# Final Report for S&T Project X5772:

# Quantitative Modeling Tools of Scour and Morphological Impact due to Large Wood Debris and Other In-Stream Structures

## 1. Introduction

In the 2012 Reclamation S&T and USACE sponsored workshop on large wood debris (LWD) in fluvial environments, a number of key research needs relevant to Reclamation were identified. The highest priority group included focusing on the risk, safety, and the resultant scour and morphological impact to streams due to placement of LWD in rivers. Despite the existence of scour estimation methods such as bridge piers, methods or guidelines do not exist for predicting responses from construction of LWD. Even with traditional scour methods, they mostly rely on flume data that suffers from scale problems and field data that are scarce and expensive to get. In recent years, however, accurate scour and morphological predictions are becoming feasible with the availability of the state-of-the-art numerical modeling tools. These tools have not been evaluated and tested at Reclamation for LWD, but they were identified at the workshop to have potential to become the primary quantitative prediction tools for LWD related scour and morphological impact.

We envision the proposed effort will involve multiple regions and participation of both internal and external engineers and scientists. Therefore, we propose a scoping study in the fiscal year 2013 with the objective of determining the feasibility of using suitable modeling tools and seeking potential collaborators.

## 2. Study Questions

The proposed study intends to answer the following questions:

1. Do modeling tools exist to predict bed scour and morphological changes caused by LWD and in-stream structures?
2. Are these modeling tools adequate, if they exist, for assisting LWD design related to risks associated with scour and morphological changes?
3. Can a strategy be defined for using models of different resolution for different project questions?

There are a number of state-of-the-art numerical modeling tools that are available to predict scour and morphological changes. However, these recently advanced models are rarely used for practical applications by engineers for LWD and in-stream structure design as they were developed mostly for research purpose. In addition, most are very difficult to use or are not developed in the desktop computing environment.

## 3. Scope of the Proposed Scoping Study

During the scoping study, we will carry out the following steps in order to answer the research questions raised above:

### 3.1 Collaborators and Potential Modeling Tools

Initially, we will work with a number of collaborators to identify potential modeling tools. Collaborators will include Professor Fotis Sotiopoulos at the University of Minnesota, Professor Peter Wilcock at the Johns Hopkins University, and Professor Jennifer Duan at the University of Arizona. Modeling tools may include:

- U2RANS: A three dimensional (3D) numerical model developed at the University of Iowa and modified and improved for realistic river flows at Reclamation.
- SRH-2D: A Reclamation two-dimensional (2D) hydraulic and sediment transport model widely used at Reclamation and by external universities, research institutions and consulting companies.
- CART3D: A NASA developed public domain model suitable for automated generation of 3D meshes to represent complex shapes of LWD and other in-stream structures.
- LES Models: High-resolution 3D large eddy simulation (LES) models may be included through collaboration with Prof. Sotiropoulos.
- Deflt3D: A recently released 3D model for river flow and sediment transport. It may be evaluated through collaboration with Prof. Duan.

Other models may be identified in the course of the study

### 3.2 Demonstration of Modeling Tools

A test case will be selected to evaluate selected modeling tools. The objective is to evaluate the suitability and accuracy of the models in predicting scour and channel morphology, and ease of use for engineers to carry out design studies in a desktop computing environment. Different usage of models at different resolutions will also be explained for different project questions.

### 3.3 Documentation

In the following we will document the findings of our scoping study and provide recommendations for future study.

## 4. Results of the Scoping Study

The scoping study has been progressed and completed as planned. A number of existing modeling tools are selected and studied; they include U$^2$RANS, SRH-2D, CART3D, VSL3D, and Deflt3D. During the study, another program is added for study: OpenFOAM. Each program is discussed below.

### 4.1 SRH-2D and U$^2$RANS

SRH-2D is a two-dimensional (2D) depth-averaged hydraulic and sediment transport model for river systems developed at the Bureau of Reclamation. It has been widely used for engineering projects at Reclamation and by outside institutions. SRH-2D has a few salient features making it ideal for engineering applications. First, SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. In practice, the hybrid mesh of quadrilateral and triangular cells is normally adopted that uses quadrilaterals in the main stream and near structures and triangles in the floodplain and transition zones. The hybrid mesh achieves the best compromise between accuracy and computing efficiency, and it is relatively easy to generate with SMS software. Second, SRH-2D adopts very robust (stable) numerical schemes with a seamless wetting-drying algorithm. Reliable and stable solutions may be obtained with few tuning parameters. Third, SRH-2D has been developed with the objective of ease-of-use. Users do not have to memorize many commands; they are guided by a preprocessor in a question-and-answer session. The preprocessor provides also guidelines on how to select input parameters. SRH-2D model, along with its manual and selected publications, are freely downloadable at the following Reclamation site: http://www.usbr.gov/pmts/sediment. Details of SRH-2D have been well documented and will not be repeated here. The primary use of SRH-2D is to predict the water surface elevation for river engineering applications so that the results may be used by U$^2$RANS to specify the water surface elevation.

U$^2$RANS is a three-dimensional (3D) Unsteady and Unstructured Reynolds Averaged Navier-Stokes solver. The code was developed by Dr. Yong Lai while he was appointed as the senior research staff and adjunct associate professor at the Iowa Institute of Hydraulic Research, University of Iowa. The model is highly accurate, well verified and validated, and has been successfully applied to many research and engineering projects.

Briefly, U$^2$RANS is a comprehensive general-purpose model. Three-dimensional hydraulic flow models such as U$^2$RANS are accurate and mature tools, which have been routinely used to address many hydraulic engineering problems such as:

- Flow hydrodynamics in pools and river reaches upstream of hydropower dams;
- Detailed flow characteristics around hydraulic structures;
- Hydraulic impact of different project alternatives;
- Fish passage facility design and evaluation;

- Thermal mixing zone determination;
- Design optimization, reservoir/lake stratification, selective cold water withdrawal, etc.

The main limitation is that they are usually applied to a river reach less than five miles in length due to their heavy requirement for computer power.

$U^2$RANS uses state-of-the-art, unstructured CFD technology, unifies multi-block structured mesh (quad or hex) and unstructured mesh (quad, triangle, tet, hex, wedge, pyramid, or hybrid) elements into a single platform, and combines 2D and 3D solvers in a common framework. A draft User's Manual is available, which provides a more detailed description about the general features and capabilities (Lai, 20023). Many difference physical processes may be modeled and sample processes include:

- Accurate solution of full three-dimensional water flows with complex geometry (the Navier-Stokes equations)
- 3D effects, such as secondary flows at the meandering bends and point bars, vortex/eddy generation due to hydraulic structures, vertical flow characteristics, are accurately captured; and
- Water temperature transport is simulated using the energy conservation equation

The current version lacks the mobile-bed sediment transport simulation. The sediment capability, however, is under development, which is funded by Taiwan Water Resources Agency (WRA).

The model inputs include detailed bathymetric data and hydraulic structure geometric data, river discharge, and water surface elevation at the downstream boundary. The model output include the 3D spatial distribution of velocity magnitude and flow direction, location and strength of flow eddies and vortices, secondary flows due to meandering, bed shear stresses, water surface elevation distribution and backwater effect. The potential use of output results include the evaluation of erosion/deposition potential at the point bar due to secondary flows, assessment of scouring potential due to hydraulic structures, hydraulic impact assessment of modified or new structures, etc.

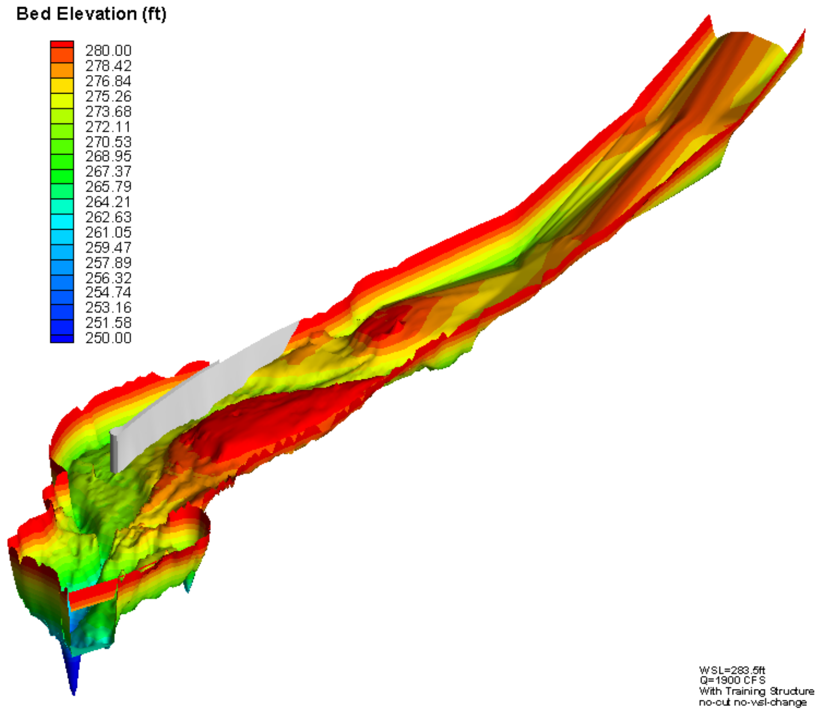
Numerous applications have been made and some are documented in many publications. $U^2$RANS related papers are listed below, with the first five papers discussing the basic theory and numerical procedures.


- Lai, Y.G. (2002). "User's Manual for U2RANS: An Unsteady and Unstructured Reynolds Averaged Navier-Stokes Solver – A Draft," IIHR draft document, University of Iowa.
- Lai, Y.G., Weber, L.J., Patel, V.C. (2003a). "A non-hydrostatic three-dimensional method for hydraulic flow simulation - part I: formulation and verification," ASCE Journal Hydraulic Engineering, 129(3), 196-205.

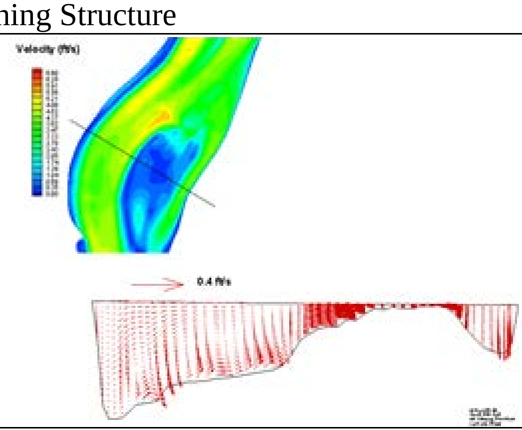
- Lai, Y.G., Weber, L.J., Patel, V.C. (2003b) “A non-hydrostatic three-dimensional method for hydraulic flow simulation - part II: application,” ASCE Journal Hydraulic Engineering, 129(3), 206-214.
- Lai, Y.G. (2000). “Unstructured grid arbitrarily shaped element method for fluid flow simulation,” AIAA Journal, 38 (12), 2246-2252.
- Lai, Y.G., Weber, L, Patel, V.C. (2000). U2RANS: a comprehensive hydraulic flow simulation code – its development and application, Proceedings 4th Int. Conference on HydroInformatics, Iowa City, IA.
- Lai, Y.G. (1997) “An unstructured grid method for a pressure-based flow and heat transfer solver,” Numerical Heat Transfer, Part B, 32, 267-281.
- Li, S., Lai, Y.G., Weber, L.J., Silva, J.M., Patel, V.C. (2003) “Validation of a three-dimensional model for water-pump intakes”, to appear in IAHR Journal Hydraulic Research.
- Lai, Y.G., Weber, L.J. (2001). Three-dimensional hydraulic simulation of wetting-drying process, 29th IAHR Congress, Beijing, China, Sept. 16-21, 2001.
- Weber, L.J., Huang, H., Lai, Y.G. (2001). Numerical modeling of total dissolved gas downstream of a spillway, 29th IAHR Congress, Beijing, China, Sept. 16-21, 2001.
- Goodwin, R.A., Nestler, J.M., Weber, L.J., Lai, Y.G., Loucks, D.P. (2001). Ecologically sensitive hydraulic design for rivers: lessons learned in coupled modeling for improved fish passage, Proc. ASCE Specialty Conference on Wetlands Engineering and River Restoration, Reno, NV, August 25-31, 2001.
- Ettema, R., Chen, Z., Lai, Y.G. (2001). Hydraulic performance of offshore water intakes: some CFD findings,” Proc. Ports and Ocean Engineering under Arctic Conditions Conference, Ottawa Canada, August, 2001.
- Lai, Y.G. and Patel, V.C. (2001). Effect of boundary conditions on simulation of flow in the T99 draft tube, Turbine99-2nd ERCOFTAC Workshop on Draft Tube Flow, Alvkarleby, Sweden, June 17-20 2001.
- Lai, Y.G. and Patel, V.C. (1999). CFD simulation and assessment of the draft tube flow,” Turbine-99, ERCOFTAC Workshop on Draft Tube Flow, Porjus, Sweden, June 1999.
- Weber, L.J., Lai, Y.G., Blank, J.C., Andrade, F.De. (2000). Rocky Reach Dam: a comprehensive look at the calibration of a numerical model applied to fish passage,” Proceedings 4th Int. Conference on HydroInformatics, Iowa City, IA.
- Lai, Y.G., Weber, L.J., Moedinger, J. (2001). A three-dimensional unsteady method for simulating river flows, Proceedings ASCE World Water and Environmental Resource Congress, Orlando, FL, May 20-24, 2001.
- Weber, L.J., Lai, Y.G., Andrade, F. (2001). Three-dimensional numerical model validation: issues and directions, Proceedings ASCE World Water and Environmental Resource Congress, Orlando, FL, May 20-24, 2001.

- Huang, W. and Lai, Y.G. (2000). A parallel implementation of a multi-block three-dimensional incompressible flow solver on a DSM machine, Proceedings 4th Int. Conference on HydroInformatics, Iowa City, IA.
- Li, S., Lai, Y.G., Patel, V.C., Silva, J. M. (2000). Water-intake pump bays: three-dimensional flow modeling, validation, and application, Proceedings 4th Int. Conference on HydroInformatics, Iowa City, IA.

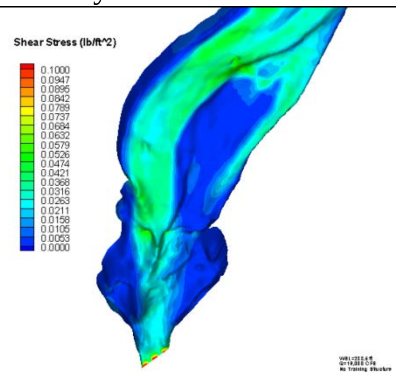
A specific application of U²RANS is to a river reach upstream of the Palo Verde Diversion Dam on the Colorado River. The model was used to simulate the flow near a point bar and to evaluate the impact of installing a training structure to protect the bank erosion. Model outputs, which are illustrated in the Figure include geometry with the training structure, simulated velocity and secondary flow, and the predicted bed shear stress.




(a) Geometry with Training Structure



(b) Simulated Velocity and Secondary Flow




(c) Predicted Bed Shear Stress

Figure 1. Example output from U2RANS Model Application on Colorado River

## 4.2 CART3D

Cart3D is a high-fidelity inviscid analysis package for conceptual and preliminary aerodynamic design. It allows users to perform automated CFD analysis on complex geometry. The package includes utilities for geometry import, surface modeling and intersection, mesh generation, flow simulation and post-processing of results. The geometry and mesh generation portion of Cart3D is our interest as the package is highly automated so that geometry acquisition, and mesh generation can usually be performed within a few minutes on most current desktop computers.

Geometry enters into Cart3D in the form of surface triangulations. These may be generated from within a CAD packages, from legacy surface triangulations or from structured surface grids. Cart3D uses adaptively refined Cartesian grids to discretize the space surrounding a geometry and cuts the geometry out of the set of "cut-cells" which actually intersect the surface triangulation. The flow solver is parallel and can take full advantage of multi-core and multi-cpu hardware.

Collaboration has been initiated and started with Professor Jennifer Duan at the University of Arizona. CART3D has been obtained from NASA and the model was installed at both the University of Arizona by Prof. Duan and at Reclamation. Prof. Duan and her students have been able to

- Establish the procedure to run CART3D
- Develop the necessary tools (Matlab) to generate CAD objects as inputs to CART3D; and
- Demonstrate the mesh generation process with CART3D using a simple test case: the flow around a sphere.

## 4.3 VSL3D

A conference call was made in January, 2013 with Professors Fotis Sotiropoulos at the University of Minnesota and Peter Wilcock at the Johns Hopkins University. Both agreed to provide in-kind service to the study; the objective is to see whether VSL3D, a high-resolution 3D large eddy simulation (LES) model developed by Prof. Sotiopoulos will be valuable to the LWD study. Basic information about the model and selected study cases are discussed below.

The computational framework is referred to as the SAFL Virtual StreamLab, the so-called VSL3D (see http://cfdlab.safl.umn.edu). Professor Fotis Sotiropoulos at the St. Anthony Falls Laboratory (SAFL) of University of Minnesota is the key developer. Since 2005, with NSF funding grants (EAR-0120914 (as part of the National Center for Earth-Surface Dynamics) and EAR-0738726) Sotiropoulos and co-workers at the St. Anthony Falls Laboratory (SAFL) have developed and validated advanced computational fluid dynamics (CFD) tools enabling multi-resolution simulations of turbulence and transport phenomena in real-life natural waterways with embedded arbitrarily complex hydraulic structures at an unprecedented level of resolution and degree of realism.

VSL3D can: 1) resolve turbulence in arbitrarily complex waterways with embedded natural and/or man-made hydraulic structures using unsteady Reynolds-averaged Navier-Stokes (URANS) or Large-Eddy Simulation (LES) turbulence models (Kang et al. 2011; Kang and Sotiropoulos 2011, 2012a); 2) account for the presence of the water-surface with level-set method in a coupled, fully non-linear manner (Kang and Sotiropoulos 2012b); and 3) simulate sediment transport (suspended- and bed-load), conservative /non-conservative contaminants and nutrients transport, density current, and morphodynamics in real-life waterways and past hydraulic structures with complex geometries (Escauriaza and Sotiropoulos 2011a,b; Khosronejad et al. 2011; 2012a; 2012b).

Two types of inputs data are required for VSL3D model to simulate a specific test case which includes geometrical data of the river and immersed subjects and model setup information. The geometrical information describes the geometry of waterway and/or any immersed subject in the simulation domain (e.g. river's bed bathymetry and mounted bridge piers or LWD). Various types of scanned or surveyed geometrical data can be imported toVSL3D. The setup information are required to be introduced to the model through a text file which will be read at the beginning of simulations and includes information about the type of desired turbulence model to be used, time step, and duration of simulation.

The output variables of VSL3D consist of 3D flow velocity components, pressure, and shear stress on the stream bed, banks, and any other immersed subjects such as LDW as a function of time. 3D distribution of bed- and suspended sediment load in the computational domain plus the bathymetry of the water way at equilibrium can be simulated by VSL3D. VSL3D is also capable of simulating the transport of any scalar term including pollutions, nutrients, and so on in the real-life stream environments.

The VSL3D employs a novel computational approach based on the Curvilinear Immersed Boundary Method (CURVIB) (Ge and Sotiropoulos 2007; Borazjani et al. 2008), which can account for arbitrarily complex domains with embedded arbitrarily complex rigid (e.g. hydraulic structures) or dynamically evolving boundaries that interact with the flow in a fully coupled manner (e.g. the sediment-water and/or the air-water interfaces in rivers). The method has been extended by Kang et al. (2011) to carry out URANS and LES of turbulent flows in natural waterways using wall models for bridging the gap between the first grid node in the fluid domain and the wall. The VSL3D model is fully parallelized using Message Passing Interface (MPI) and scales efficiently on thousands of CPUs. The predictive capabilities of the model in simulating flows in natural streams were demonstrated through extensive comparisons with laboratory and field scale measurements in Kang et al. (2011) and Kang and Sotiropoulos (2012a, b). Kang and Sotiropoulos (2012b) extended the VSL3D to incorporate a two-phase flow formulation with level-sets to enable coupled simulations of free-surface effects in turbulent flows over arbitrarily complex bathymetry. Khosronejad et al. (2011; 2012; 2013) developed the coupled hydro-morphodynamic version of the VSL3D and applied it to simulate streambed erosion and scour under clear water conditions in curved open channels (Khosronejad et al 2011) and past bridge piers (Khosronejad et al. 2012) and stream-restoration rock structures (Khosronejad et al. 2013). More recently the VSL3D has been further extended to simulate sediment transport under live-bed conditions and has been shown to be able to simulate realistic bedforms across a range of scales: from cm-scale ripples in a laboratory flume to tens of meters scale dunes in large meandering rivers.

In the following a few applications of VSL3D are presented. Figure 1 illustrates the capability of the VSL3D to carry out quantitatively accurate simulations of ripples (amplitude $\Delta$~5 cm, wave-length $\lambda$~10 cm) in a 1 m wide and 0.1 m deep laboratory flume studied experimentally by Venditti et al. (2005). Highly resolved LES on a grid with 30 million grid nodes was required to excite the bed instability at this scale and obtain these results. Figure 2, on the other hand, illustrates the ability of the method to simulate the growth and downstream migration of dunes (($\Delta$~12 cm, $\lambda$~50 cm) in a 2-m wide and 0.3 m deep field-scale meandering stream (the SAFL Outdoor StreamLab) with three bendway weirs made of rocks installed along the outer bank to suppress streambank erosion. Relatively coarse grid LES on a grid with 5 million nodes was sufficient for these simulations presumably due to the large disturbance imposed on the bed by the structures and complex bathymetry. Finally, Figure 3 shows a snapshot of simulated bedforms in a 27 m wide and 0.9 m deep meandering river. A major finding from these results is that at this scale the URANS version of the VSL3D on grids with 2-3 million nodes is sufficient for exciting the bed instability and giving rise to realistic bed forms that look similar to those observed in nature (($\Delta$~0.5-1.5 m, $\lambda$~10-30 m). By placing the computed bed-form characteristics on Van Rijn's (1984) classical diagrams, we find that the computed bedforms are dunes with dimensions similar to those measured in the field.

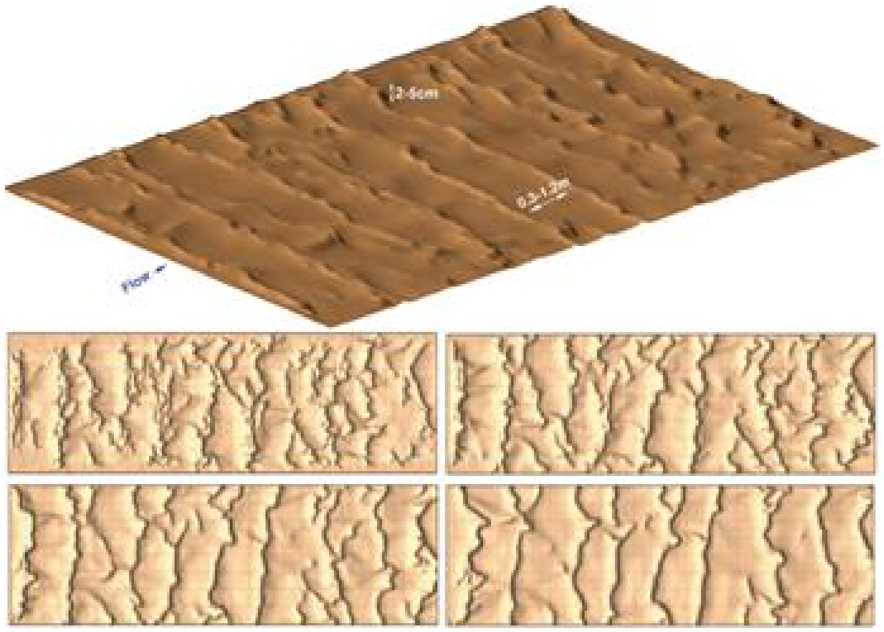



Figure 1. Coupled hydro-morphodynamics LES for the Venditti et al (2005) rectangular flume experiment. Top: Instantaneous 3D view of the calculated ripples after about 10 min. Bottom: top view of calculated bedforms at four instant in time (time increases from up-left to bottom-right). Ripple-like sand waves grow in length and amplitude, propagate downstream and merge to form larger bedforms. The flow is from left to right.

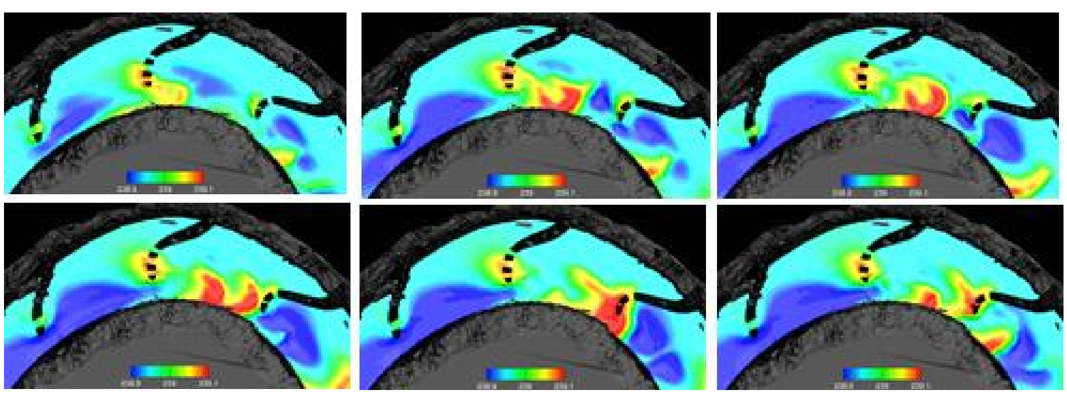


Figure 2. Coupled hydro-morphodynamics LES of turbulent flow in a mobile-bed natural meandering stream with 3 J-hook vanes. Contour plots of bed elevation are shown at three instant in time (time increases from top-left to bottom-right). Mini-dune-like bedforms grow, propagate downstream and merge to form larger bedforms. Flow is left to right.

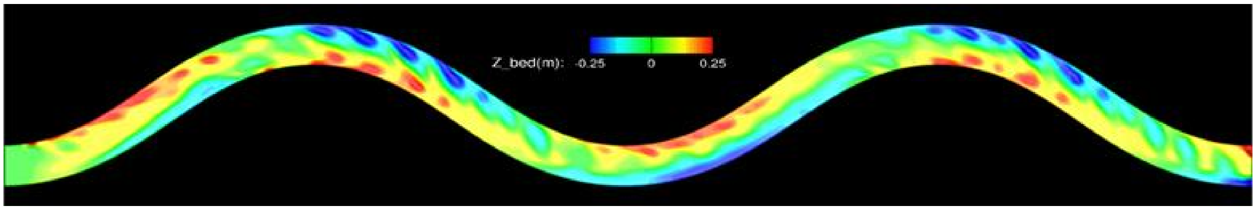



Figure 3. A snapshot of simulated bedforms (mega-dunes) in a live-bed field-scale meandering gravel-bed River. As shown, contours are in meter and flow is left to right.

While several other numerical methods have been proposed in the literature to carry out RANS and LES of open channel flows, to the best of our knowledge the VSL is the most complete computational framework of its kind today. The model is unique in its ability to carry out coupled hydro-morphodynamic simulations in real-life waterways with arbitrarily complex hydraulic structures on massively parallel supercomputers and at a desired level of user-specified resolution. Computational models of this level of sophistication have yet to be translated from the academic research arena to the hydraulic engineering practice, which, for the most part, relies almost exclusively today on simple 1D and 2D models. Yet the efficiency of the VSL coupled with the exponential growth of computing power pave today the way for transforming hydraulic engineering practice by adopting such sophisticated computational approaches to tackle real-life problems.

Some of the references of this study are as follows:

Borazjani, I., Ge, L., and Sotiropoulos, F. (2008). "Curvilinear immersed boundary method for simulating fluid structure interaction with complex 3D rigid bodies." *Journal of Computational Physics*, 227, 7587–7620.

Ge, L. and Sotiropoulos, F. (2007). "A numerical method for solving the 3D unsteady incompressible Navier Stokes equations in curvilinear domains with complex immersed boundaries." Journal of Computational Physics, 225, 1782–1809.

Gilmanov, A. and Sotiropoulos, F. (2005). "A hybrid Cartesian/immersed boundary method for simulating flows with three-dimensional, geometrically complex, moving bodies." *Journal of Computational Physics*, 207, 457.

Kang, S., Lightbody, A., Hill, C. and Sotiropoulos, F. (2011), "High-resolution numerical simulation of turbulence in natural waterways," *Advances in Water Resources*, Volume 34, pages 98-113.

Kang, S. and Sotiropoulos, F. (2011), "Flow phenomena and mechanisms in a field-scale experimental meandering channel with a pool-riffle sequence: Insights gained via numerical simulation," *J. of Geophysical Research*, Volume 116, F03011.

Kang, S., Borazjani, I., Colby, J. A. and Sotiropoulos, F. (2012), "Numerical simulation of 3D flow past a real-life marine hydrokinetic turbine," *Advances in Water Resources*, Volume 39, pages 33-43.

Kang, S. and Sotiropoulos, F. (2012a), "Assessing the predictive capabilities of isotropic, eddy-viscosity Reynolds-averaged turbulence models in a natural-like meandering channel," *Water Resources Research*, Volume 48, W06505, doi:10.1029/2011WR011375.

Kang, S. and Sotiropoulos, F. (2012b), "Numerical modeling of 3D turbulent free surface flow in natural Waterways," *Advances in Water Resources*, Volume 40, pages 23-36.

Kang, S., Khosronejad, A., and Sotiropoulos, F., (2012) "Numerical simulation of turbulent flow and sediment transport processes in arbitrarily complex waterways," Environmental Fluid Mechanics,

Memorial Volume in Honour of Prof. Gerhard H. Jirka, Eds. W. Rodi & M Uhlmann, CRC Press (Taylor and Francis group), pp. 123-151.
Khosronejad, A., Kang, S., Borazjani, I. and Sotiropoulos, F. (2011), “Curvilinear immersed boundary method for simulating coupled flow and bed morphodynamic interactions due to sediment transport phenomena,” *Advances in Water Resources*, Volume 34, pages 829-843.
Khosronejad, A., Kang, S. and Sotiropoulos, F. (2012), “Experimental and computational investigation of local scour around bridge piers,” *Advances in Water Resources*, Volume 37, pages 73-85.
Khosronejad, A., Hill, C., Kang, S., and Sotiropoulos, F. (2013), “Computational and experimental investigation of scour past laboratory models of stream restoration rock structures,” *Advances in Water Resources*, Volume 54, pages 191-207.

## 4.4 Deflt3D

Collaboration has been initiated and started with Professor Jennifer Duan at the University of Arizona. Dr. Duan has evaluated Deflt3D, a recently released 3D model for river flow and sediment transport, using a number of application cases.

Delft3D is developed by Delft Hydraulics. Its application areas include the following:

- Tide and wind-driven flows (i.e.. storm surges)
- Stratified and density driven flows
- River flow simulations
- Simulations in deep lakes and reservoirs.
- Simulation of tsunamis, hydraulic jumps, bores and flood waves
- Fresh-water river discharge in bays
- Slat intrusion
- Thermal stratification in lakes, seas and reservoirs
- Cooling water intakes and waste water outlets
- Transport of dissolved material and pollutants
- Sediment transport and morphology
- Wave-driven currents
- Non-hydrostatic flows

Delft3D-FLOW is an open source program (http://oss.deltares.nl/web/opendelft3d/source-code). Delft3D model suite (or software) consists of six modules: Deflt3D-flow, Delft3D-Wave, Delft3D-Part, Delft3D-Eco, Delft3D-Sed. The Delft3D-flow is the hydrodynamic module that provides hydrodynamic conditions including velocities, water elevations, density, salinity, vertical eddy viscosity, and vertical eddy diffusivity to other modules. Delft3D-FLOW is capable of simulating three dimensional (3D) unsteady incompressible flow and transport phenomena resulting from tidal and/or meteorological forcing.

Delft3D-flow solves the Navier-Stokes equations for an incompressible fluid under the shallow water and the Boussinesq assumptions. In the vertical momentum equation the vertical accelerations are neglected that leads to the hydrostatic pressure equation. In the 3D models the vertical velocities are computed from the continuity equation.

In the horizontal direction Delft3D-flow uses orthogonal curvilinear co-ordinates. Two coordinate systems are supported: 1) Cartesian co-ordinates (ξ, η) and 2) spherical co-ordinates (λ, φ). The vertical coordinates can be 1) σ-grid (body-fitted grid); or 2) Z-grid (Cartesian uniform grid). The governing equations are written in curvilinear coordinate. The continuity equation is written as:

$$\frac{\partial \zeta}{\partial t}+\frac{1}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\left[(d+\zeta)U\sqrt{G_{\eta\eta}}\right]}{\partial \xi}+\frac{1}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\left[(d+\zeta)V\sqrt{G_{\xi\xi}}\right]}{\partial \eta}=Q \qquad (1)$$

in which $Q=H\int_{-1}^{0}(q_{in}-q_{out})d\sigma+P-E$

where Q representing the contributions per unit area due to the discharge or withdrawal of water, precipitation and evaporation, $q_{in}$, $q_{out}$ are local sources and sinks of water, P is non-local precipitation, E is non-local evaporation.

The momentum equations in ξ- and η- direction are given by:

$$\begin{aligned}&\frac{\partial u}{\partial t}+\frac{u}{\sqrt{G_{\xi\xi}}}\frac{\partial u}{\partial \xi}+\frac{v}{\sqrt{G_{\eta\eta}}}\frac{\partial u}{\partial \eta}+\frac{w}{d+\zeta}\frac{\partial u}{\partial \sigma}+\frac{v^2}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\sqrt{G_{\eta\eta}}}{\partial \xi}\\&+\frac{uv}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\sqrt{G_{\xi\xi}}}{\partial \eta}-fv=-\frac{1}{\rho_0\sqrt{G_{\xi\xi}}}P_\xi+F_\xi+\frac{1}{(d+\zeta)^2}\frac{\partial}{\partial \sigma}(\nu_v\frac{\partial u}{\partial \sigma})+M_\xi\end{aligned} \qquad (2)$$

$$\begin{aligned}&\frac{\partial v}{\partial t}+\frac{u}{\sqrt{G_{\xi\xi}}}\frac{\partial v}{\partial \xi}+\frac{v}{\sqrt{G_{\eta\eta}}}\frac{\partial v}{\partial \eta}+\frac{w}{d+\zeta}\frac{\partial v}{\partial \sigma}+\frac{uv}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\sqrt{G_{\eta\eta}}}{\partial \xi}\\&-\frac{u^2}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\sqrt{G_{\xi\xi}}}{\partial \eta}+fu=-\frac{1}{\rho_0\sqrt{G_{\eta\eta}}}P_\eta+F_\eta+\frac{1}{(d+\zeta)^2}\frac{\partial}{\partial \sigma}(\nu_v\frac{\partial v}{\partial \sigma})+M_\eta\end{aligned} \qquad (3)$$

where $\nu_v$ is the vertical eddy viscosity coefficient and defined as

$$\nu_v=\nu_0+\max(\nu_{3D},\nu_v^{back}) \qquad (4)$$

where $\nu_0$ =kinematic viscosity, $\nu_{3D}$ = turbulence viscosity calculated from 3D model; $\nu_v^{back}$ =ambient or background viscosity set by the user.

The vertical velocity is solved from the continuity equation as:

$$\frac{\partial \zeta}{\partial t}+\frac{1}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\left[(d+\zeta)u\sqrt{G_{\eta\eta}}\right]}{\partial \xi}+\frac{1}{\sqrt{G_{\xi\xi}}\sqrt{G_{\eta\eta}}}\frac{\partial\left[(d+\zeta)v\sqrt{G_{\xi\xi}}\right]}{\partial \eta}+\frac{\partial w}{\partial \sigma}=H(q_{in}-q_{out}) \quad (5)$$

The flow model can also simulate floating structures, such as ship, or floating shafts.

Delft3D-FLOW offers two different vertical grid systems σ-grid and Z-grid, and four turbulence closure models: constant eddy viscosity coefficient, algebraic eddy viscosity model, *k*-*L* model, and *k*-*ε* model. Delft3D-FLOW also has a facility to define the bed and flow resistance on each sub-grid using a function called Trachytopes. Three classes are available in the Trachytopes function, area class, line class and point class. The area class has three types: the first type is a constant coefficient for bed roughness, such as White Colebrook, Chezy, and Manning's coefficients, the second type accounts for the form resistance resulting from sand dunes, and the third type is for the roughness coefficient in vegetated channels. The first type often remains a constant with time, while the second type is determined by dune height, and the third by vegetation properties. The line class of Trachytopes function can be used to approximate flow resistance for elements with hedge, bridge piers, and other structures. The point class can be used to represent a set of point flow resistance elements, such as groups of individual tree or smaller scale plant,

Geometric data of simulation domain including bathymetry and boundaries, boundary friction coefficient (e.g., Manning's n values), bed sediment gradation, cohesive or non-cohesive sediment, upstream and downstream boundary conditions for flow (e.g. flow discharge, water surface elevation), upstream boundary conditions for sediment load.

Delft3D-Flow solves the Reynolds Averaged Navier-Stokes equations on a structured staggered curvilinear grid using a finite difference scheme (Stelling and van Kester 1994). The governing equations are solved with an Alternating Direction Implicit (ADI) technique (Stelling 1984).

A number of cases have been simulated with Delft3D. The first case is to simulate the vegetation impacts on flow field. The case has been documented by a paper of Khalid and and Duan, "Case Study: Three-dimensional Hydrodynamic Simulation of Tidal Flow through a Vegetated Marsh Area," which was submitted to J. Hydraulic Engineering for review.

The second case is to model sediment plug occurring in Rio Grande. The case is documented in a manuscript by Posner and Duan, "3D numerical modeling of sediment plugs in Rio Grande River," which is submitted to Earth Surface Process and Land Forms for review.

The third case is to simulate Lake Mills Drawdown Experiments. This study is funded by Reclamation and results have been documented in a project report by Khalid and Duan, "Simulation of Lake Mills Drawdown Experiments using Delft3D."

## 4.5 OpenFOAM

A new collaboration has been established in the course of this study with Prof. Xiaofeng Liu at the University of Texas at San Antonio. Prof. Liu is an expert in 3D computational fluid dynamics (CFD) model development and applications for river engineering. He builds his models around an open-source, publically available model, OpenFOAM. Prof. Liu developed a novel immersed boundary technique (IBT) that takes complex geometries into consideration automatically by the CFD model. The IBT has high potential to be suitable for modeling flows around LWD structures and it is the reason Prof. Liu was invited to participate in the present study.

Dr. Liu has been using OpenFOAM® for almost ten years. The history of Dr. Liu's involvement of the open source platform OpenFOAM® goes back to his Ph.D. thesis work at University of Illinois at Urbana-Champaign. His doctoral research studied the flow field and sediment scour around structures such as bridge piers and offshore foundations. He also used OpenFOAM® to study the erosion problem in the St. Clair River in Great Lakes region. Dr. Liu is now an assistant professor at University of Texas at San Antonio. He continues to use OpenFOAM® as one of the major tools in his research. Besides doing the research using OpenFOAM® by himself, he has also shared his experience with the research community through workshops and short courses on OpenFOAM®.

Regarding OpenFOAM® itself, it is an open source computational physics platform developed by OpeonCFD Ltd at ESI group (http://www.openfoam.com/). It has gained a lot of popularity in many disciplines across most areas in engineering and science. It was originally designed for computational fluids. Now it has been used as a general framework for solving differential equations. It uses finite volume method as its main discretization scheme. Key required inputs by a user include bathymetry of the river section, geometry of the LWD or any other in-stream structures, inflow and outflow (discharge, stage, etc.), sediment composition in the stream, selection of turbulence models, selection of numerical schemes, etc. Key outputs include 3D flow field, shear stresses on the river bed and banks for the evaluation of erosion potentials, forces on the LWD or other structures for the evaluation of their stability.

OpenFOAM uses finite volume method to discretize governing equations. For 3D river flows, the equation it solves is the Navier-Stokes equation which describes the conservation of mass and momentum of fluids. For turbulent flows, which are usually the case for rivers and streams, it provides a library of turbulence models. It also provides parallel computation capability using MPI.

Some of the applications performed by Prof. Liu are discussed below.

Local scour around object was carried out. Dr. Liu implemented a 3D local scour simulation solver in OpenFOAM, which has three components: the flow field solver, the sediment transport solver, and the mesh deformation solver. The flow field solver comes with the OpenFOAM package and the user can choose from a library of turbulence models (RANS or LES). It also has the capability for free surface flows. The fluid solver simulates the flow field and gives the bottom shear stresses. With the results from the flow solver, sediment transport (both bed load and suspended load) can be solved. Erosion and deposition will change the bathymetry around the object, which changes the computational domain. Since unstructured mesh is used in OpenFOAM, a special solver is designed to automatically deform the mesh. The details can be found in Liu and García (2008).

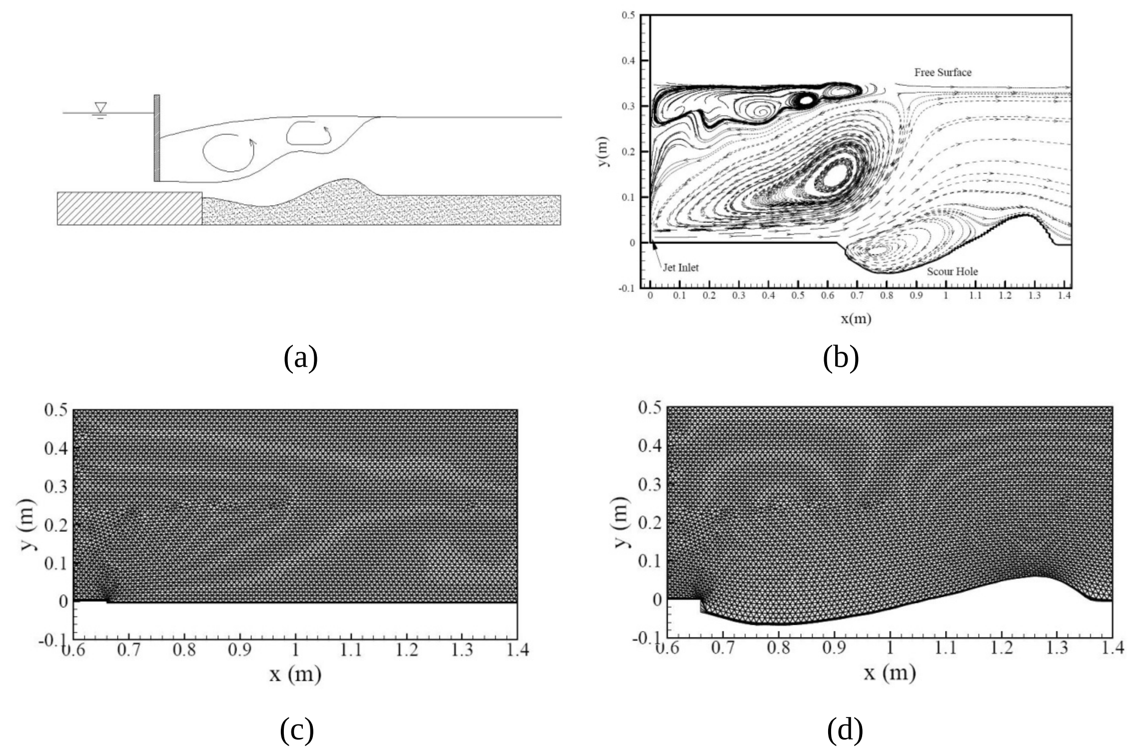



Figure 4. Turbulence wall jet scour simulation using OpenFOAM (Liu and García, 2008): (a) Scheme of the case, (b) Flow field, (c) Initial mesh, and (d) Final mesh.

Hydrodynamics in large rivers is simulated using OpenFOAM for bend flows in the St. Clair River for the evaluation of scour and erosion potentials. In this study, the morphological change was not activated. Instead, only the hydrodynamics were simulated with RANS turbulent model. The bed shear stress was evaluated at different flow discharges (low, medium, and high) to check the potential of erosions. The details can be found in Liu et al. (2011).

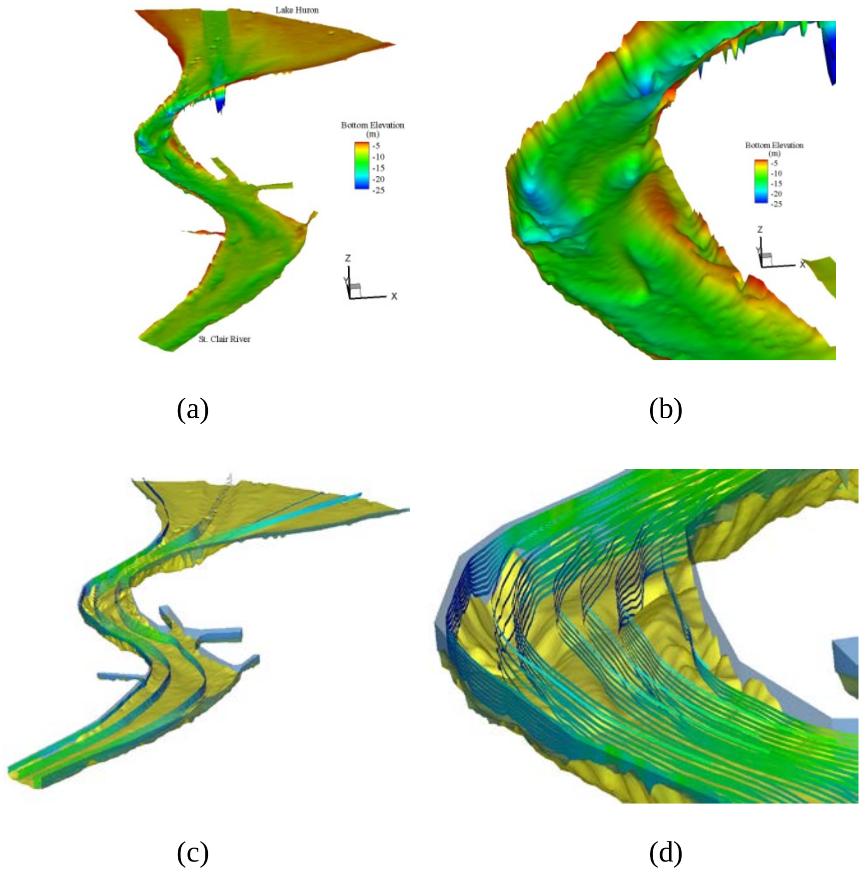



Figure 5. OpenFOAM simulation of 3D hydrodynamics in a large river (Liu et al., 2011): (a) Overall bathymetry of the St. Clair River entrance, (b) Local bathymetry in the bend, (c) Overall flow streamlines, and (d) Flow streamlines over the dunes in the bend.

Scour protection assessment is modeled using OpenFOAM. A new immersed boundary method is developed to do pore-scale modeling of ripraps and other porous scour protections. The details can be found in Liu et al. (2012) and similar idea has also been documented in Nielsen et al. (2013).

The major incentive, similar to the idea of this proposal, is to relieve the burden of the user to generate meshes for complicated geometry and objects. In a typical design of using LWD for river restoration, the geometries of wood stems and other auxiliary components are so complicated that a body-fit mesh is almost impossible. The same situation happens in ripraps where large amount of armor units are loosely packed together.

Dr. Liu has developed an innovative way to handle problems like these. In his model, he used the collision detection and rigid body dynamics algorithms to generate the physical and realistic spatial arrangement of different objects (LWD or rocks) in the domain. This part of the model mimics the physical process of placing these objects in the rivers during construction.

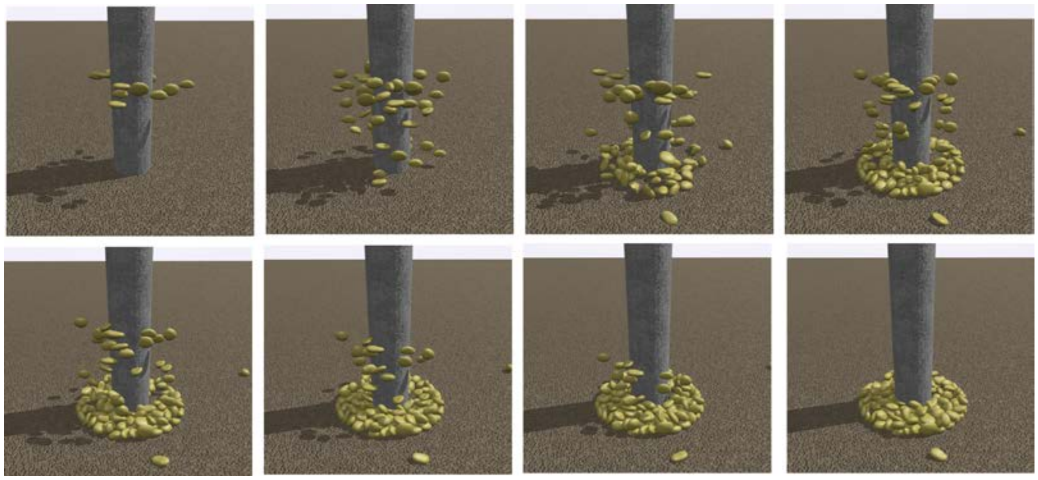


Figure 6. Computer simulation of the placing (dumping) process of ripraps rocks into the domain around a pile. This part of the model uses the collision detection and rigid body dynamics algorithms.

When the spatial arrangement of the objects is available, they are represented in the fluid solver model by the immersed boundary method. Again, there is no need to generate a body-fit mesh for them.

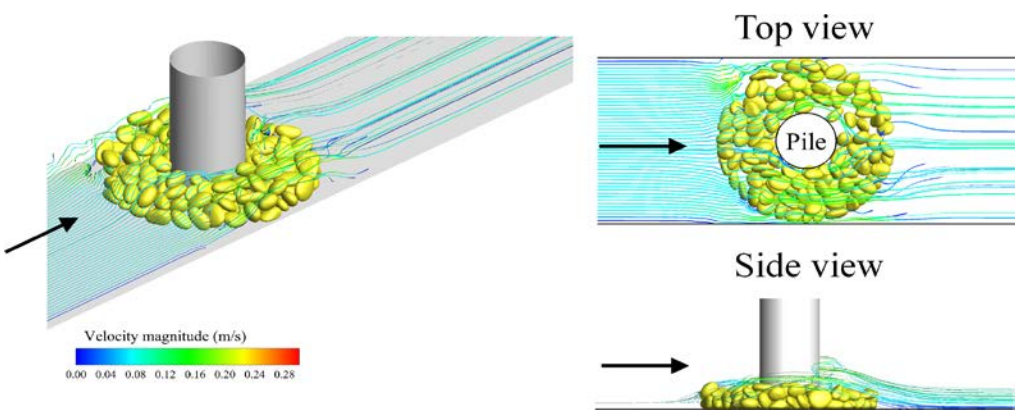



Figure 7. Simulated flow structure around the ripraps (scour protection) and the pile. Immersed boundary method was used and no body-fit mesh for the rocks is needed.

Some of the references of the study are included below:

- X. Liu and M.H. García (2008). A 3D Numerical Model with Free Water Surface and Mesh Deformation for Local Sediment Scour. *Journal of Waterway, Port, Coastal, and Ocean Engineering*. 134(4): 203-217.
- X. Liu, G. Parker, J. Czuba, K. Oberg, J.M. Mier, J.L. Best, D.R. Parsons, P. Ashmore, and M.H. Garcia (2011). Sediment Mobility and Bed Armoring in the St. Clair River: Insights from Hydrodynamic Modeling. *Earth Surface Processes and Landform*, 37(9):957-970.

- X. Liu, Y. Jiang, and R. Sinir, Simulation of flow field around and inside porous scour protection with physical and realistic particle configurations from computer graphics, Computational Methods in Water Resources XIX International Conference, Champaign, IL, 2012
- A.W. Nielsen, X. Liu, B.M. Sumer, J. Fredsoe (2013). Flow and Bed Shear Stresses in Scour Protections around a Pile in a Current, *Coastal Engineering*, 72:20-38

## 5. Future Recommendations

The scoping stud has evaluated five modeling tools as discussed above. The following are our recommendations for future full-proposal study.

$U^2$RANS and SRH-2D are developed by Dr. Yong Lai, the PI of this study. Both models have been updated and demonstrated. This scoping study finds that $U^2$RANS is potentially applicable to the LWD modeling since it adopts the arbitrarily shaped mesh elements and has the capability to import the complex cut cells produced by CART3D and perform the CFD modeling. $U^2$RANS is found to be practical and suitable for modeling flows around LWD structures. The morphological components of the SRH-2D model are found to be easily incorporated into $U^2$RANS so that the sediment transport and morphological modeling can also be carried out with $U^2$RANS. $U^2$RANS and SRH-2D will be included in the full-proposal study.

CART3D is a open source mesh generation software developed by NASA. CART3D has been installed at Reclamation. Dr. Lai has hired a student intern, Monica Sullivan, to work with CART3D. Despite some challenges, CART3D is shown to be an ideal candidate to achieve the LWD modeling purpose and it will be included in the full-proposal study. Continued collaboration with Prof. Duan is to be secured.

OpenFOAM and the immersed boundary technique (IBT) implemented in OpenFOAM has been developed by Prof. Liu. It is found to have great potential in modeling flows and sediment transport around the LWD structures. Major advantages of OpenFOAM include: easy mesh generation as no boundary meshes are needed; easy accessibility due to its open-source public-domain policy; and wide range of technical developments and supports in the future. IBT and OpenFOAM will be included in the full-proposal study and Prof. Liu will be the collaborator of the study effort.

VSL3D, developed by Prof. Fotis Sotiropoulos at the Univ. of Minnesota, has been demonstrated with several cases. Evaluation of the model shows that VSL3D is a very good and accurate modeling tool for flows around LWD structures; it is probably one of the best modeling tools available today. However, VSL3D requires the use of supercomputers, along with tremendous modeling expertise required of a user. The scoping study concludes that VSL3D is not yet practical for engineers, who have availability of only desktop computers, to use for project design purposes. The model, however, can be a candidate in the future when computing power has been drastically increased. VSL3D is not to be included in the full-proposal study.

Delft3D model has been evaluated by Prof. Duan using selected cases for its capabilities for morphological modeling. Our evaluation determines that Delft3D is very limited in its ability to simulate flows and sediment transport around LWD structures due to the use of restrictive meshes the model can handle. Delft3D is not to be included for the full-proposal study.