U.S. Department of the Interior
Bureau of Reclamation
Research and Development Office
Science and Technology Program

Robust Eco-Hydraulic 3D Modeling Tools for Rivers with Complex In-Stream Structures

Research Bulletin
S&T Project 1734

Mission Issue

Protect the river ecosystem by using the current state-of-the-art computational modeling tools.

Lead Researcher

Yong Lai
Hydraulic Engineer,
Technical Service Center
ylai@usbr.gov

Research Office Contact

Jennifer Bountry
Environmental Issues for Water Delivery and Management Research Coordinator, Technical Service Center
jbountry@usbr.gov

Problem

Providing large wood (LW) in a stream can help restore rivers and recover endangered species by reestablishing flow patterns and enhancing habitat. However, these LW features may cause scour and other undesirable changes—and these processes are not well understood. Methods and guidelines for determining risk and designing structures are unavailable. Traditional methods to measure and predict scour mostly rely on data from flumes, which suffers from scale problems. Field data for scour are scarce and expensive.

However, accurate flow and morphological predictions for these LW forms are becoming feasible with recent state-of-the-art modeling tools. These tools need to be evaluated and tested for LW structures to determine their risk, safety, and flow features. Computational fluid dynamics (CFD) modelling is an attractive alternative. However, CFD modelling of LW has its own challenges: the difficulty of mesh representation and lack of reliable and practical 3D flow and mobile-bed models.

Solution

A major challenge of high resolution LW simulation is the representation of complex geometry. It is well known that 3D mesh development for complex geometry can be very time consuming. Inadequate meshing practices can leave cells highly distorted, causing numerical errors and solution instability. Another challenge is the CFD solver itself with many 3D CFD models assuming fixed shapes of mesh cells. Our study tackles both challenges: (a) Develop a semi-automated 3D mesh generator that may represent complex LW shapes efficiently; (b) Develop an integrated 3D CFD model that can handle mesh cells with any shapes.

Our solution is summarized as: (1) Stream representation by a background mesh without LW. (2) LW geometry is scanned. (3) LW surface is processed into a 3D solid model. (4) LW is added to the background mesh. (5) A suitable 3D mesh is generated automatically. (6) Modeling with a specially developed 3D CFD model. (7) Results analysis.

"State of the art Computational fluid dynamics (CFD) modelling could help us understand how large wood structures work and interact in a river environment."

Yong G. Lai, Ph.D.
Hydraulic Engineer
Bureau of Reclamation

More Information

https://www.usbr.gov/research/projects/detail.cfm?id=1734

Application and Results

A structure consisting of six pieces of LW is used to demonstrate the modeling procedure and the modeling tools. The same structure is also constructed and placed in a flume so that experiment measurements may be conducted for model verification. Comparison between the model results and the measured data are made. Figure 1 compares the water surface elevation. The agreement is qualitatively good considering that the contours of the measured elevation are generated from a very scarcely distributed measured points (shown as black dots in the figure). Next the model predicted velocity near the water surface is also compared with the measured data in Figure 2. It is seen that the CFD model reproduces the measured data quite well.

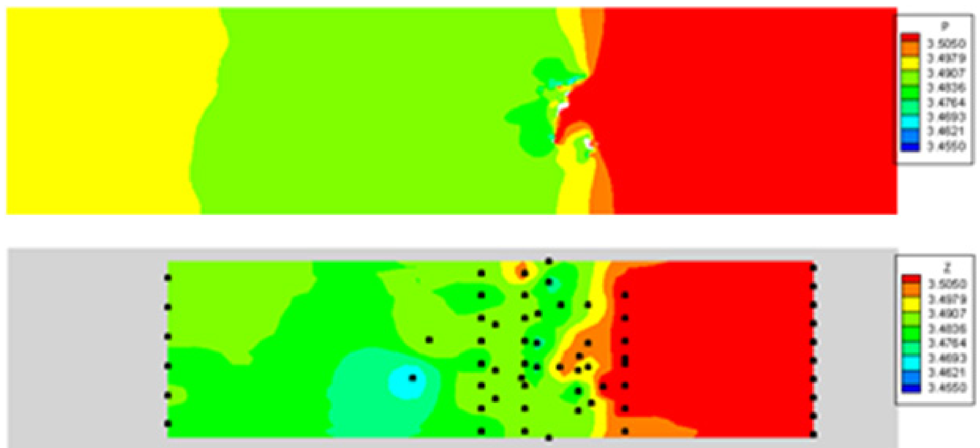

Figure 1. Comparison of predicted (top plot) and measured (bottom plot) water surface elevation. Measured zone is a smaller subset of the CFD modeling zone.

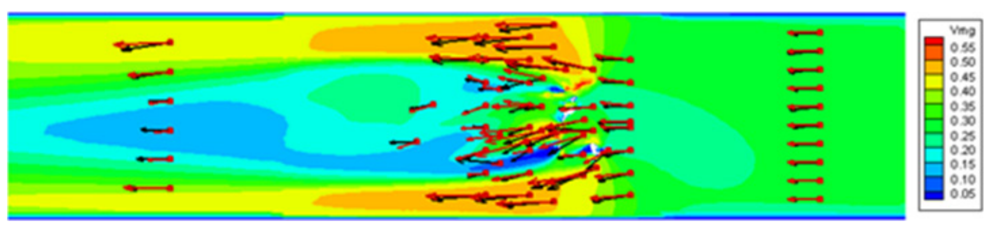

Figure 2. Comparison of predicted (red) and measured (black) velocity vector near water surface (background contours are the CFD predicted surface velocity).

Future Plans

We plan to continue this research in two directions. One is to develop a new sediment transport module into the 3D solver so that scour and morphological impacts of LW may also be simulated. Another is to develop new immersed boundary method into the existing flow solver so that the complex geometry of LW can be more easily handled by the flow solver, not the mesh generation.