In order to simplify simulation and shorten the computational time, the six-girder bridge geometry is replaced with a rectangular block with the same aspect ratio as of the six-girder bridge. The operating conditions and other geometric parameters correspond to the following: upstream velocity $V_u = 0.41\ m/s$, channel size: h_u = 25 *cm*, and h_b = 11.5 *cm*. The critical shear stress value is estimated as 0.58 *Pa* based on 1 *mm* sand diameter and assumed as constant over the scour-pit surface. The bed roughness is assumed as constant having $\frac{k_s}{d_{50}} = 1.0$, which leads to 1 *mm* roughness for median sediment diameter of d_{50} = 1 *mm*. The stair-case-like scour-pit profile obtained after each iteration

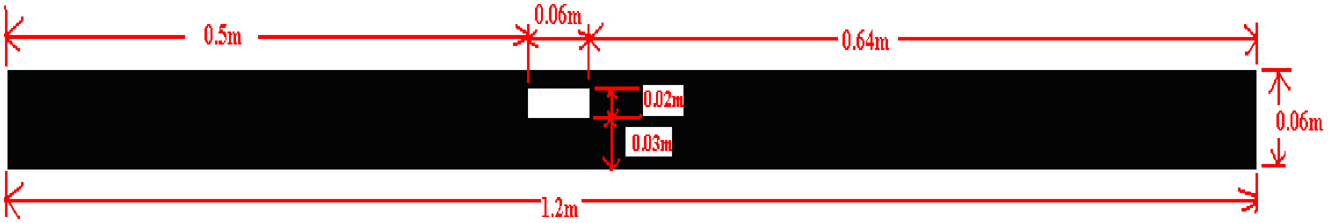


Fig. 2: Computational Domain for Scour Analysis

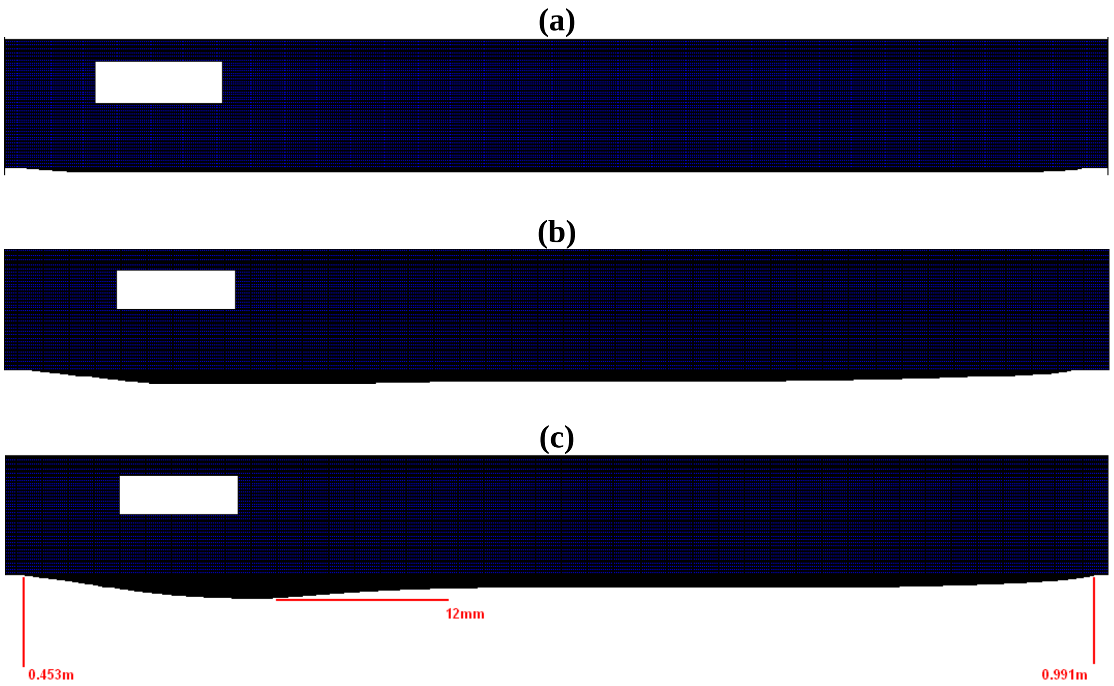
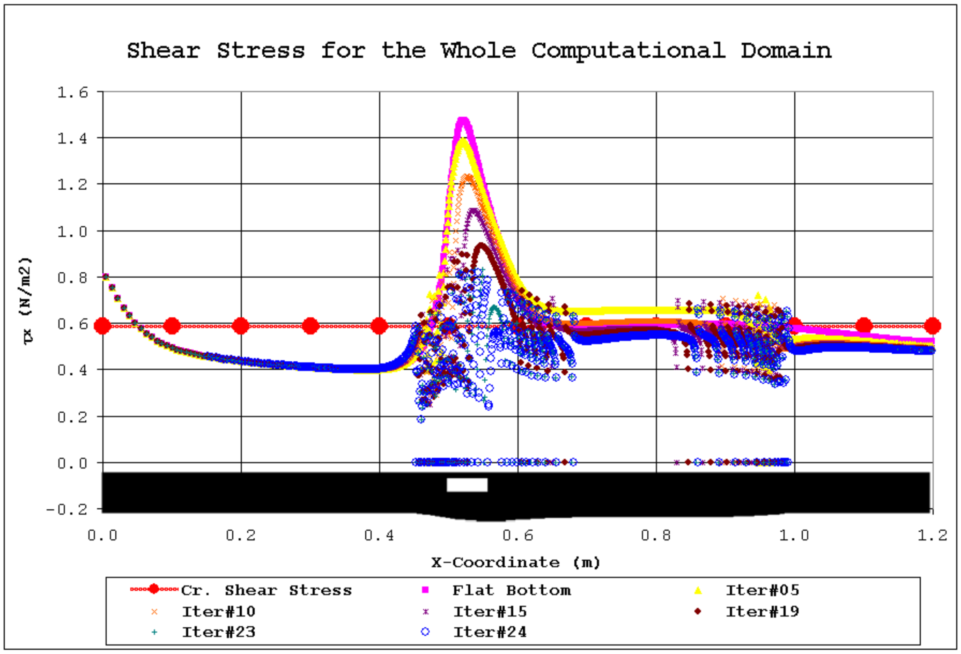


Fig. 3: Scour formation: white box represents flooded bridge, while black area is fluid computational domain with scouring at the bottom, layer-by-layer.
From top-down: after (a) 5 iterations, (b) 10 iterations, and (c) 24 iterations.

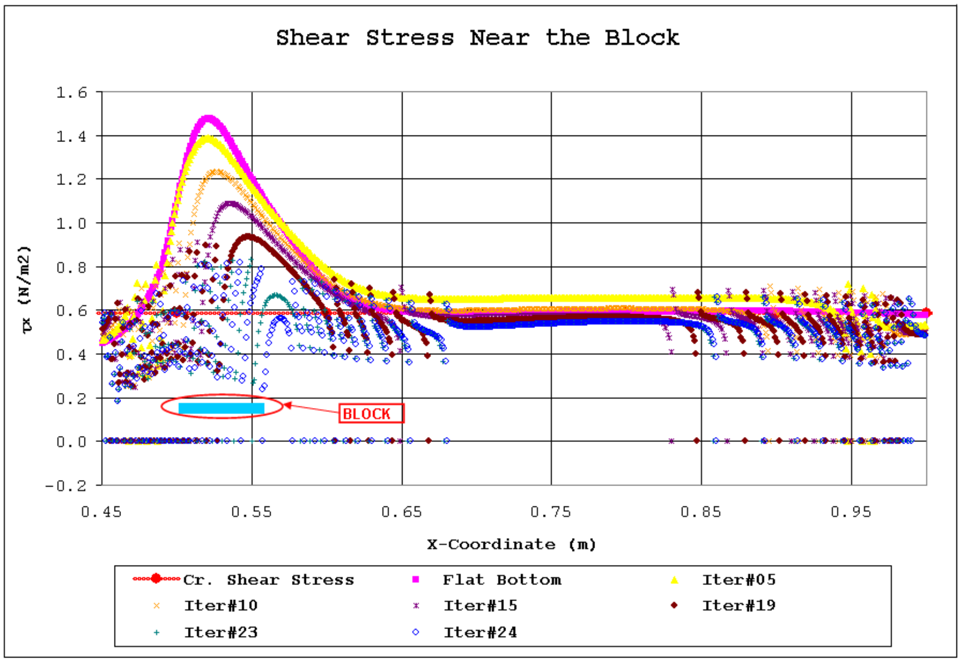
is represented by a smoother scoured bottom geometry using a spline interpolation function. The new bed geometry contour is re-meshed, and the simulation is repeated to determine a new bed shear stress distribution.

Initial sets of results are obtained by removing the bottom boundary with layers of 2 *mm* thickness (4-cell thick layer with a cell size of 0.5 *mm*). Figure 3 shows iterative evolution of scour-pit profile at several selected iterations, while Fig. 4 shows the corresponding shear stress distribution, including the scour-pit surface.

Results show continuous decrease in shear stress distribution with the evolving scour bottom surface with increasing number of iterations. The scour shape and size reach a steady equilibrium state by about 24 iterations. As shown, the local shear stress values at the scoured surface are reduced progressively as the scour-pit deepens with iterations and reaches below the critical shear stress value of 0.58 N/m^2. In the scour-pit surface regions near the sharp corners edges, a considerable jumps and discontinuities can be observed. This is primarily caused by sharp corners of the stair-case-like scour-bottom geometry. This is not a physical phenomenon, but rather, they are purely computational artifacts and may lead to the misleading conclusion that scour profile and maximum scour depth have not reached an equilibrium state. This discrepancy can be eliminated by making the scoured bottom smoother, using the spline interpolation function to replace the stair-case scour-pit profile obtained after each iteration step. This procedure is currently being implemented. Other important factors, that may cause errors in the simulation results, are the selection of bed roughness values and the assumption of constant critical shear stress values at the scoured surface. Further work aims to improve the simulation model by evaluating results with additional experimental data and implementing more realistic representative bed roughness.



(a)



(b)

Fig. 4: Shear stress distribution along bottom boundary at different iterations: (a) along the bottom of whole domain, and (b) along the scoured bottom.

5. Conclusions

An iterative computational methodology is developed for the evolution of scour-pit profile using the single-phase turbulent flow model with (re)moving boundary formulation by means of an empirical correlation for critical shear stress and using a representative bed roughness value. The simulation model is based on the computational fluid dynamics solution for velocity, stresses and turbulence flow fields, and resulting stresses around the bridge deck and flow bottom surface. The single-phase, slip-flat-top steady-state model was chosen over the transient VOF two-phase model, considering improved stability and convergence, and shorter computational time, while achieving similar accuracy of obtained results. The computational model demonstrated stable and converged solutions for the evolution of the scour-pit shape and size using a constant, empirical critical-shear-stress value. Use of the constant critical shear stress value over the entire scour pit rather than the functional critical shear stress profile is one of the main deficiencies of this methodology. Additionally, there is considerable uncertainty in the use of the representative bed roughness value. Further numerical studies need to be performed to optimize critical simulation parameters while validating the results with available experimental data.

Acknowledgments:

The authors like to acknowledge support by Dean Promod Vohra, College of Engineering and Engineering Technology of Northern Illinois University (NIU), and Dr. David P. Weber of Argonne National Laboratory (ANL); and especially the contributions by Dr. Tanju Sofu, and Dr. Steven A. Lottes of ANL, as well as financial support by U.S. Department of Transportation (USDOT) and computational support by ANL's Transportation Research and Analysis Computing Center (TRACC).

References:

[1] Smith, Heather D., *Modeling the flow and scour around an immovable cylinder,* M.S. Thesis, Ohio State University, 2004.

[2] Guo, J., Hunter Rouse and Shields Diagram, *Advances in Hydraulic and Water Engineering*, Vol. 2, 2002, pp. 1096-1098.

[3] Singh, Vikas, *Two dimensional sediment transport model using parallel computers,* M.S. Thesis, Banaras Hindu University, 2005.

[4] Camenen, Benoit, Bayram, Atilla, Larson, Magnus, Equivalent Roughness Height for Plane Bed under Steady Flow, *Journal of Hydraulic Engineering*, Vol. 132, No. 11, November 1, 2006.

[5] Zhao, Zhihe, Fernedo, H. J. S., Numerical Simulation of Scour around Pipelines Using an Eulerian-Eulerian Coupled Two-phase Model, *Journal of Fluid Mechanics*, Vol. 7, 2007, pp. 121-142.

[6] Van Rijn, L. C., Sediment transport I: Bed load transport, *Journal of Hydraulic Engineering, V*ol. 110, No. 10, 1984, pp. 1431-1456.

[7] Adhikary, B. D., Flow *and Pressure Scour Analysis of an Open Channel Flow Having an Inundated Bridge Deck Under Various Flooding Condition*, M. S. Thesis, Northern Illinois University, 2008.