Development of CFD Simulation for 3-D Flooding Flow and Scouring Around a Bridge Structure

B.R.TULIMILLI, P.MAJUMDAR, and M.KOSTIC
Department of Mechanical Engineering
Northern Illinois University
DeKalb, IL 60115
U.S.A.

S.A.LOTTES
Argonne National Laboratory
West Chicago, IL
U.S.A.

Abstract: - Analysis of drag and lift forces and evaluation of scour around bridge structures are necessary to assess risk of bridge failure during storms and floods. Simulation of scour-hole formation under the bridge deck and around the bridge piers, due to sediment entrainment and transport caused by flooding flow conditions, is of significant interest to computational fluid dynamics (CFD) and hydraulics researchers. This study is focused on simulation of 3-D open channel turbulent flow over an inundated bridge deck to obtain the final shape and size of the scour-hole. This 3-D study extends the previous 2-D simulation scouring methodology. Solutions for flow field and turbulence, using an effective bed roughness value, are based on the Reynolds Averaged Navier-Stokes (RANS) equations, and a k–ε turbulence closure model using a commercial CFD code. An iterative computational methodology is developed to predict the equilibrium 3-D scour-hole using a single phase model moving boundary formulation, based on an empirical correlation for critical shear stress to determine the condition for sediment removal. The computational model has demonstrated capability to reach a converged solution for the equilibrium scour-hole shape and size that agree reasonably well with experiments. The method provides a basis for implementing enhanced particle entrainment physics and other scour model improvements in well benchmarked commercial CFD software.

Key-Words: - Bridge structures, CFD simulation, Computational fluid dynamics, Flood flows, Scour, Turbulence closure model

1 Introduction

One of the major reasons for the failure of bridges is the formation of scour-holes that undermine bridge pier and abutment foundations during major flood events. This causes weakening of bridge structure and hence increases the risk of bridge failure. Analysis of the formation of scour-holes under bridge decks and around bridge piers due to entrainment and transport of sediment caused by flooding and flow conditions are important for application to the design, construction, and maintenance of bridges.

Argonne National Laboratory’s Transportation Research and Analysis Computing Center (TRACC) [10] and Northern Illinois University are collaborative partners with the Federal Highway Administration’s Turner Fairbank Highway Research Center (TFHRC) [7] Hydraulics Laboratory and the University of Nebraska in a bridge hydraulics research program to investigate, develop, and validate CFD methods to evaluate scour at bridges during major flood events using commercial CFD software. Two Northern Illinois University students (B. Adhikary and D. Biswas) have completed Master’s Degrees in this effort. Adhikary [6] analyzed an open channel turbulent flow and resulting stresses over inundated bridge decks and developed a 2-D iterative computational methodology for predicting equilibrium scour profiles using a single-phase moving boundary formulation. The methodology relies on an empirical correlation for critical bed shear stress that is used to determine the condition for onset of sediment motion and an effective bed roughness that is a function of sediment particle size. Biswas [5] extended this study by further refining the methodology taking into account the slope dependent variable critical shear stress. The method was automated and focused on the prediction of pressure flow scour for partially submerged decks.

To save computational resources during development of the method and allow the problem to be reduced to two dimensions (streamwise and vertical) the 3D bridge deck geometry was replaced by a block of the same aspect ratio and the simulation results were compared with available experimental data.

This study extends the 2-D iterative scouring methodology with an objective to develop an automated iterative computational methodology for simulating a 3-D domain with an inundated bridge deck to predict the final shape and size of the scour-hole. The 3-D scour methodology has been developed using the STAR-CCM+ and STAR-CD commercial CFD software. Equilibrium scour is computed using an iterative procedure that moves the sediment bed boundary proportional to the excess shear stress over the critical shear stress. The method is being validated by comparing results for the equilibrium scour shape and size with the experimental data obtained from TFHRC [7].

2 Description of the Physical Problem

A series of flume experiments were conducted by TFHRC [7] to measure pressure flow scour under an inundated bridge deck. The setup of the flume with sand bed and bridge deck is shown in Fig. 1.

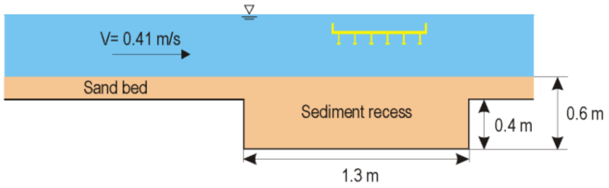

Fig. 1: Experimental flume at test section (front view)

A honeycomb flow straightener was used at the inlet to create a uniform flow inlet condition. Experiments were conducted until the equilibrium scour shape was achieved. The scour-hole size was measured using a laser distance sensor after equilibrium scour conditions were reached. A series of experiments were conducted at TFHRC for different bridge deck heights above the sand bed and for a mean sand diameter of 1 *mm* and 2 *mm*.

A schematic of the open channel flow configuration for flow over a six girder bridge deck is shown in Fig. 2.

The characteristic dimensions shown in Fig. 2 are as follows: h_u = upstream water height from free surface to the sand bed, h_b = water depth from the bridge bottom to channel bottom, s = bridge height, b = width of the channel, L = length of the channel,

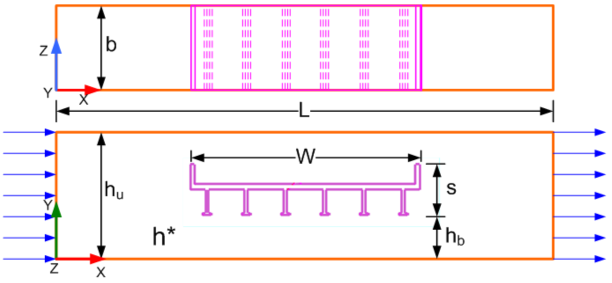

Fig. 2: Front and top view of computational domain with characteristic dimensions (not to scale)

W = width of bridge deck and h^* is the inundation ratio. Inundation ratio h^* is a dimensionless number which indicates bridge flooding conditions and is given as

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

A value of $h^* > 1$ denotes a fully submerged bridge deck, $h^* < 1$ denotes a partially submerged bridge deck and $h^* = 1$ when the deck is just submerged.

3 Computational Model for Scour

In this section, all the computational approaches and other parameters required for setting up the simulation are discussed along with the available modeling options.

3.1 Modeling Approach

A number of modeling options were possible for computing scour under bridge decks using CFD and were investigated by different researchers. The multiphase VOF model can be used to simulate a free surface flow with scouring of the bed, but the VOF model requires large computational resources to run a transient simulation till convergence for equilibrium scour depth is achieved. Eulerian two phase modeling includes more of the fundamental physics, including sediment transport, however, good models for the entrainment of sediment from a stationary particle bed are not currently available in commercial CFD software. FLOW-3D does have a primitive particle entrainment and transport model, but does not have a means to bridge the flow and erosion time scales making computation of a 40 hour erosion infeasible because it would require many months of computation time even on a large parallel computer.

In the present study, a quasi-steady-state single-phase model was used. It was assumed that the scouring is due to the supercritical shear stress, which is defined as the excess shear stress over a critical shear stress. A computational methodology is designed to move the bed after each iteration to a quasi-steady-state in which bed erosion time is much longer than the fluid cell residence time. Between iterations that compute shear stress for flow over the current bed contour, the bed location is moved down in small increments at locations where shear stress is higher than the critical shear stress.

3.2 Computational Domain

The computational domain was designed based on the experimental flume dimensions. The STAR-DESIGN commercial CFD software was used for generating the CAD model and is shown in Fig. 3.

An inlet length of 4 *m* was provided for the flow to develop before it reaches the bridge deck, and 1.74 *m* was provided after the bridge to place the outlet sufficiently far downstream of recirculation zones that occur in the wake of the bridge deck. Also the width of the computational domain and the dimensions of the bridge deck are the same as that of the experiments. STAR-CCM+ was used to re-mesh the domain between iterations adjusting the bed contour because it generates high quality meshes. A uniform velocity boundary condition was specified at the inlet (0.53 *m/s*), which is about 95% the mean velocity needed to produce critical shear stress in a fully developed flow. A bed surface roughness equivalent to 2 *mm* sand was specified at the bed surface. A standard zero gradient stream wise outflow condition was assigned at the outlet, and the water surface was assigned a symmetry boundary condition. The side walls and bridge deck were set to be hydraulically smooth walls to match the experimental conditions. Computational cell layers were created near the bottom and side walls to be within the correct range for application of wall functions to determine bed and wall shear stress, and the grid around the bridge deck was refined to more accurately capture the flow field around it.

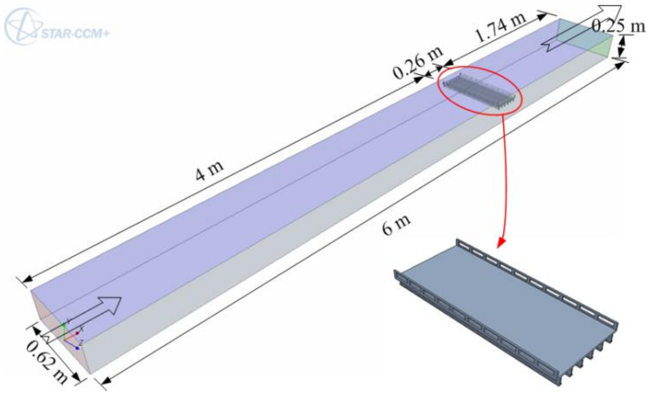

Fig. 3: Computational domain with zoomed view of bridge deck

At the start of each iteration, the re-meshed domain was imported into STAR-CD software and all properties, turbulence closure model, initial conditions, boundary conditions, and solver parameters were assigned. The STAR solver was then used for running the simulation to determine bed shear stress for the new bed contour.

3.3 Turbulence Model

There are many turbulence modeling options available. Use of Large Eddy Simulation (LES) may provide the most accurate flow field for scour computation, but its use for practical scour computations that can be completed within a day or two on a high performance computer cluster is not currently feasible. Therefore, to obtain reasonable computation times, a high Reynolds number k–ε turbulence model was used in this study.

3.4 Effective Bed Roughness

The sediment bed is modeled as a rough wall. A review of correlations of effective roughness as a function of sediment size is given by Camenen *et al* [4]. While the range of these correlations varies by more than 100%, an effective roughness equal to twice the diameter of the 2 *mm* sand particles modeled, or 4 *mm*, is within the range of many correlations and was used here. To further improve the accuracy of this modeling approach, it may be necessary to re-adjust the equivalent roughness by comparing the equilibrium scour-hole depth with that of the experimental.

4 3-D Scouring Methodology

In this section the methodology used for scouring the bed and its implementation in commercial CFD software are discussed.

4.1 Critical Shear Stress

In general, the critical shear stress is not a constant, but a function of flow and other properties like slope of bed, porosity, and type of sediment, etc. In this study a constant value of critical shear stress was used and is reasonable for this initial effort to model the lab scale pressure flow experiments with a commercial CFD code because the slope of the scour holes was very small. A review of various approaches to determine the critical shear stress at the bottom wall for an open channel flow is given by Singh [1].

The critical shear stress correlation used in this study was taken from Guo [2]

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

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:

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

Based on the above correlation a constant value of critical shear stress for 2 *mm* sand was calculated and used in this study.

4.2 Methodology for Computing the Equilibrium Scour-Hole Contour

This methodology involves moving an initial flat bed downward in increments proportional to the excess shear stress above critical at a point on the bed. The iterative scour rate used could be any value as long as the bed contour evolution is numerically stable and converges to an equilibrium scour shape. The local scour (increment in gravity direction) is given as:

$$\Delta y_i = \max(0, \min(\sigma\Delta\tau_i, \Delta y_{\max})) \qquad (5)$$

Where, $\Delta\tau_i = \tau_i - \tau_c$ is the supercritical shear stress for i^{th} iteration, σ is the proportionally constant used to determine a bed displacement from excess shear above critical, and Δy_{max} is the maximum allowable scour per iteration. To maintain numerical stability, Δy_{max} is taken to be equal to the sediment particle diameter. The local scour for each iteration is added to the previous scoured bottom coordinate. Therefore, a new scour bottom coordinate generated after each iteration will be:

$$y_{i+1} = y_i + \Delta y \qquad (6)$$

4.3 Implementation of Methodology

The initial step is to generate the CAD geometry in STAR-DESIGN. The flow chart in Fig. 4 gives the overview of the scour computation procedure.

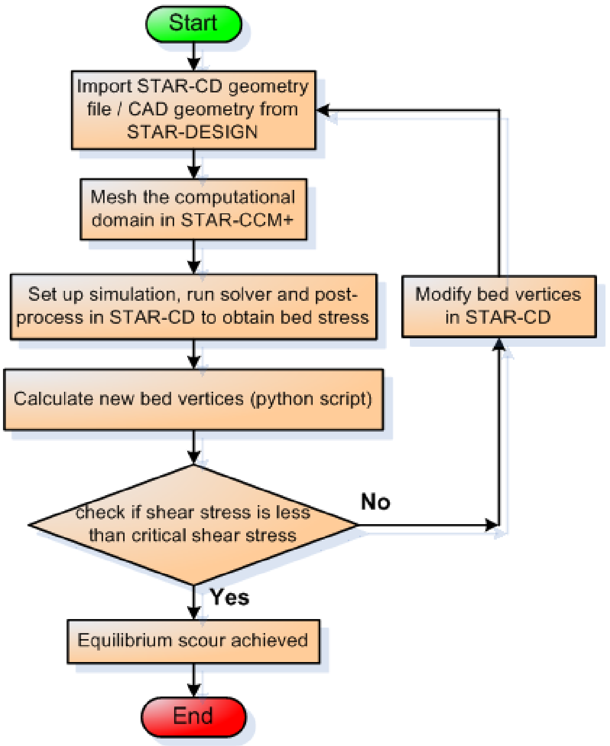

Fig. 4: Scouring methodology algorithm

This methodology has been implemented to iteratively compute the equilibrium bed profile under pressure flow scour conditions starting from a flat bed. The flow field is solved by STAR-CD yielding the mean bed shear stress distribution. Bed displacements and new bed vertices are computed in a separate *Python* program as a function of bed shear. Pro-STAR, the STAR-CD pre-processor is used to modify the bed vertex positions, and the auto-meshing capabilities of STAR-CCM+ are used to re-mesh the domain with hexahedral cells to maintain mesh quality as the bed is deformed. To run these programs iteratively without human intervention, a *BASH* script was written. The *BASH* script runs the programs sequentially in a loop until convergence to an equilibrium scour bed profile has been reached.

The *Python* program reads the bed stresses and checks for convergence. Convergence is considered to be achieved if the bed shear stress is everywhere no more than 2% above the critical value. If convergence is not achieved, bed displacements are computed using Eqn. 5, and new bed vertices are computed from Eqn. 6. The Python program also writes the maximum shear stress and its location, maximum scour depth and its location, shear stress data at the *z*-location where shear stress is maximum and scour depth data at the *z*-location where scour depth is maximum. In addition, it also prepares all other files for the next iteration.

5 Results and Discussions

The computational domain for a typical simulation case is shown in Fig. 3. The channel height (h_u) was 0.25 *m*, and the water depth from bottom of bridge to bed (h_b) was 13 *cm*. The inundation ratio was 2.07, which is a completely submerged case. The operating upstream velocity at the inlet is specified as 0.53 *m/s* and the turbulence parameters specified are the turbulence kinetic energy and dissipation rate.

The proportionality constant, σ, in Eqn. 5 to compute bed displacements is taken to be 0.001 *m/Pa*. This value is carefully chosen to avoid instability in the evolution of scour contour on the bed. For this value the maximum displacement of the flat bed is 2.5 *mm* in the first iteration and the maximum displacement of the bed contour gradually decreases as the scour hole approaches the equilibrium and the maximum shear decreases. The maximum allowable scour depth per iteration (Δy_{max}) is chosen as 3 *mm*. This parameter helps in cutting down the peaks which arise due to instabilities and thus prevents instabilities in the iterative procedure.

Fig. 5 shows velocity vectors for flat and scoured computational domains. Fig. 5(a) shows a velocity vector plot for the flat bottom intersected by plane at the midsection [$z = b/2$]. Maximum velocity occurs near and below the upstream end of bridge deck. Fig. 5(b) shows the velocity vector plot for the final scoured bed on a plane at the midsection [$z = b/2$]. Scouring increases the flow area under the deck and results in a lower velocity and shear stress on the bed.

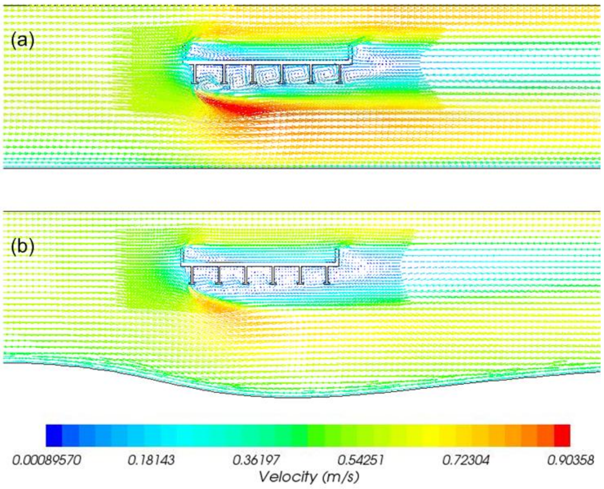

Fig. 5: Velocity vectors at midsection [$z = b/2$] (a) Flat bottom, (b) Final scoured bottom

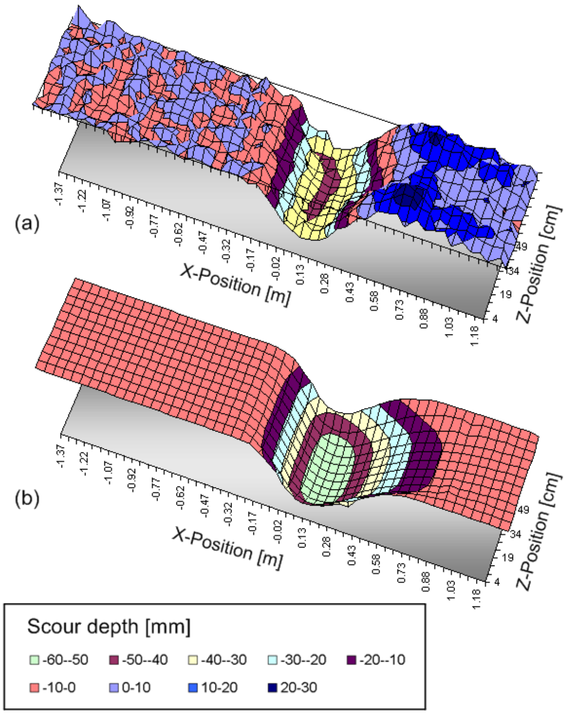

Fig. 6: 3-D scour map (a) Experimental (b) Numerical simulation

Experiments were conducted by TFHRC [7] for a range of bridge deck depths. For the h_b = 13 *cm* case presented here, the equilibrium scour hole had a maximum depth of 5.11 cm in the experiment. Fig. 6 shows the experimental and computational scour bed profiles. In the model mean bed shear from the turbulence model is used to compute scour displacements of the bed, and it is everywhere below critical in the upstream. Therefore, bed elevation remains flat at its initial value in the upstream. In reality in the experiment, there are turbulent fluctuations that will generate super-critical shears periodically at random locations in the upstream and result in the creation of non-flat bed forms seen in Fig. 6a. The model does not yet include sediment transport and settling, and therefore there is no deposition of sediment downstream of the deck. This result is considered satisfactory in the current work because it does not underestimate the scour depth or lead to the consequent underestimation of the risk of bridge failure due to scour.

The over-all shapes of experimental and simulated scour-hole are similar, with differences that are expected with the use of a RANS turbulence model and not including sediment transport and settling. Noteworthy in the model prediction is the cup shape of the scour hole due to the presence of the side walls of the flume. This is a 3D effect that is not captured in 2D models.

Fig. 7 shows the shear stress profiles for different iterations. Shear stresses were plotted at the cross stream *z*-location where the shear stress is maximum and these locations are very close to *z* = *b*/2. The simulation converged after 135 iterations when the maximum shear stress was reduced to be within a tolerance of 2% of the critical shear stress. At this stage changes in bed elevation of a fraction of a millimeter are sufficient to reduce bed shear several percent below critical, and therefore using more computation time to further refine the scour contour is not needed given other sources of error in the model.

Fig. 8 shows the scour profiles for different iterations. These scour bottom profiles were plotted at the cross stream *z*-location where the scour depth is maximum and they are at or very close to *z* = *b*/2. The model over predicts the scour depth by about 11%, which is considered acceptable because it errs on the side of safety. The maximum experimental scour depth was 5.11 *cm* while the simulation maximum scour depth was 5.72 *cm*.

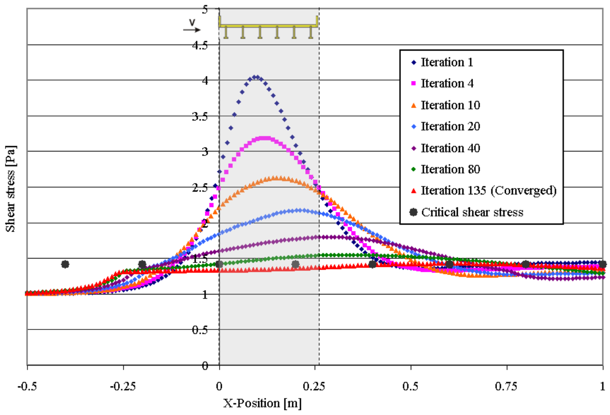

Fig. 7: Evolution of shear stress along with target constant critical shear stress

Fig. 9 shows the experimental and simulated cross sectional scour profiles at maximum scour *x*-location. Again, these profiles are reasonably close and the model does not under predict the scour depth. The model contour is smooth because it relies on a RANS turbulence model and uses the consequent mean bed shear in the scour bed displacement computation.

In Fig. 10 the convergence of maximum shear stress towards the critical shear stress has been plotted. Results show that maximum shear stress values approach the critical shear stress asymptotically.

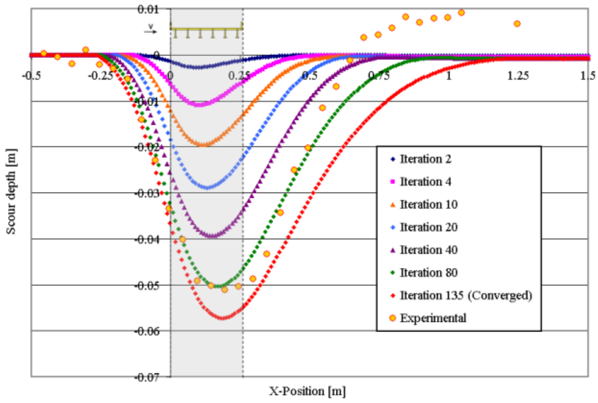

Fig. 8: Numerical evolution of scour and its comparison with the experimentally data

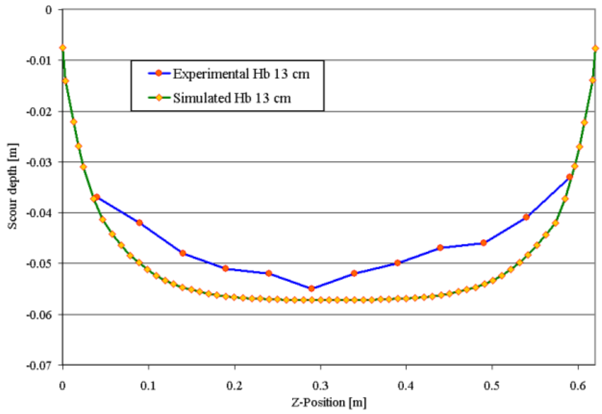

Fig. 9: Comparison of experimental and simulation cross sectional scour profiles at the maximum scour *x*-location

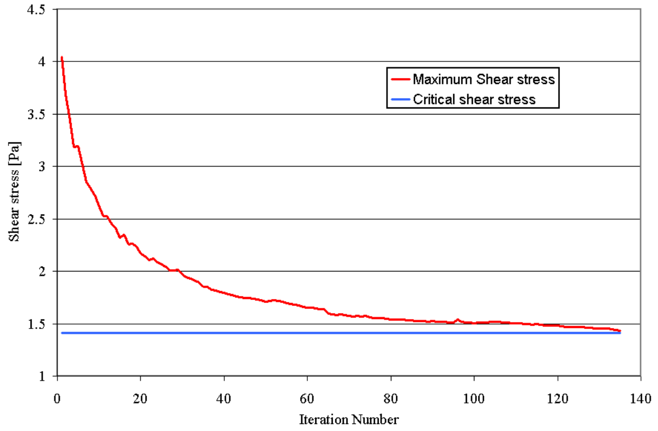

Fig. 10: Convergence of maximum shear stress with constant critical shear stress

6 Conclusions

An automated iterative computational methodology was developed for conservatively computing the equilibrium 3-D scour-hole depth and contour using pressure flow scour conditions as a first test of the method. The current approach uses single phase quasi-steady computations to obtain the bed shear and a moving boundary formulation based on an empirical correlation for critical shear stress to iteratively deform the bed toward equilibrium scour conditions The model uses Reynolds Averaged Navier-Stokes (RANS) equations, and high Reynolds number k–ε turbulence model in commercial CFD software to solve for the flow field. This single phase, quasi-steady-state model was chosen over multi-phase models for its simplicity and computational efficiency as the first step in developing methods to utilize more complex multiphase models. Even this simple approach, however, has been shown to be conservative in predicting pressure flow scour for the case of the inundated bridge deck presented in this paper and reasonably close to the experimental result, including most notably the three dimensional effects resulting from the presence of the flume sidewalls. Validation with comparison against experimental data over a range of bridge deck flood depths and 1 mm sand particle diameter is currently underway.

The 3D CFD scour computation procedure developed to use well benchmarked commercial CFD software provides a basis for adding additional physics models while also providing a tool that can be used immediately by engineers engaged in scour risk analysis and assessment.

Acknowledgements:
The authors like to acknowledge support by Dean Pramod Vohra, College of Engineering and Engineering Technology of Northern Illinois University (NIU), and Dr. David P. Weber, Director of TRACC as well as the financial support by U.S. Department of Transportation (USDOT).

References:

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

[2] Guo, J., Hunter Rouse and Shields Diagram, *Advances in Hydraulics and Water Engineering*, Vol. 2,2002,1068 – 1098

[3] 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.

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

[5] Biswas, D., *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, 2009

[6] Adhikary, B.D., *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, 2008

[7] Turner-Fairbanks Highway Research Center (TFHRC) http://www.fhwa.dot.gov/publications/research/infrastructure/structures/09041/index.cfm. Accessed in May 2010.

[8] Smith, Heather D., *Modeling the Flow and Scour Around an Immovable Cylinder*, M.S. Thesis, Ohio State University, 2004

[9] Melville, B.W., *Bridge Scour*, Water Resources Publications, LLC, 2000

[10] Transportation Research and Analysis Computing Center (TRACC). http://www.anl.gov/TRACC/index.html. Accessed in May 2010.