TRANSPORTATION RESEARCH SYNTHESIS

Minnesota Department of Transportation
Office of Policy Analysis, Research & Innovation
Research Services Section
(651) 366-3780
www.research.dot.state.mn.us

TRS 1311
Published JUNE 2013

Applications of Computational Fluid Dynamics for River Simulation: State of the Practice

Abstract

The increasing availability of powerful and affordable computing platforms has enabled application of numerical modeling to solve flow problems in channels involving complex geometries and sediment transport. Computational Fluid Dynamics (CFD) typically pertains to using the full Navier-Stokes equations in three dimensions (3-D) to solve fluid flow problems. The purpose of this technical report is to assist the State in determining the state of the practice of CFD for river simulation. The work focuses on investigating the following:

- Finding out how CFD is being used to model river flows around piers; single and parallel bridges; ice and debris; bendway weirs and other river training structures.
- Determining if CFD has been used to simulate sediment transport and scour under bridges with and without piers.
- Identifying who (which state DOTs) uses CFD for the above mentioned purposes, what software is being used, what this software costs, and the cost to conduct studies of this kind.

APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS FOR RIVER SIMULATION: STATE OF THE PRACTICE

PROJECT INTRODUCTION AND BACKGROUND

The purpose of this technical report is to assist the State in determining the state of the practice of Computational Fluid Dynamics (CFD) for river simulation. The work focuses on investigating the following:

- Finding out how CFD is being used to model river flows around piers; single and parallel bridges; ice and debris; bendway weirs and other river training structures.
- Determining if CFD has been used to simulate sediment transport and scour under bridges with and without piers.
- Identifying who (which state DOTs) uses CFD for the above mentioned purposes, what software is being used, what this software costs, and the cost to conduct studies of this kind.

The work is completed over a two week period for the Minnesota Department of Transportation Office of Policy Analysis, Research and Innovation – Research Services Section with guidance from the Bridge Office.

The increasing availability of powerful and affordable computing platforms has enabled application of numerical modeling to solve flow problems in channels involving complex geometries and sediment transport. CFD typically pertains to using the full Navier-Stokes equations in three dimensions (3-D) to solve fluid flow problems.

1-D, 2-D, and 3-D models are widely used to analyze flow hydrodynamics and sediment transport. The choice of a particular model dimension depends on the type of problem being analyzed. For example, 2-D or 3-D models are applied in the investigations of flow problems such as scour and structural protection, navigation channels, intake structures, river restoration etc. that require detailed knowledge of flow, sediment transport and channel evolution (1, 2). 1-D models are used to study sediment transport, scour and deposition in channels where the lateral variations of hydraulic and sediment conditions can be ignored, and have wide application in the simulation of morphological changes that typically occur in a one year or longer period (3).

The choice of a model for a specific river flow and sediment transport problem should take into account all the parameters that will have a significant effect on the problem. The specific requirements of each problem should be analyzed and the model should be chosen such that these requirements are satisfied. The successful choice of a model, the model run and analysis and interpretation of the results will, to a great extent, depend on the modeler's understanding of the fluvial processes, associated theories, and the capabilities and limitations of the numeric models (1).

3-D models should be used to investigate flow problems that are influenced by three dimensional flow situations that are not amenable to solution using 2-D models. Turbulent flows in meandering river reaches with irregular bends and spurs, flows at bridge crossings, and temporal development of local scour in the vicinity of bridge piers and abutments are some of the flow situations that need 3-D models for successful flow analysis and problem solving (2).

If a long river or channel reach is to be studied over a long time period, it may be more cost effective to use an integrated 1-D, 2-D and/or 3-D model. The basic idea is to divide the whole study domain into several subdomains (reaches), and apply the 1-D model in less important subdomains with simple geometry and the 2-D model and/or 3-D model in more important subdomains with complex geometry (2).

With the advent of parallel computing capabilities, the use of 3-D models in solving fluid flow and sediment transport problems is becoming less time consuming and more affordable. Many CFD programs have parallel computing features that save computational time.

PROJECT TASKS

Task 1: Project Kickoff Meeting

The kickoff meeting was held by telephone conference with Solomon Woldeamlak, Hydraulic Design Engineer of the Bridge Office. Project plans and contacts were discussed to initiate the project. Primary contacts at the Federal Highway Administration (FHWA) were provided. Drs. Kornel Kerenyi and Larry Arneson provided guidance and information related to other state transportation departments working with CFD.

This investigation extends Mn/DOT's experience with CFD including investigating flow coefficients for inlet grates, and fish passage flows through culverts.

Task 2: Discussion with Other State Transportation Departments (DOTs)

The hydraulic engineers from Florida, Maine, and Wyoming were reached by telephone to discuss their experience with CFD. Information was also compiled from work related to Caltrans and Texas. These states and Minnesota provide the majority of state DOT experience for this investigation. Available presentations are provided in the appendix.

FHWA representatives indicated more than three quarters of the state DOTs use 2-D modeling for investigating flow problems. Many fewer have delved into using 3-D (CFD). Florida recently used CFD to simulate the water flow under temporary concrete barriers. Maine recently used CFD to evaluate hydraulic outlet diffusers for increasing culvert capacity. Given the assistance by FHWA and Argonne National Laboratory's Transportation Research and Analysis Computing Center (TRACC), other states are looking into using CFD.

Caltrans' goal is to improve scour analysis with 3-D modeling. The current practice is the CFD modeling of scour needs to be calibrated to physical modeling or results.

Task 3: Internet Research

The internet research for this investigation focused on providing references for the introduction to CFD and experience with the software provided in the following sections. Portions of papers used are cited in parentheses in this report and are listed in the references at the end. Available copies are provided in the appendix if more in depth research is desired by the reader.

Example presentations were also found as part of the internet research. In addition to the presentations from Florida and Maine, two more presentations were found from Caltrans and Texas. Copies are provided in the appendix.

Task 4: Software Investigation

STAR CD/STAR CCM+, FLUENT and FLOW-3D are three popular commercial CFD codes that are currently being used for the analysis of three dimensional fluid flow and sediment transport in river channels.

STAR CD/STAR CCM+ is an advanced commercial code of CFD developed by CD-adapco. Currently, it does not include any specific model of sediment transport (4). However, there are potential features in STAR CD/STAR CCM+ that can be used to develop the scour model, such as the moving mesh, porous medium material and user defined subroutines. The moving mesh theme as the potential approach is capable of achieving the simulation of river bed erosion or distortion by applying the user-defined field function (4). The field function as a user-defined parameter is computed by a sediment transport model developed by the user. Furthermore, the time step size related to the entrainment rate is variable with the computation to satisfy the convergence of solution. Therefore, STAR CCM+ potentially can be employed to visualize the erosion profile and analyze the effectiveness of a pickup function (4).

Argonne National Laboratory's Transportation Research and Analysis Computing Center (TRACC) and Northern Illinois University in collaboration with the Federal Highway Administration's Turner Fairbank Highway Research Center (TFHRC) Hydraulics Laboratory and the University of Nebraska have undertaken 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 (5). As part of this program, a 3-D scour methodology has been developed using the STAR CCM+ and STAR CD 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. Validation of the method is done by comparing results for the equilibrium scour shape and size with the experimental data obtained from TFHRC (6).

Ansys FLUENT is a powerful and flexible general purpose computational fluid mechanics software package that enables CFD analysis of a wide range of fluid flow problems. FLUENT uses a finite volume approach to solve 3-D incompressible continuity and Reynolds-averaged Navier-Stokes equations. The code gives a number of options for simulation of two phase flow including Lagrangian particle tracking technique, Discrete phase modeling, and Eulerian two phase modeling technique. It has the option of user defined functions which can be used to incorporate the users own code (7).

The Lagrangian particle tracking technique available in the code has been used for modeling sediment movement and deposition. The model, initially run and validated for the flow conditions, can be used for simulation of sediments and obtaining a detailed picture of sediment deposition and transportation. The model used with appropriate turbulence model and the Lagrangian particle tracking technique has been found to give fairly good predictions of sediment distribution patterns in meandering channels (7).

The Eulerian two phase model embedded in FLUENT has been implemented in an open channel with loose bed based on two phase mass and momentum equations (8). These equations have been used in conjunction with the constitutive relations that are obtained by applying kinetic theory. Different from traditional sediment transport models, this model uses the two phase theory, and thus, has no need to invoke any empirical sediment transport formulae. In addition to simulating sediment transport, the model can also provide some ideas for simulating scour and bed deformation.

The Discrete Phase Model in a Lagrangian frame, with the sediment phase occupying a low volume fraction and particle-particle interactions neglected, has been applied to modeling sediment deposition and transport in stormwater ponds (8).

FLOW-3D, developed by Flow Science Inc., uses a non-hydrostatic finite difference model to solve the 3-D Navier-Stokes equations, and has a powerful capability to deal with free surface flow and sediment transport issues (4). The model works by emulating both the entrainment of sediment at the packed bed interface and the drifting and deposition of sediment due to gravity (9). When coupled with the three-dimensional fluid dynamics computed by the equations of mass and momentum conservation, the model is able to simulate the deposition and entrainment of sand, silt and other non-cohesive sediment. The model enables specifying multiple sediment species, and includes a bed-load transport model, a nonlinear drift-flux model, and empirical equations to predict the entrainment and erosion of sediment. The model also has a particle tracking feature built in.

Few studies that validate the model results to some extent have been reported. These include comparison with experimental results from a flume study of sediment distribution pattern downstream of a lock gate (9), qualitative evaluation of the results of scour downstream of a dam with outflows over a spillway and several sluice outlets (9), assessing sediment movement by the Lagrangian particle tracking feature of the model (10), evaluation of scour at a bridge pier (11), qualitative simulations of local scour in the vicinity of a group of bridge piers under tidal flow conditions (12).

Based on the limited model validation data that could be obtained, the three CFD modeling programs described above were comparatively evaluated with respect to modeling 3-D flow conditions and sediment transport in river channels and in the vicinity of structures within the flow domain.

i) All the three models appear equally adequate to model the 3 dimensional flow patterns.

ii) Since FLOW-3D has a built in sediment model routine that gives reasonable validated results, and STAR CD/STAR CCM+ has a validated methodology for bed change evaluation, these models are considered slightly better suited for modeling the sediment transport in relation to FLUENT. Based on the limited available data, it is difficult to make a more refined comparative evaluation between FLOW 3D and STAR CD/STAR CCM+ with respect to sediment transport modeling. Engineering judgment and analysis of model run time taking into account the model domain size and configuration, model input parameters, probable model grid size, and available computer capabilities should be made before making the model selection for a particular application. For example, assuming that due to a particular combination of model configuration and flow parameters, the estimated FLOW-3D model run time for an application is very large (6), recourse may be taken to explore using STAR CD/STAR CCM+ as an alternate option.

It should be noted that the sediment transport models are based on several assumptions and empirical relationships. Considering this, the qualitative nature of the results should be recognized, and the model results should be verified using engineering judgment and other means of verification such as validation by analytic solutions, laboratory results, and/or field observations at the site under consideration or similar sites.

In addition to the commercial models discussed above, numerical models CCHE1D, CCHE2D and CCHE3D are programs widely used in several 1-D, 2-D and 3-D flow and sediment transport applications (13). These programs were developed by the University of Mississippi National Center for Computational Hydroscience and Engineering (NCCHE). CCHE1D and CCHE2D are available for use free of charge. CCHE3D is also available for use by collaborating with NCCHE researchers (14). The programs are also available for purchase from private vendor (Computational Hydro-engineering Technology-http://comp-hydro-tech.com/). The CCHE3D flow model simulates open-channel flows using the hydrostatic pressure assumption or solving the full Navier-Stokes equations. The CCHE3D sediment transport model is capable of computing general channel aggradation and degradation, local scour around hydraulic structures, sediment transport near water intake facilities, etc. (15).

The University of Minnesota St. Anthony Falls Laboratory is also actively involved in CFD research (16). An example that directly relates to this technical report includes large eddy simulation of turbulent flow and sediment transport in a straight open channel with a cross vane. http://cfdlab.safl.umn.edu/content/bed-morphodynamics is a link to a video clip showing contours of instantaneous bed elevation.

River ice jams are generally modeled using 1-D and 2-D models. Widely used 1-D models include CRISSP1D, HEC-RAS, RICE, RIV1D, and RIVJAM (17, 18). The CRISSP1D model has been developed by adding several enhancements to the model RICE (19). The two dimensional models that are widely used are DynaRICE and CRISSP2D. The CRISSP2D model is essentially an enhanced version of DynaRICE (18). The 1-D models are normally used for the global evaluation of many ice problems. The 2-D models are useful for analyzing conditions in river reaches where a 1-D analysis is not sufficient. As regards 3-D modeling of river ice, one model that could be located is a Discrete Element Model used by Hopkins and Daly to model the formation of ice jams and to estimate ice forces on structures (20). This model has a three-dimensional DEM, which explicitly models the dynamics of a system of discrete ice floes, coupled with a one-dimensional unsteady flow hydraulic model, which includes the influence of the ice on the channel flow.

The cost to use CFD varies widely depending on the application. State DOTs may use the Argonne National Labs STAR CCM+ software and computing facilities on a FHWA grant basis for research related uses. Otherwise CFD software may be purchased or rented. Given a purchase price in the range of $50,000, it is often rented. Typical rental rates are $4,000 per month, $8,000 for three months, and up to $24,000 for a year. Less complex configurations such as flow around a bridge pier may require in excess of 80 person-hours of time for modeling depending on the modelers experience level.

Task 5: Project Report

Contents of this technical report compile the results of the investigation.

REFERENCES

1. U. S. Department of Interior, Bureau of Reclamation, 'Erosion and Sedimentation Manual', November 2006.

2. Wang, Sam S. Y. and Weiming Wu, 'River Sedimentation and Morphology Modeling - The State of the Art and Future Development', National Center for Computational Hydroscience and Engineering, University of Mississippi, October 2004.

3. Mashriqui, H.S, 'Hydrodynamic and Sediment Transport Modeling of Deltaic Sediment Processes', Dissertation, Louisiana State University, August 2003.

4. Xie, Zhaoding, 'Theoretical and numerical research on Sediment transport in Pressurized Flow Conditions', Civil Engineering Thesis, University of Nebraska, Lincoln, July 2011.

5. Tullimilli, B.R, Majumdar, P, Kostic, M, and Lottes, S.A, 'Development of CFD Simulation for 3-D Flooding Flow and Scouring Around a Bridge Structure' Latest Trends in Urban Planning and Transportation, 3rd WSEAS International Conference on Urban Planning and Transportation, Corfu Island, July 2010.

6. Bhaskar, Rao T, et al, 'Three-Dimensional Souring Analysis for Open Channel Pressure Flow Scour Under Flooded Bridge Decks', ASME 2011 International Mechanical Engineering Congress and Exposition, Colorado, November 2011.

7. Ghani, Usman, Peter Richard Wormileaton et al., 'Numerical Simulation of Sediment Movement and Deposition in a Meandering Channel', Mehran University Research Journal of Engineering and Technology, Vol. 30, No. 4, October 2011.

8. Leying, Zhang, '3-D Numerical Modeling of Hydrodynamic Flow, Sediment Deposition and Transport in Stormwater Ponds and Alluvial Channels' , Old Dominion University, 2009.

9. Brethour, James and Jeff Burnham, 'Modeling Sediment Erosion and Deposition with Flow-3D Sedimentation and Scour Model', FSI-10-TN85, Flow Science, May 2010.

10. Vasquez, Jose (Pepe), 'Assessing Sediment Movement by CFD Particle Tracking', 2nd Joint Federal Interagency Conference, Las Vegas, NV, June 2010.

11. Kayser, M. and M. A. Gabr, 'Scour Assessment of Bridge Foundations Using an In Situ Erosion Evaluation Probe (ISEEP)', 92nd Transportation Research Board Annual Meeting, Washington, D.C, January 2013.

12. Vasquez, J. A, and Walsh, B. W, 'CFD Simulation of Local Scour in Complex Piers under Tidal Flow', 33rd IAHR Congress, 2009.

13. Wu, Weiming and Sam S.Y. Wang, 'Development and Application of NCCHE's Sediment Transport Models', US-CHINA Workshop on Advanced Computational Modeling in Hydroscience and Engineering, September 2005, Oxford, Mississippi, USA.

14. NCCHE (National Center for Computational Hydroscience and Engineering), 'Software', <www.ncche.olemiss.edu/sw_download>.

15. NCCHE, 'CCHE3D - Testing and Validation, Engineering Applications', <www.ncche.olemiss.edu/cche3d>.

16. Khosronejad, A., Kang, S., and Sotiropoulos, F., "Experimental and Computational Investigation of Local Scour Around Bridge Piers," Adv. in Water Resources, 2011.

17. Debolskaya, E.I, 'Numerical Modeling of Ice Regime in Rivers', Hydrologic Systems Modeling, Vol. 1, Encyclopedia of Life Support Systems, Eolss Publishers, Oxford (UK), 2009.

18. She, Yuntong, Faye Hicks, Peter Steffler and Dan Healy, ' Effects of Unsteadiness and Ice Motion on River Ice Jam Profiles', 19th IAHR International Symposium on Ice, Vancouver, July 2008.

19. Shen, Hung Tao, 'Development of a Comprehensive River Ice Simulation System', Proceedings of the 16th IAHR International Symposium on Ice, Dunedin, New Zealand, December 2002.

20. Hopkins, Mark and Steven F. Daly, "Recent Advances in Discrete Element Modeling of River Ice", 12th Workshop on the Hydraulics of Ice Covered Rivers, Edmonton, AB, June 2003.

PEOPLE CONTACTED FOR THIS INVESTIGATION:

Dr. Kornel Kerenyi
Hydraulics R&D Program Manager
Federal Highway Administration, TFHRC
Office of Infrastructure R&D, T111
6300 Georgetown Pike
McLean, VA 22101
(202) 493-3142 (Voice)
kornel.kerenyi@dot.gov

Larry Arneson, Ph.D., P.E.
Hydraulics Technical Service Team Manager
Federal Highway Administration
12300 West Dakota Avenue, Suite 340
Lakewood, CO 80228
(720) 963-3200
larry.arneson@dot.gov

Rick Renna, Florida DOT, (850)-414-4351
Alex Mann, Maine DOT, (207) 215-6328
Bill Bailey, Wyoming DOT, (307) 777-4045

Bruce Walsh, PEng, Principal
Northwest Hydraulic Consultants
30 Gostick Place, North Vancouver, BC V7M 3G3 Canada
(604) 980-6011
bwalsh@nhcweb.com | www.nhcweb.com

John Wendelbo, MEng MSc
Engineer with Flow Science inc
FLOW-3D
(505) 982-0088 x119
john.wendelbo@flow3d.com
www.flow3d.com

Other contacts:

Kevin Flora, Caltrans, (916) 227-8015

Jean-Louis Briaud, Professor Texas A&M University, College Station, TX 77843-3136, USA. (979) 845-3795, briaud@tamu.edu (Texas DOT scour investigations)

Fotis Sotiropoulos, Professor University of Minnesota, Director, St. Anthony Falls Lab, (612) 624-2022, fotis@umn.edu

Example CFD Consultants (This not being the focus of this research is a very limited list):

V. Ramanathan, PhD (Co-writer of this technical report)
(952) 995-9990
9920 Squire Ln
Eden Prairie, MN55347-3161

Bruce Walsh, PEng, Principal
Northwest Hydraulic Consultants
30 Gostick Place, North Vancouver, BC V7M 3G3 Canada
(604) 980-6011
bwalsh@nhcweb.com
www.nhcweb.com

Alden (See web site below for details on physical modeling and CFD services)
30 Shrewsbury Street
Holden, MA 01520-1843
(508) 829-6000
www.aldenlab.com

See also Dr. Fotis Sotiropoulos, St. Anthony Falls (U of M) reference listed above.

APPENDIX

Presentations:

Alexander Mann, Maine DOT, CFD Evaluation of Hydraulic Outlet Diffusers for Increasing Culvert Capacity, 28 slides, 2013.

Dr. Steven Lottes and Dr. Cezary Bojanowski, Transportation Research and Analysis Computing Center (TRACC), Simulating the water flow under the temporary concrete barriers using Computational Fluid Dynamics, 7 slides, 2012.

Kevin Flora, Caltrans, Scour Monitoring and Prediction, 84 slides, 2010.

Anne Lightbody, University of New Hampshire, Fotis Sotiropoulos, Seokkoo Kang, and Craig Hill, University of Minnesota, Panayiotis Diplas, Virginia Tech, Improving Design Guidelines for Rock Vanes and Other Flow Training Structures, 2011

Jean-Louis Briaud, Texas A&M, New Orleans Bridges and Levees, A Look at the Future in Light of Katrina Observations, 66 slides (not dated, post 2006).

Local Research:

St. Anthony Falls Laboratory – Computational Hydrodynamics and Biofluids Laboratory
http://cfdlab.safl.umn.edu/

Khosronejad, A., Kang, S., and Sotiropoulos, F., "Experimental and Computational Investigation of Local Scour Around Bridge Piers," Adv. in Water Resources, 2011.
http://cfdlab.safl.umn.edu/content/bed-morphodynamics

Information from FHWA Transportation Pooled Fund – Study Details.

FLOW-3D Vendor provided information.

Copies of available cited references.