Proceedings of the ASME 2011 International Mechanical Engineering Congress & Exposition
IMECE2011
November 11-17, 2011, Denver, Colorado, USA

IMECE2011-65529

THREE-DIMENSIONAL SCOURING ANALYSIS FOR OPEN CHANNEL PRESSURE FLOW SCOUR UNDER FLOODED BRIDGE DECKS

Bhaskar Rao Tulimilli
Northern Illinois University
DeKalb, IL, U.S.A.

Steven A. Lottes
Argonne National Laboratory
Argonne, IL, U.S.A.

Pradip Majumdar
Northern Illinois University
DeKalb, IL, U.S.A.

Milivoje Kostic
Northern Illinois University
DeKalb, IL, U.S.A.

ABSTRACT

A three-dimensional stream bed scour modeling methodology was developed using well-benchmarked commercial Computational Fluid Dynamics (CFD) software to compute the bed shear stress distribution used to calculate bed displacements and to re-mesh the computational domain as the bed is displaced. This study extends a previously developed two-dimensional iterative scouring procedure to predict the final shape and size of the scour-hole under pressure-scour flow conditions for flooded bridge decks using commercial CFD software. The current approach uses single phase flow models with an assumed flat water surface using a symmetric slip top boundary to simulate a free-surface flow condition, quasi-steady simulation to obtain the bed shear, and a moving boundary formulation based on an empirical correlation for critical shear stress to iteratively deform the bed under supercritical shear conditions until an equilibrium scour condition is obtained. The model solves the flow field using Reynolds Averaged Navier-Stokes (RANS) equations and the high Reynolds number *k*–epsilon turbulence model using the commercial CFD software STAR-CD. A Bash script was developed to use a Python script to compute bed displacements from the computed shear stress distribution and generate a STAR-CD processor command file to displace the bed followed by a step using the STAR-CCM+ software to remesh the domain as the bed is displaced and bed shear distribution is recomputed in an iterative procedure until the equilibrium bed contour is reached. Simulations were performed for different inundation ratios and for mean sand diameters of 1 mm and 2 mm. The model agrees reasonably well with limited experimental data for equilibrium scour shape and size with fully submerged cases compared to the cases where the bridge deck is partially submerged. This developed three-dimensional CFD scour computation procedure provides a basis for testing of additional scour related physical models while also providing an evaluation tool that can be used immediately by engineers engaged in scour risk analysis and assessment.

NOMENCLATURE

Symbol	Unit	Description
b	m	Cross flume bridge deck length
d_{50}	mm	Median channel-bed material diameter
d_*	-	Dimensionless diameter
Fr	-	Froude number
g	m/s^2	Acceleration due to gravity
h_b	m	Height of deck bottom above upstream bed
h_u	m	Height of the water column from flat channel bottom
h^*	-	Inundation ratio

k	m^2/s^2	Turbulent kinetic energy
L	m	Length of the bridge
Re	-	Reynolds number
s	m	Bridge height
W	m	Width of the bridge
y	cm	Scour depth at a specific x location

Greek Symbols

Symbol	Unit	Description
ε	m^2/s^3	Turbulent dissipation rate
ν	m^2/s	Kinematic viscosity of water
ρ	kg/m^3	Density of water
ρ_s	kg/m^3	Density of sediment
σ	m/Pa	Constant in bed displacement function
τ	N/m^2	Bed shear stress
τ_c	N/m^2	Critical shear stress
τ_{max}	N/m^2	Maximum bed shear stress

INTRODUCTION

Design and construction of new bridges and evaluation of the safety of existing bridges is increasing in importance as the U.S. transportation infrastructure ages. One of the major reasons for the failure of bridges is scour-hole formation around bridge support structures during floods. Under flooding conditions, bridges may be subjected to a greatly increased failure risk due to the accelerated scouring of the river bed at the bridge support structures. In the past, physical modeling was used as the primary approach to understand and evaluate the scouring process. Experiments were generally conducted by reducing the geometry scale by between 1 and 2 orders of magnitude. However, experimental investigations take a long time, are very costly, and cannot simultaneously match all of the relevant non-dimensional parameters at the laboratory scale. An alternate way to evaluate and better understand the process of scouring is to perform modeling and simulations at full scale. A CFD based simulation model can be used as an analysis tool on modern high performance computers for evaluating scour and failure risk of existing bridges and in the design of new bridges [1]. Argonne National Laboratory, in collaboration with Northern Illinois University (NIU) [2,3] and the Federal Highway Administration Turner-Fairbank Highway Research Center (TFHRC), has been developing and testing methods to use well-benchmarked commercial CFD software to model scour processes at bridges under flooding conditions. TFHRC has been conducting experiments to provide data for validation of computational and physics based empirical models [4]. The FLOW-3D software has a simple transient scour model evaluated by Smith [5] and [6] for modeling scour around cylinders on a seabed under high current flow conditions. Argonne evaluated this model applied to computing scour under flooded bridge decks over a one to two day period and found that the transient computation would take many months of computer wall clock time due to the several orders of magnitude difference in the flow (milliseconds) and erosion (minutes to hours) time scales. To bring the scour computation time down to a more feasible one to two days, Argonne and NIU developed a modeling approach that uses a series of fully automated quasi-steady CFD runs each followed by a bed displacement and domain remeshing step to compute the final equilibrium scoured bed bathymetry.

DESCRIPTION OF THE PHYSICAL PROBLEM AND COMPUTATIONAL MODEL

The CFD computational model's domain representing the experimental scour flume at (TFHRC) is shown in Figure 1. The entry section of the scour flume contains a honeycomb to straighten the flow and strip off the boundary layer. The position of the honeycomb is the inlet to the computational domain, and a uniform velocity inlet boundary condition is applied there equal to the mean cross section flow velocity. Just after the honeycomb a short length region of high shear stress exists where the boundary layer reforms. The high shear may exceed the critical shear stress, but there is no loose sand on the bed in this region and therefore no bed erosion.

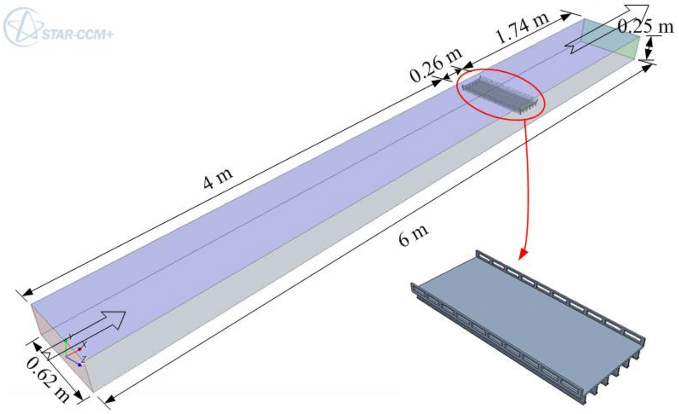

Figure 1: Diagram of the modeled 3D domain

Figure 2 is a diagram of the test bridge deck submerged in the flume test section with the reference parameters. The height of the bottom of the submerged deck above the initial flat bed of the sand in the flume is h_b, h_u is the upstream height of the water surface, W is the width of the deck in the flow direction, s is the height of the deck, and h^* is inundation ratio, a non-dimensional parameter indicating how deeply the deck is submerged given by:

$$h^* = \frac{h_u - h_b}{s} \tag{1}$$

When $h^* > 1$, the deck is fully submerged and partially submerged otherwise. For all test cases, $s = 5.8$ cm and $h_u = 25$ cm. Therefore the condition for a fully submerged bridge deck is $h_b < 19.2$ cm. To conserve computational resources and obtain results within one to two days for a case, the analysis was done as a single phase flow with the water surface assumed to be flat using a symmetric boundary condition that allows for a slip velocity. This assumption is good when the deck is submerged well below the surface but is not as good for h^* near one or less than one (partially submerged).

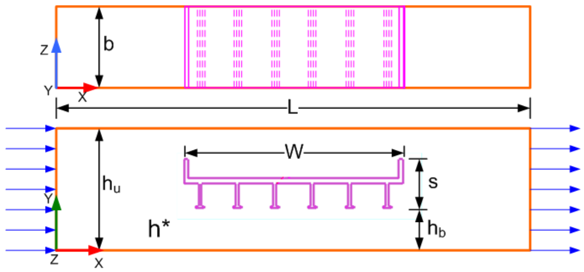

Figure 2: Diagram of flooded deck in flume

The scour process that changes the location of the sediment bed through entrainment of stationary particles at the bed interface with the moving stream takes place within the boundary layer at the bed. Maintaining an accurate position of the bed interface with a well resolved bed contour and high quality computational mesh in the vicinity of the bed is an approach to scour modeling that appears to provide the basis for a robust model. In this approach, processes that are occurring within the porous media stationary sediment bed would be modeled as a separate subdomain of the problem. Because the primary goal in the current study was to establish and evaluate procedures for displacing the sediment bed under conditions where erosion will occur, the process is taken to be a series of steps in which the bed is displaced in proportion to the shear stress above the critical shear stress for the onset of particle entrainment from the bed. The entrainment rate, sediment transport, and settling are not included in this initial modeling effort. The model was kept as simple as possible to concentrate on testing iterative displacement of the bed surface in a three-dimensional domain to obtain the asymptotic scour hole contour after a long period of time.

As shown in Figure 3, the process of displacing the bed includes many steps using several commercial and open source software components to maintain a high quality mesh during iterative bed displacement driven by the computed shear stress distribution. Because the flow time scale is on the order of milliseconds while the erosion time scale is initially on the order of minutes and extends to hours as the asymptotic bed contour is approached, the RANS turbulence model flow computation step can be treated as quasi-steady and computed as a steady state. The modeling process is an iterative procedure in which 1) the flow field is solved using the current bed contour to obtain the bed shear stress distribution, 2) the bed shear stress distribution is used to displace the bed, 3) the flow domain is remeshed to maintain a high mesh quality as the bed is displaced, and 4) the process is repeated until the bed shear stress is everywhere at or below the critical shear stress within a small tolerance value.

Each solve of the flow field is done as a batch job submitted to a high performance computer cluster. The mesh contains about 1.3 million cells and the flow solve step completes in about 30 minutes when solved on 32 processors. Starting from flat bed conditions as in the experiments, reaching the final scour hole contour required between 100 and 200 iterations of the procedure depending on the case. When a human carried out the steps to run the various software needed to complete one iteration, only a few iterations could be completed per day, which would have required at best one to several months to complete a single case analysis. Therefore the entire process was automated with various programming scripts as shown in Figure 3 to bring the iterative case analysis time down to 1 to 2 days with minimal human intervention.

A Bash script acts as the master process to carry out the sequence of steps in each iteration. Because Python is proficient in parsing and manipulating text, a Python program was developed to process a file containing the bed shear stress distribution and generate a file of thousands of commands to displace the bed at vertices in a CFD preprocessor software component.

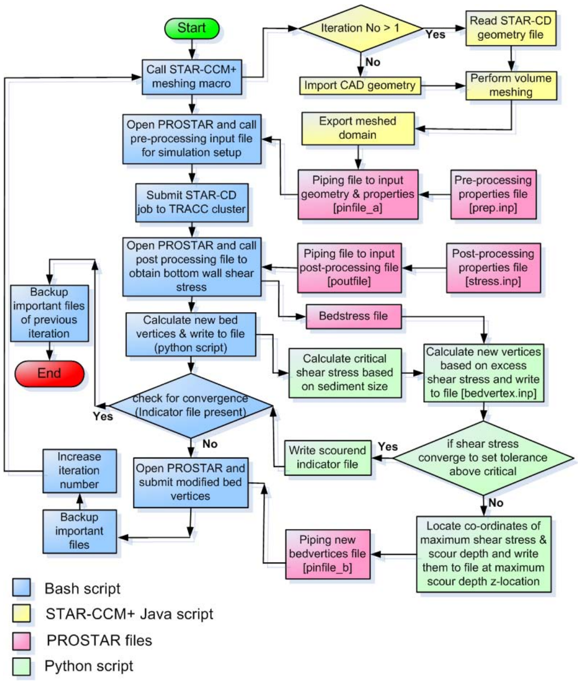

Figure 3: Major components of the computational model showing the CFD solver step as only one important task among many to automate the iterative erosion modeling procedure

Guo's relation [7] for critical shear stress, τ_c, for the onset of particle motion for a flat bed was used in the model:

$$\frac{\tau_c}{(\rho_s - \rho) g d_{50}} = \frac{0.23}{d_*} + 0.054\left[1 - \exp\left(-\frac{d_*^{0.85}}{23}\right)\right] \tag{2}$$

where d_{50} = mean diameter of sediment, ρ_s = density of sediment, ρ = density of water, g = acceleration due to gravity, τ_c = critical shear stress, and the dimensionless diameter d_* is given by:

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

where ν = kinematic viscosity of water.

The critical shear stress for 2 mm sand is 1.41 Pa and for 1 mm sand it is 0.59 Pa. A correction for sloped beds is often applied based on the bed slope with respect to the flow velocity and gravity vectors. In the cases included in this study, the slope of the bed in the scour hole that forms under the deck is very small and neglected in this initial work.

Sediment bed displacements are computed from the shear stress distribution using:

$$\Delta y_i = \max(0, \min(\sigma \Delta\tau_i, \Delta y_{\max})) \tag{4}$$

where $\Delta\tau_i = \tau_i - \tau_c$ is the excess shear stress above critical for i^{th} iteration, $\Delta y_{\max}$ is the maximum bed displacement per flow solver iteration chosen to maintain numerical stability, which was taken to be the sediment grain size of 1 or 2 mm, and σ is taken to be $\Delta y_{\max}/\Delta\tau_{\max}$ after the first several iterations. The value of σ ranged between 1 and 3 mm/Pa. The max function prevents displacing the bed when the shear stress is less than critical.

After each quasi-steady flow solve step i, the bed contour is updated according to:

$$y_{i+1} = y_i + \Delta y_i \tag{5}$$

In the k-epsilon turbulence model used in the flow computation, the sediment bed boundary is modeled as a rough wall. The bed shear is computed through the use of wall functions, and the computed shear is a function of the effective roughness of the wall. Camenen et.al. [8] reviewed a variety of formulae for determining the effective wall roughness as a function of bed particle size for mono-sized particles. Depending on particle diameter, the values from the different formulae may vary by a factor of more than 2. A common value is an effective roughness of twice the value of d_{50}, and that value was used for the 1 and 2 mm sand cases in this study.

RESULTS AND DISCUSSION

The primary goals of this project were to develop and test a basic methodology for modeling riverbed scour in a 3D domain that can be applied in the future to an entire bridge using well benchmarked and maintained commercial CFD software supplemented by well-maintained open source software as needed to automate the series of 100 to 200 quasi-steady CFD runs needed to obtain the final result. Data was obtained from a series of scour experiments conducted at TFHRC on bridge pressure flow scour under clear water conditions. The experiments were designed to refine a two dimensional analysis procedure, and while there were no piers in the experiments to introduce 3D effects, bed stress near the side walls of the flume is lower than in the center region under the flooded bridge decks and does produce a 3D effect. Data for extensive 3D bathymetry maps of the scoured test section was collected using a laser distance sensor [9]. This data compares well to the results of analysis with the CFD modeling procedure given the simplicity of the procedure and that it is intended to provide a basis for implementing and testing more extensive erosion physics models.

The flow domain with dimensions is shown in Figure 1. The experiments were designed to come close to the maximum clear water scour under the flooded bridge deck. Therefore the mean upstream flow velocity was set at 95% of the mean velocity needed to reach critical shear stress on the bed in an unobstructed fully developed flow. For 1 mm sand the mean velocity is 0.41 m/s, corresponding to a discharge of 0.065 m^3/s. The Reynolds number based on water depth is Re = 1.03 x 10^5, and the Froude number is Fr = 0.26. For the 2 mm sand cases, the unobstructed channel mean flow velocity was 0.53 m/s, giving a discharge of 0.084 m^3/s, a Reynolds number based on water depth of 1.33 x 10^5, and a Froude number of 0.34.

Results for a case with a fully submerged bridge deck 16 cm (h_b) above the initial and upstream bed using 2 mm sand are presented for illustration and discussion. Detailed results for the range of cases included in the study can be found in Tulimilli's master's thesis [10].

The relaxation of accelerated flow under a flooded bridge deck blockage when the scour hole forms as a consequence of the elevated shear stress generated on the bed is shown in Figure 4. The upper part of the figure shows a high velocity zone in red to orange (0.9 to 0.7 m/s) that extends down very close to the bed. These higher velocities above the bed yield a shear stress that is more than two times the critical shear stress on the initial flat bed. The lower plot in the figure shows the equilibrium scour hole profile down the flume centerline with the lower velocity distribution that is a consequence of increased cross section flow area. The shear stress on the bed in the lower figure is at or below critical in the vicinity of the deck.

A comparison of the bathymetry of the scour hole and up and downstream bed is shown in Figure 5. The scoured bed

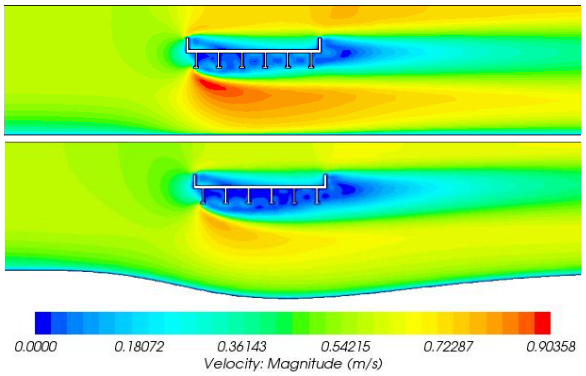

Figure 4: Velocity magnitude for initial flat bed and final scoured bed showing reduction of high velocity under the bridge deck

surface shows the 3D effects of the presence of the flume side walls. Lower shear stress at the bed near the corners with the side walls yield less scour and create a scour hole that is slightly bowl shaped. The scales in the three coordinate directions in the figure are not the same, and therefore the depth is not to scale to provide a better visualization of the scour hole.

The shape and depth of the experimental and computed scour holes are close, with several differences worth noting. The model does not currently include sediment transport, and therefore there is no settling of sediment back to the bed in the downstream where turbulence intensity is not sufficient to keep the sediment suspended. Consequently in the experimental results, there is a pile of sediment that has settled back to the bed downstream of the deck blockage that does not appear in the computed results.

The computational model moves bed material as a function of the excess mean bed shear computed from the RANS turbulence model above the critical shear stress. Upstream conditions in the experiment were designed to have a flow that produced a mean bed shear about 5% below critical shear stress. Consequently, the model does not yield any movement of bed material in the upstream region. In reality turbulent fluctuations will produce bed forces that are periodically high enough to move particles, and consequently roughness in the bed of the experimental result that is in a range of 10 mm above and below the initial bed height can be seen in Figure 5a. Large eddy simulation (LES) or a similar computationally intensive eddy resolving technique would be needed to model the flow physics required to compute the small scale particle movement in the upstream. Such computations are currently not feasible on commodity high performance computer clusters for use in routine analysis. This degree of fine detail may not be needed, if the intent is to improve the scour analysis over current methods in order to obtain better safety evaluation of scour at bridges.

A significant amount of effort in preparing the experimental flume went into obtaining a velocity profile in the upstream of the test section that was as uniform as possible across the width. The asymmetric scour hole in Figure 5a indicates that uniform inlet flow conditions were not achieved in the experiment. In the iterated CFD computation the inlet velocity profile can be maintained to be highly uniform to machine precision. Consequently, the computed scour in Figure 5b is highly symmetric about the flume centerline.

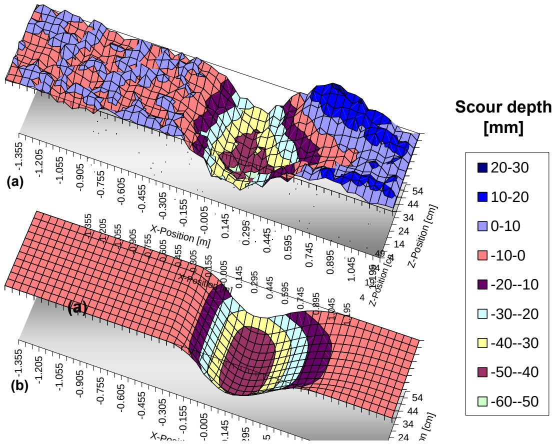

Figure 5: Comparison of experiment and computed scour hole for h_b = 16 cm case with 2 mm sand

The cross section profile of the scour hole at its deepest point is shown in Figure 6. The computed scour hole profile is smooth and symmetric, while the experimental profile is not for reasons already discussed. While the vertical and cross stream scales are not the same, a significantly steeper slope in the vicinity of the flume side walls is apparent. The current physics model does not account for the variation in critical shear stress with the slope of the bed. In addition, it does not have a model for sand slides when the slope exceeds the angle of repose of the sand. While the additional models can be added in the future to improve accuracy, the model with limited simple physics captures 3D effects and does not under predict the scour hole depth, which is important in performing engineering safety evaluations of risk to bridges due to scour.

Figure 7 shows the longitudinal shear stress profile under the submerged bridge deck for the initial flat bed condition and its reduction at intervals through the CFD computation iterations with bed displacement until the shear stress is at or below critical. For this 2 mm sand case the critical shear stress is 1.41 Pa, and the peak shear stress under the submerged deck at the initial flat bed condition is 3.7 Pa.

The corresponding displaced bed profiles are shown in Figure 8. The current model is not transient and does not use an entrainment rate function based on experimental data to compute bed displacement as a function of bed shear. The bed displacements are simply proportional to the excess shear stress above critical, and therefore the intermediate bed profiles may not be physically realistic. The measured experimental points

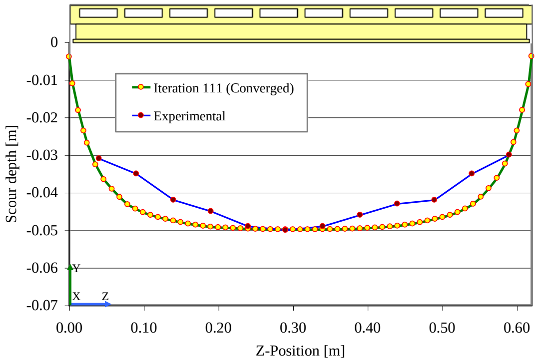

Figure 6: Comparison of experiment and computed cross section scour hole profile at deepest point for h_b = 16 cm, 2 mm sand case

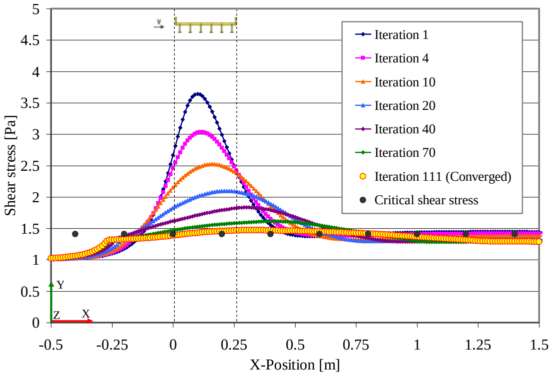

Figure 7: Relaxation of shear stress profile under the center of the deck with iteration toward final scour hole profile

of final longitudinal bed profile at flume center are plotted for comparison with the final computed profile. The upstream slope matches well even though the reduction of critical shear stress on the downslope is not accounted for in the model. Note that the vertical and horizontal axis scales are different in the figure to separate the curves, and consequently the slope is much less than it appears. On the downstream side, the computed scour hole profile is deeper than the experimental result because sediment transport and settling of particles onto the bed are not included in the model. This simplification is conservative for engineering safety analysis, and sediment transport and settling models exist in the major CFD software and will be tested in the future.

A comparison of the computed maximum scour hole depths and the measured depths for scale bridge decks immersed at heights above the sand flume bed ranging from 11.5 to 22 cm for 2 mm sand is shown in Figure 9 and for 1 mm sand in Figure 10. The experimental points were computed by averaging laser depth measurements across the middle third of the cross section away from the flume walls where the computed profile as shown in Figure 6 is nearly flat. The range bars around the experimental points show the minimum and maximum of the measured values that were averaged. For 2 mm sand fully submerged cases, the agreement is excellent. Note, however, that the scour hole depth computed from the CFD procedure depends on the effective roughness and critical shear stress parameters. Published formulas for these parameters as a function of sediment properties have values that vary significantly. Therefore, caution and checking against data are needed when applying this methodology to other geometries and sediments. The computed 1 mm sand cases shown in Figure 10 were close to experiment, but not as close as those for 2 mm sand cases. As previously discussed, the top of the bridge deck is at the upstream water surface for h_b = 19.2 cm, and the flat surface assumption with slip symmetric boundary breaks down in this range of deck positions. The effect on the computation is that more water is forced under the deck increasing the scour hole depth compared to experiment, which at least errs on the side of increasing the safety margin in a risk assessment. The difference between the experimental and computed scour hole depth is near 50% for the worst case. Experimental uncertainty for the partially submerged cases with shallow scour holes is also quite high and difficult to quantify, and shallow scour holes are of less engineering concern.

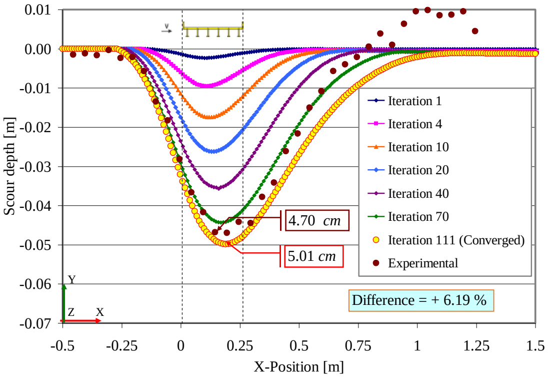

Figure 8: Iterative displacement of bed as a function of excess shear with final experimental and computed profile at the flume centerline

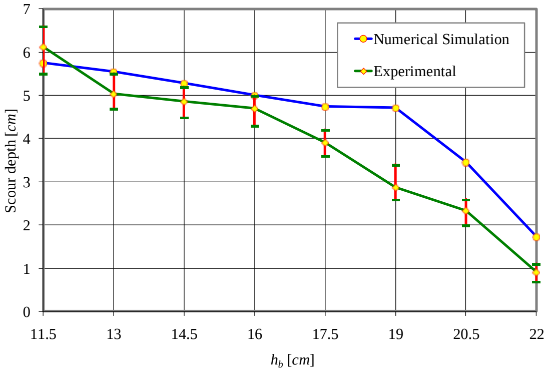

Figure 9: Comparison of computed and experimental scour hole depth variation with deck submergence level for 2 mm sand

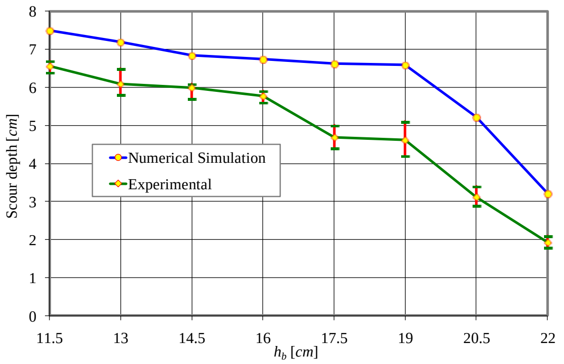

Figure 10: Comparison of computed and experimental scour hole depth variation with deck submergence level for 1 mm sand

CONCLUSIONS

A 3D stream bed scour modeling methodology was developed using well-benchmarked commercial CFD software to compute the bed shear stress distribution used to calculate bed displacements and to re-mesh the computational domain as the bed is displaced. The methodology uses a fully automated series of CFD runs, bed displacements, and re-meshing to compute the equilibrium scoured bed contour with shear stress distribution at or below the critical shear stress in a 3D domain. The method was applied to model a series of pressure flow scour experiments and was shown to capture the 3D effects of the presence of the flume walls in the overall shape and depth of the computed scour holes. The procedure provides a good foundation for testing more complex physics models of stream bed erosion and sediment transport to improve scour risk assessment at bridges.

ACKNOWLEDGMENTS

The authors would like to acknowledge support by Dr. Promod Vohra, College of Engineering and Engineering Technology of Northern Illinois University (NIU), Dr. David P. Weber of Argonne National Laboratory, use of the high performance computer at Argonne's Transportation Research and Analysis Computing Center (TRACC) and financial support from the U.S. Department of Transportation provided through TRACC.

REFERENCES

[1] Singh, V. (2005). Two Dimensional Sediment Transport Model Using Parallel Computers. M.S. Thesis, Banaras Hindu University.

[2] Adhikary, B. D. (2008). Flow And Pressure Scour Analysis Of An Open Channel Flow Having An Inundated Bridge Deck Under Various Flooding Conditions, M.S. Thesis, Northern Illinois University.

[3] Biswas, D. (2009). Development of an Iterative Scouring Procedure for Implementation in CFD Code for Open Channel Flow Having an Inundated Bridge Deck Under Various Flooding Conditions, M.S. Thesis, Northern Illinois University.

[4] Guo, J., Kerenyi, K., Pagan-Ortiz, J. E., and Flora, K (2009). Bridge Pressure Flow Scour at Clear Water. Threshold Condition, Tianjin University and Springer-Verlag, DOI 10.1007/s12209-009-0016-3

[5] Smith, H. D. and Foster, D. L. (2007). Three-Dimensional Flow around A Bottom-Mounted Short Cylinder. Journal of Hydraulic Engineering, Vol. 133, No. 5, pages 534-544.

[6] Smith, D..(2004) Modeling the Flow and Scour Around an Immovable Cylinder, M.S. Thesis, Ohio State University

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

[8] Camenen, B., Bayram, A. and Larson, M (2006). Equivalent Roughness Height For Plane Bed Under Steady Flow, Journal of Hydraulic Engineering, Vol. 132, No. 11, pages 1146-1158.

[9] Guo, J., Kerenyi, K. and Pagan-Ortiz, J. E. (Oct. 2009). Technical Report No. FHWA-HRT-09-041, Bridge Pressure Flow Scour for Clear Water. Conditions

[10] Tulimilli, B.R.(2010), Development of Generalized 3D scouring Methodology for Implementation in a CFD Code, M.S. Thesis, Northern Illinois University

[11] Melville, B.W.(2000), Bridge Scour, Water Resources Publications, LLC