Assessment of computer modeling accuracy in floodplain hydraulics

P. Rao, T. V. Hromadka II, C. Huxley, D. Souders, N. Jordan, C. C. Yen, E. Bristow, C. Biering, S. Horton & B. Espinosa

Published online: 20 Dec 2016.

Submit your article to this journal

Article views: 16

View related articles

View Crossmark data

Taylor & Francis
Taylor & Francis Group

Assessment of computer modeling accuracy in floodplain hydraulics

P. Rao[a], T. V. Hromadka II[b], C. Huxley[c] (iD), D. Souders[d], N. Jordan[e] (iD), C. C. Yen[f] (iD), E. Bristow[g] (iD), C. Biering[g], S. Horton[b] and B. Espinosa[e] (iD)

[a]Department of Civil & Environmental Engineering, California State University, Fullerton, CA, USA; [b]Department of Mathematical Sciences, United States Military Academy, West Point, NY, USA; [c]Wood Rodgers, Inc., Sacramento, CA, USA; [d]Flow Science, Santa Fe, NM, USA; [e]Hromadka & Associates, Rancho Santa Margarita, CA, USA; [f]TetraTech, Irvine, CA, USA; [g]Department of Civil and Mechanical Engineering, United States Military Academy, West Point, NY, USA

ABSTRACT

In this paper, we examine the performance of several popular hydraulic computer models in predicting the magnitude and location of a hydraulic jump. The computer programs considered include the standard HEC-RAS one-dimensional (1-D) models, as well as computational fluid dynamics models. Other computer programs considered include one-, two-, and three-dimensional (2-D and 3-D) models of both the steady and unsteady flow types. The computational model results are compared with experimental data from two different laboratories. It is shown that a few of these commonly used computer models miss the prediction of the hydraulic jump entirely and produce water surface profiles that are not proper estimates. Although the laboratory tests are one-dimensional steady-state scenarios, the failure of some of the multi-dimensional computer models to predict the likely occurrence of the hydraulic jump, or even provide an indication that the flow conditions are favorable for a jump to occur, indicates that care should be taken both while choosing the model for the application of interest and in interpreting the end results.

1. Introduction

Computational fluid dynamics (CFD) is emerging as one of the powerful techniques for modeling flow in open channels. Over the last two decades, many CFD software packages have been developed for solving the mathematical equations that describe the flow of water over topographic surfaces as well as in channels, pipes, and outlet works, among other items of interest. The increase in computational power has given the opportunity for the modelers to simulate an ever-broadening range of applications that was earlier handicapped either by the required computational time or computer memory limitations. The code in each of the software packages solves a system of equations based upon conservation of mass, energy, and momentum, typically using either finite difference, finite volume, or finite element techniques. The complexity of the partial differential equations numerically approximated in these software packages can range from those that describe one-dimensional steady flow conditions to three-dimensional unsteady flows.

All the CFD models in hydraulics like their counterparts from other disciplines are imperfect to various degrees and they may not accurately predict the variable of interest for all possible flow situations. The typical reasons for the marginal to large deviation between experimental and numerical results generally include incorrectly specified boundary conditions (including their values and types), inadequate spatial and time discretizations (or some other computational parameter or factor), and inadequate modeling of a hydraulic situation which the modeler is unaware of or may have discounted. The models also involve modeling flow regime and computational parameters, the values of which need to be 'tuned' to arrive at meaningful results. In cases where experimental or field data are available, tuning these parameters and factors can help arrive at numerical results that are in close agreement with the measured data. These calibrated values often may be inappropriate for use in a different test scenario or a different location.

However, having such data to calibrate a computer model is typically not available in the usual computer model application. Instead, the modeler applies the off-the-shelf computer model to the problem at hand using the procedures outlined in the software documentation. Default values provided by the software are sometimes used. Oftentimes, little attention is paid as to whether the governing flow equations are being adequately solved or whether the computer model resolves the problem into proper components for approximate solution by the computer model's menu of algorithms. For many applications, high attention is paid toward discretizing the problem domain into small modeling cells or components or finite volumes, but proportionately little attention may be paid toward assessing whether the computer program adequately solves the governing flow equations for the situation at hand or assesses the validity and accuracy of the

computer model in solving the problem flow regime's relevant hydraulic principles involved. Because of the user's confidence in the computer modeling software, oftentimes, computational results are not adequately checked for computational accuracy or for adequate resolution of the problem flow regime into proper modeling components. The computer modeling representation of the problem geometry is increasingly handled by higher degrees of discretization involving up to millions of modeling cells (finite volumes, grids, nodal domains, and/or control volumes) which provide the appearance of remarkable conformance to the true problem geometry of the topography and related appurtenances. Similarly, the computer modeling results of flow velocity, depth, and water surface elevation (among other output variables) are depicted by visualization software with remarkable animation and color that achieve nearly photographic levels of displays. Such output quality, although very useful in interpreting the modeling outcome predictions, also may provide false confidence in the modeling results as being accurate. In view of this important observation, the end user should cautiously choose the model for their application.

The levels of confidence heightened by the realism of the visualization of the modeling results can be misplaced. For example, CFD programs present remarkable realism in visualization of computational results, such that results are frequently used in films and other entertainment venues, depicting tsunamis, dam break scenarios, and floods which are completely hypothetical but so realistic to the viewer that the credibility of the visualization is not questioned. Indeed, in order to increase entertainment value, such visualizations may be based upon unrealistic flow parameters and assumptions, resulting in outcomes that appear realistic and exciting to watch, but nonetheless are not physically possible. Confidence in the computer model may be sometimes misplaced and consequently, computational results in approximating the principles of fluid mechanics may actually be flawed, sometimes with serious consequences. According to Toombes and Chanson,[1]

> All numerical models are required to make some form of approximation to solve these principles, and consequently all have their limitations. Sadly, these limitations are usually neither advertised by the software developers, nor investigated and understood by the users. The consequences of misusing a model can be catastrophic.

The reliability of any 1-D, 2-D, or 3-D CFD software should be linked to how well the fundamental theories of fluid mechanics, such as the Navier–Stokes equations, are embodied in the CFD software, and how well the computational methods and analysis match experimental observations and field experience. There is a good chance that the end user is neither aware nor interested to know about the theory behind the computer model code or the computational algorithm/sequence of steps. All CFD computer software packages generate computational outcomes, and the end user should not assume, of course, that the computational results are accurate simply because the computer program terminated its sequence of operations and arrived at a computational outcome. Computer model end users need to investigate the important issues of model stability, consistency, and convergence when assessing the accuracy of their computational results. End users must adequately validate the computational results rather than relying on their trust in the model developers and the realism of the modeling visualization.

Verification and validation (V&V) tools that are now gaining momentum in aeronautics,[2] including methods of manufactured solutions,[3] have not yet been adequately applied to other flow modeling such as hydraulic software. In the absence of these V&V tools, the end user must be careful not to fall victim to trusting modeling results which simply appear to be plausible and realistic. Toombes and Chanson [1] said it well: 'While the advantages of numerical models cannot be ignored, we run the risk of becoming mindless automatons, plugging raw data into our numerical models and blindly accepting the results that they produce.'

Computational hydraulics is a subset of CFD. The computer programs tested in this paper include 1-D, 2-D, and 3-D models of the governing steady and unsteady flow equations that describe flow of water over a topographic surface. All of these models include various simplifications of the Navier–Stokes equations. These equations are complex and cannot be solved analytically; therefore, numerical approximations are involved. Additionally, boundary conditions and initial conditions are required to be specified as part of the modeling solution which results in further simplifications and approximations being made. All of the computer models tested herein are based on derivations of the basic principles of fluid flow. All are required to make some form of numerical approximation to solve these equations. All have their limitations. The use of a model in a manner for which it was not designed, or that contravenes the approximations upon which it was based, can lead to gross errors in the model predictions. The consequences may lie anywhere between negligible and catastrophic, potentially leading to property damage and loss of life.[1] In this paper, we examine an important hydraulic effect that creates a type of boundary condition within the interior of a complex network or model mesh of cells and grids, namely: the hydraulic jump.

Several of the models examined herein do not attempt to model the occurrence of this effect. However, the prediction of such an effect is an important part of design for flood control works, as well as ascertaining whether the flow field around the hydraulic jump results in unexpectedly high or low water surface elevations. The model outcomes might still appear plausible with the jump being incorrectly modeled as being drowned by modeled backwater (or inaccurately modeled by other types of approximations) with associated inaccurate estimates of water levels being made. Consequently, the model user needs to be aware that such modeling details may not be properly handled even though the modeling products may appear convincing and plausible. In this paper, we assemble modeling outcomes from several commonly used computer models in the analysis of a two-dimensional topographic flow situation where a hydraulic jump occurs in the interior of a detailed modeling mesh of the problem domain. The hydraulic jump is not specified as part of the modeling exercise. Consequently, how the model handles the test flow regime and how the computed results indicate to the model user and viewer of model results that there is actually a hydraulic jump involved are a key part of this paper. Some of the models predict the occurrence of the hydraulic jump. Others miss the jump entirely and produce model results that are seriously inaccurate, yet still would appear to be plausible to an uninformed viewer.

The importance and relevance of the occurrence of a hydraulic jump embedded within a two-dimensional flow regime is highlighted in the HEC Newsletter [4] from Spring 2015. Here, a breach flow situation is modeled as a two-dimensional flow regime with the breach flow encountering an obstacle to flow which results in a hydraulic jump. Although the occurrence of a hydraulic jump may oftentimes be predicted within a two-dimensional flow regime, there are occasions where the jump may not be predicted beforehand and boundary conditions are not specified by the computer program user to force the jump modeling process to happen. As a result, the flow energy is not properly calculated. However, the computer programs may produce well-visualized approximation results which are then utilized for subsequent planning and design, all based upon an inappropriate analysis of the hydraulic jump. That is, software visualization of the computational results may far outweigh the validity of the computational modeling accuracy and appropriateness.

CFD computer models examined

The focus of this work is to test the performance of several popular computer models for predicting the location of a hydraulic jump in a one-dimensional channel flow situation. Benchmarking the performance of these models with experimental data can shed new light on the strengths and possible limitations in each of the models. Although the experimental data assembled are based upon steady-state flow conditions in a one-dimensional channel where a hydraulic jump situation occurs, the target hydraulic jump condition can similarly occur within the interior of a dense two-dimensional mesh of a broad expanse of topography where stream tubes of flow can be well represented as a one-dimensional flow regime. The occurrence of the target effect of a hydraulic jump still follows the same hydraulic principles as encountered in the one-dimensional laboratory experiments considered in this paper, including those of Toombes and Chanson [1; test data obtained from the authors in preparation of the current paper]. Of course, at issue is the accuracy of the computer modeling results in the event that the target hydraulic jump is not properly predicted and its hydraulic effects included in the total modeling outcome.

The following models are selected to be examined (also see Table 1):

(1) *USGS diffusion hydrodynamic model*: Diffusion hydrodynamic model (DHM) solves the two-dimensional overland flow coupled with one-dimensional open channel flow equations, and includes

interfaces between these two flow regimes by means of source and sink term approximations. The model is capable of approximation of such hydraulic effects as backwater, drawdown, channel overflow, storage, and ponding.[5] The cited report includes numerous example problems and applications, including demonstrations of outcomes from the DHM modeling as applied in situations involving supercritical flow for Froude numbers up to 4.

(2) *HEC-river analysis system*: HEC-*river analysis system* (RAS) contains one-dimensional river analysis components for: (1) steady flow water surface profile computations and (2) unsteady flow simulation. In steady-state mode, the energy equation between two sections is solved. Energy losses are evaluated by friction and contraction/expansion. The momentum equation may be used in situations where the water surface profile is rapidly varied. In unsteady mode, HEC-RAS solves the fully dynamic, 1-D Saint Venant equations using an implicit, finite difference method.[6] The unsteady flow equation solver was adapted from the UNET package.[7] Using the mixed flow option, the computer program predicts the location of the hydraulic jump by balancing pressure-plus-momentum.

(3) *Water surface pressure gradient (WSPG)*: WSPG solves the Bernoulli energy equation between any two cross sections, using the standard step method. The program computes uniform and non-uniform steady flow water surface profiles. As part of the solution, it can automatically identify a hydraulic jump in the channel reach. The computer program is written in the DOS environment for the Los Angeles County Department of Public Works.[8]

(4) *Water surface pressure gradient* 2010: Water surface pressure gradient (WSPG 2010) is an updated version of WSPG upgraded for the Windows environment. The flow equations and the computational algorithm are as used in WSPG; however, the interface in WSPG 2010 gives greater flexibility to the user for entering the channel and flow input data.[8]

(5) *Two-dimensional unsteady flow*: Two-dimensional unsteady flow (TUFLOW) solves the two-dimensional depth-averaged shallow water equations using a structured grid system with an alternating direction implicit scheme (Stelling Finite

Table 1. Summary details of the tested software.*

Software	Vendor	Version	Flow equations	Ability	Cost
DHM	USGS		2-D	Steady, Unsteady	Free
HEC-RAS	U.S. Army Corps of Engineers	4.1.0	1-D	Steady, Unsteady	Free
WSPG	Civil Design	14.07	1-D	Unsteady	$
WSPG 2010	XP-Solutions	14.05	1-D	Unsteady	Free
TUFLOW	BMT WBM	2013	2-D	Unsteady	$
MIKE 11	DHI	2008	1-D	Unsteady	$
MIKE 21	DHI	2008	2-D	Unsteady	$
FLOW-3D	Flow Science Inc.	9.2.1	3-D	Unsteady	$
FLO-2D	FLO-2D Software Inc.	Pro	2-D	Unsteady	$
HY-8	U.S. Federal Highway Administration	7.30	1-D	Steady	Free

*Table 1 is not intended to be a full and complete list of all versions available of each software.

Difference). The scheme can approximate transitional flow regimes, wetting and drying, and effects of a hydraulic structure. The model incorporates the 1-D hydrodynamic network software ESTRY or quasi-2D modeling system based on the full one-dimensional free surface flow equations.[9] TUFLOW CPU uses a computer's central processing unit (CPU) for its calculations. TUFLOW GPU uses a computer's graphics processing unit (GPU, also called a video or graphics card) for its computations, instead of the computer's CPU.

(6) *Mike 11*: Mike 11 solves the one-dimensional Saint Venant equations by means of an implicit finite difference scheme. Based on the applications, the model provides add-on modules for a wide-ranging applications, including dam break, rainfall–runoff, water quality, and sediment transport.[10]

(7) *Mike 21*: Mike 21 solves the two-dimensional free surface flows where stratification can be neglected. It was originally developed for flow simulation in coastal areas, estuaries, and seas. The various modules of the system simulate hydrodynamics, advection–dispersion, short waves, sediment transport, water quality, eutrophication, and heavy metals. [10]

(8) *FLOW-3D*: FLOW-3D is a general-purpose CFD simulation software that solves the three-dimensional transient Navier–Stokes equations using the Fractional Area/Volume Obstacle Representation and the volume of fraction method. The model can simulate two-fluid problems, incompressible and compressible flow, as well as laminar and turbulent flows. The solver is based on a finite volume or finite difference formulation, in Eulerian framework, of the equations describing the conservation of mass, momentum, and energy in a fluid.[11]

(9) *FLO-2D*: The paper of O'Brien [12] states that, 'The FLO-2D model evolved from the diffusive hydrodynamic model (DHM) [5] … The model uses a central finite difference routing scheme (an explicit numerical technique) for the approximation of the equations of motion. The surface topography is discretized into uniform square-grid elements…' Because of the stated roots of FLO-2D and similarity to DHM, and because the modeling tests of FLO-2D conducted for the current paper were found to be similar to the testing results from DHM, the FLO-2D test results are not plotted separately in the figures. It is noted that additional tests of FLO-2D and DHM were conducted using larger scale grid sizing and hydraulic jump situations, and the resulting outcomes were also found to be similar.

(10) *HY-8*: HY-8 is a culvert modeling software sponsored by the U.S. Federal Highway Administration. It is used to compute the upstream headwater depth and flow profile for different culvert configurations. The software can be used to delineate a watershed upstream from a culvert, compute a hydrograph, and determine the inundated area behind a culvert.

2. Literature review

Hromadka and Yen [4] reviewed the fundamental equations for modeling flow in topographic floodplains and channels. Through various simplifications and assumptions, they derived the diffusion formulation for modeling unsteady two-dimensional flow. Validation and verification of the DHM program is included in the report that includes testing for dam break scenarios vs. the USGS K-634 computer program. The report and software are downloadable at the USGS site and other web locations at no cost. The report was later published as a journal paper [13] and also appears in several papers published in various journals. [14,15]

Toombes and Chanson [1] also reviewed the basic equations which govern fluid motion and tested the performance of four popular hydraulic software packages for flow in a channel with a weir and hydraulic jump. The software programs they tested were HEC-RAS (steady mode), MIKE 11, MIKE 21, and FLOW-3D. Our current work supplements and expands upon their work, and as presented later, extends the results of their analysis by the inclusion of additional laboratory data and other computer modeling results. Many modelers have benchmarked their CFD codes either with other codes (i.e. not with experimental data) or for flows where there is no hydraulic jump. Toombes and Chanson [1] tested the steady and unsteady modes in the CFD software for flows with hydraulic jump. Among the modeling community, a critical test for a CFD code lies in its ability to capture a jump as part of the solution, at the right location and without significant numerical instabilities in the vicinity of the jump.

Ever since Kuipers and Vreugdenhill [16] developed the first two-dimensional finite difference model for solving the depth-averaged flow equations, many researchers have solved these equations or variations for predicting hydraulic jump characteristics in open channel flow regimes.[17,18] The advent of computational power and a better need to analyze hydraulic jump properties at a microscopic scale motivated researchers to solve the Navier–Stokes equations [19–21] for simulating hydraulic jump extent and location. Pineda et al. [22] solved the Navier–Stokes equations using computer program ANSYS CFX for predicting the jump characteristics. They noted that to arrive at good accuracy, special care with grid selection and entrance boundary conditions is crucial. Jowhar and Jihan [23] compared the performance of HEC-RAS and the 2-D Adaptive Hydraulics (ADH) software for predicting steady-state jump characteristics. Their results indicate that the jump location from ADH may be more accurate than those from HEC-RAS.

3. Sensitivity of modeling results to boundary conditions

The governing partial differential equations that describe the flow regime are sensitive to the boundary conditions imposed at the upstream end of the problem domain. The theory behind the number of required boundary conditions at the upstream and downstream ends has been reviewed in Chapter 16 of the standard book on CFD.[24] The nature of flow (subcritical or supercritical) determines the number of boundary conditions that need to be specified at the domain boundaries. From a mathematical perspective, a boundary condition is a constraint

Table 2. Channel and flow conditions for the two laboratory tests.

	Cluster 1	Cluster 2
Width (m)	0.46	0.5
Length (m)	15.24	12
Q (m³/s)	0.036	0.035
Upstream depth (m)	0.04	0.062
Downstream depth (m)	0.24	0.235
Channel slope	0.012	0.028
Roughness factor (Manning's)	0.01	0.007

imposed at the boundary node to arrive at a unique solution to a well-posed equation set. Specifying more or less than the required number may make the problem 'ill posed' [25] and can lead to incorrect solutions if indeed the model runs at all.

4. Experimental setup

To better understand the abilities of the computer models tested in the current paper, a series of physical model experiments were conducted in the open channel flume located in the hydraulics laboratory at California State University, Fullerton. The rectangular channel measured 15.24 m long, 0.46 m wide, and 0.6 m deep. The channel slope was 0.012. The flow rate was varied by adjusting the pump speed which feeds water to the flume. This experiment is identified as 'Cluster 1,' details of which are contained in Table 2 below. As mentioned earlier, this current work parallels that of Toombes and Chanson [1] who performed similar experiments at the University of Queensland. Their rectangular test flume was 7 m long, 0.5 m wide with a constant slope of 0.028. Since we will be referring to their model results (Figure 2) in our discussion, to aid the readers, their test data are identified as Cluster 2 in Table 2 below.

5. Initial conditions and boundary conditions in the models

Particular details regarding boundary and initial conditions imposed on the examined computer models are given below:

- *DHM*: The downstream boundary condition for the DHM model was the flow depth in the grid element corresponding to the downstream end of the flume, or DHM grid element #14. Grid Element #14 was constrained to have a depth of 0.696 ft (DHM accepts only US units) or 0.21 m. No upstream boundary condition at Grid Element #1, other than inflow of 0.036 m³/s, exists in the DHM model. This model starts calculations in numerical order of the grid numbering system, in this case starting at the upstream end with Grid #1 and ending with Grid #18 at the downstream end of the DHM model. The upstream boundary condition is flow into Grid #1, specified as 0.036 m³/s. The downstream boundary condition is critical depth at the most downstream Grid #18. The grid elevation was iteratively raised such that the water depth in Grid #14 was equal to the flume depth of 0.22 m at that point.
- *HY-8*: At the tailwater, a constant elevation of 0.24 m was specified. The discharge in the channel was set to 0.036 m³/s for minimum flow, design flow, and maximum flow in Discharge Data.

- *HEC-RAS (steady mode)*: At the upstream and downstream ends of the domain, the water surface elevations of 0.04 m and 0.0.057 m were specified. The flow in the channel was specified as 0.036 m³/s.
- *HEC-RAS (unsteady mode)*: At the upstream end, the available boundary options are (a) stage hydrograph, (b) flow hydrograph, and (c) stage/flow hydrograph. At the downstream end, there are two additional options (d) rating curve and (e) normal depth. In this work, at the upstream end, a flow hydrograph was used. The flow was linearly raised from 0 to 0.036 m³/s and then held constant for the rest of the simulation period. At the downstream end, a stage hydrograph was specified. A constant stage of 0.057 m was used for all time periods. The steady-state solution was obtained by iterating the solution with time. Starting from the initial conditions at model time = 0, the solution advances in time until it reaches steady state.
- *WSPG & WSPG-2010*: At the downstream end (system outlet), a flow depth of 0.24 m was assigned. At the upstream end (system headworks), a depth of 0.04 m was specified. The flow in the channel was specified as 0.036 m³/s.
- *TUFLOW*: The model was initialized using an initial water level of 0.24 m. A time series downstream stage hydrograph and upstream flow hydrograph were applied as the model boundary conditions. The stage hydrograph was set to a constant level of 0.24 m. The flow was linearly raised from 0 to 0.036 m³/s and then held constant. Results were extracted at the conclusion of the simulation when the boundary condition and modeled flow conditions reached steady state.
- *FLOW-3D*: The inlet boundary condition is set to a volumetric flow of 0.036 m³/s with a fixed elevation of 0.04 m above the ramp. Downstream, a hydrostatic pressure condition is set because the flow is subcritical. The depth at that boundary is maintained at 0.24 m above the ramps elevation. Initial conditions are a simple, uniform elevation matching that of the inlet boundary.

The computational results from the above models are compared with those of Mike 11, Mike 21, and FLOW-3D.[1] Specific details about these additional models can be found in the given references.

6. Results

The resulting water surface elevations along the central stream tube for both the physical laboratory model and the various CFD computer models are shown in Figure 1.

- *DHM*: Given the boundary condition limitations stated previously, DHM computed flow depths that gradually and smoothly decreased in the downstream to upstream direction through the flume hydraulic jump location. There was no abrupt change in computed depth. At the upstream end of the model, flow depth was roughly the same as the flume. Looking at the results in the upstream to downstream direction, DHM computed a flow transitioning
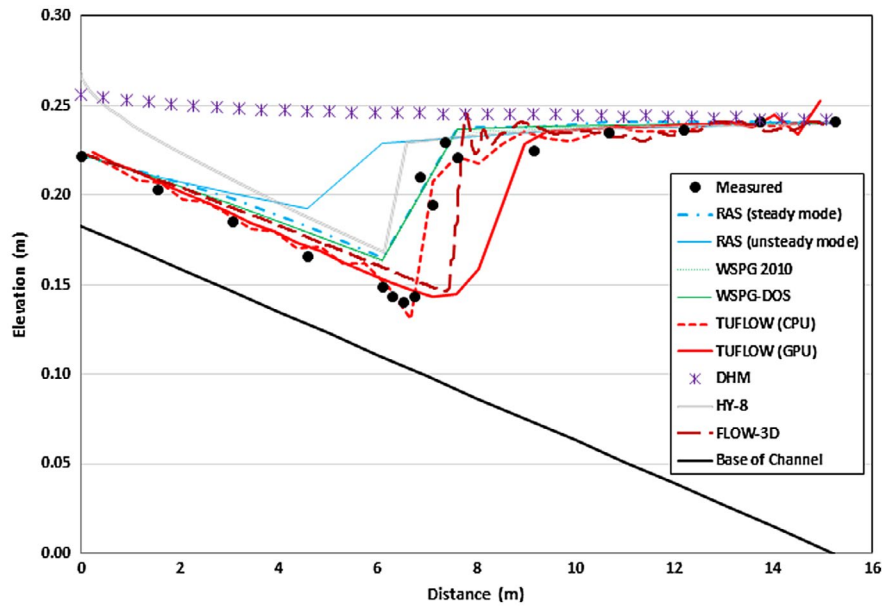
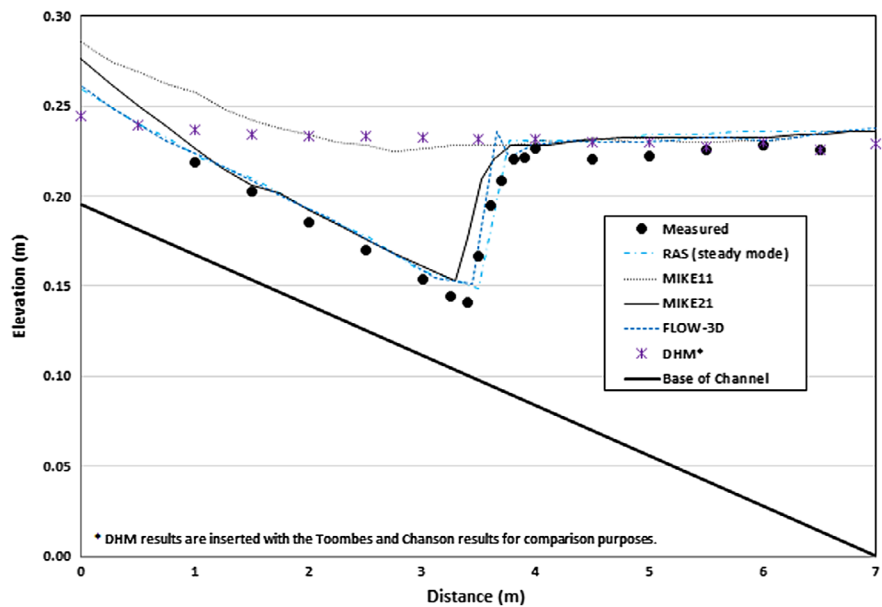**Figure 1.** Comparison of model results for Cluster 1 data.

Figure 2. Comparison of Cluster 2 model results [1].

from supercritical to subcritical without going through a hydraulic jump as required by theory and observed in the flume model. The governing flow equations for DHM include only the first three of the five terms in the Saint Venant equations: friction, gravity, pressure, convective acceleration, and local acceleration. It was found during development of the original model that the two acceleration terms summed to nearly zero for Froude numbers less than 4. If both acceleration terms were eliminated, computational efficiency and speed would be improved for the small personal computers in use in the mid-1980s. Absence of the acceleration terms results in DHM not accounting for the large localized energy losses in a hydraulic jump.

- *HY-8*: The location of the jump is similar to RAS (steady mode) and WSPG output (described below). However, the computed headwater depth differs from the observed data.
- *HEC-RAS (steady mode), WSPG, and WSPG 2010*: The computational results from these software packages are similar to each other and are similar with the experimental data. While the WSPG models identify the jump characteristics automatically, in HEC-RAS, the user has to select the program's 'mixed flow' regime option. No specific variable needs to be tuned to arrive at the optimal results. Given the simple geometry domain of the test problem, all three models were easy to use.
- *HEC-RAS (unsteady mode)*: The model predicted the likely occurrence of a hydraulic jump at a different location than

as predicted with the steady-state condition (see Figure 1). This modeling outcome was also mentioned by Toombes and Chanson.[1] A possible reason for this modeling difference may be the relatively short distance between the upstream boundary and the location of the jump. The results highlight how boundary condition assumptions influence the computational results. Some practitioners believe that the steady-state solution and the unsteady mode HEC-RAS computations should be similar, as the time differential term in the unsteady equation vanishes. However, as the computational results for this test case show, such may not be the case for HEC-RAS. The jump was located between two adjacent nodes and changing the cross-section spacing had no effect on the location of the jump, for the steady-state mode of operation.

- *TUFLOW*: The results of the TUFLOW CPU and GPU software are similar with each other and also with the experimental data. Both models automatically execute in mixed mode, and as such transitioned from supercritical flow to subcritical flow conditions without the need for user fine-tuning to arrive at the optimal results. The model framework for the development of the TUFLOW CPU and GPU is GIS based.

- *FLOW-3D*: This particular setup is two-dimensional in that the boundary conditions per the width of the channel are in symmetry with only one computational cell set at a width of 0.46 m. The results shown are for a given quasi-steady-state solution. The jump location in FLOW-3D remains fairly constant but the surface elevations post the jump location tend to vary at any given time step. This is expected and matches reality in that the same thing should be observed physically. Overall, the data match well with respect to surface elevations. The jump occurs later than the experiment; however, changes in surface roughness and turbulence models can affect the results. More variations may hone in on a precise match. It has also been found, and expected, that including air entrainment option changes the jump location due to the air induced into the fluid, which changes both density and volume. This particular validation did not include air entrainment.

The trends of the obtained computational results are similar to those of Toombes and Chanson,[1] who focused on testing the computer programs MIKE11, MIKE21, and FLOW-3D to model their hydraulic jump experiment. Figure 2 captures the Toombes and Chanson's results from Ref. [1]. Since DHM (and programs similar to DHM) did not predict the hydraulic jump in Cluster 1, we reran the DHM model for Cluster 2 data and added this additional output to Figure 2. From this, it can be seen that while MIKE11 and DHM missed the jump, the results of MIKE21 and FLOW-3D are in close agreement. In the current testing effort, other software were tested as well; for example, FLO-2D was examined and it produced outcomes similar to DHM and MIKE11 as shown, namely: they did not match the performance as accomplished by the other computer programs tested and shown in the diagrams.

It is noted that an approximate solution to the problem can be obtained using 'off the shelf' calculators found on the web. To test their reliability for modeling Cluster 1, we used two calculators by McNoldy and Ponce[26,27] that solve the 1-D open channel flow equations. Although these calculators present reasonable analogies to the tested flow conditions, they are limited in that they cannot take into account all the channel geometry parameters (e.g. bottom slope and roughness).The predicted downstream depth from both the calculators is 0.16 m while the actual depth is 0.24 m. Clearly, end users need to be aware of the assumptions in the models.

7. Conclusions

In this paper, multiple computer programs are tested as to their outcomes in the prediction of properties associated to a rapidly varied flow hydraulic jump. Several popular hydraulic computer programs based upon one-, two-, and three-dimensional spatial formulations with steady and unsteady flow conditions are examined. It is demonstrated that some commonly used computer models miss the prediction of the hydraulic jump entirely and instead produce water surface profiles that are not proper estimates of the true water surface profile. These demonstrations are compared to actual laboratory measurements of two similar hydraulic jump situations conducted in different hydraulic laboratories. Although the laboratory tests are one-dimensional steady-state scenarios, there is failure of some of the multi-dimensional computer models to predict the likely occurrence of the hydraulic jump, as well as failure to provide an indication that a jump is likely at a particular location. The models that failed to predict the hydraulic jump still produced plausible though incorrect water surface predictions. That some failing models still produced animation and color with high levels of discretization should be a cause for concern in the modeling community.

Notes on contributors

P. Rao is an associate professor at the Department of Civil & Environmental Engineering, California State University, Fullerton, CA 92831, USA.

T. V. Hromadka II is a professor at the Department of Mathematical Sciences, United States Military Academy, West Point, NY 10996, USA.

C. Huxley TUFLOW, and Wood Rodgers, Inc., Sacramento, CA 95816, USA.

D. Souders FLOW-3D, Flow Science, Santa Fe, NM 87505, USA.

N. Jordan is a consultant at Hromadka & Associates, Rancho Santa Margarita, CA 92688, USA.

C. C. Yen is a consultant at TetraTech, Irvine, CA 92614, USA.

E. Bristow is an associate professor at the Department of Civil and Mechanical Engineering, United States Military Academy, West Point, NY 10996, USA.

C. Biering is an instructor at the Department of Civil and Mechanical Engineering, United States Military Academy, West Point, NY 10996, USA.

S. Horton is a professor and deputy department head at the Department of Mathematical Sciences, United States Military Academy, West Point, NY 10996, USA.

B. Espinosa is a consultant at Hromadka & Associates, Rancho Santa Margarita, CA 92688, USA.

ORCID

C. Huxley http://orcid.org/0000-0001-5393-6193
N. Jordan http://orcid.org/0000-0002-8762-9957
C. C. Yen http://orcid.org/0000-0001-7792-6264
E. Bristow http://orcid.org/0000-0002-9244-3161
B. Espinosa http://orcid.org/0000-0002-8041-3504

References

[1] Toombes L, Chanson H. Numerical limitations of hydraulic models. The 34th International Association for Hydraulic Research (IAHR) World Congress. Brisbane, Australia; 2011. p. 2322–2329.

[2] Oberkampf WL, Trucano TG. Verification and validation in computational fluid dynamics. Prog Aerosp Sci; 2002;38:209–272.

[3] Roache PJ. Code verification by the method of manufactured solutions. J Fluids Eng. 2002;124:4–10.

[4] Brunner G. Combined one- and two-dimensional hydraulic modeling with HEC-RAS. HEC Newsletter. Spring 2015. Available from: http://www.hec.usace.army.mil/newsletters/HEC_Newsletter_Spring2015.pdf

[5] Hromadka II TV, Yen CC. A diffusion hydrodynamic model. Water resources investigations report. U.S. geological survey; 1987: 87–4137.

[6] U.S. Army Corps of Engineers (USACE). HEC-RAS river analysis system. User's Manual. Hydrologic Engineering Center. Davis, CA. 2015. Version 4.1.0. Available from: http://www.hec.usace.army.mil/software/hec-ras/

[7] U.S. Army Corps of Engineers (USACE). Features in HEC-RAS. Hydrologic Engineering Center. Davis, CA. 2015. Available from: http://www.hec.usace.army.mil/software/hec-ras/features.aspx

[8] Water Surface Pressure Gradient User's Manual. 2010. Version 14.05. Available from: http://xpsolutions.com/assets/dms/xpwspg_Reference_Manual.pdf

[9] BMT-WBM Australia. User's Manual for TUFLOW. Spring Hill: WBM Oceanics Australia. 2008.

[10] DHI, MIKE 11 & MIKE 21 Flow Model. Scientific Documentations. 2012. Available from: https://www.mikepoweredbydhi.com/

[11] Flow Science. FLOW-3D User Manual. 2014. Available from: www.flow3d.com

[12] O'Brien JS, Julien PY, Fullerton WT. Two-dimensional water flood and mudflow simulation. J Hydraulic Eng ASCE. 1993;119:244–261.

[13] Hromadka TV, Yen CC. A diffusion hydrodynamic model (DHM). Adv Water Res. 1993;9:118–170.

[14] Hromadka TV, Yen CC, Bajak PA. Application of the USGS diffusion hydrodynamic model (DHM) in evaluation of estuary flow circulation. Adv Eng Softw. 1992;14:291–301.

[15] Hromadka II TV, Walker TR, Yen CC. Using the diffusion hydrodynamic model (DHM) to evaluate flood plain environmental impacts. Environ Softw. 1988;3: 4–11.

[16] Kuipers J, Vreugdenhill CB. Calculations of two-dimensional horizontal flow. Rep. S163. Part I. Delft Hydraulics Lab. Delft, The Netherlands. 1973.

[17] Fennema RJ, Chaudhry MH. Explicit methods for 2-D transient free surface flows. J Hydraulic Eng ASCE. 1990;116:1013–1034.

[18] Katopodes ND, Strelkoff T. Computing two-dimensional dam-break flood waves. J Hydraulics Div. 1978;104:1269–1288.

[19] Chippada S, Ramaswamy B, Wheeler MF. Numerical simulation of hydraulic jump. Int J Numer Methods Eng. 1994;37:1381–1397.

[20] Chanson Hubert. Hydraulics of open channel flow. Oxford: Butterworth-Heinemann; 2004.

[21] Babaali H, Shamsai A, Vosoughifar H. Computational modeling of hydraulic jump in the stilling basin with convergence walls using CFD codes, Arab J Sci Eng. 2015;40:381.

[22] Pineda S, Blanco A, Luis Rojas-Solórzano L. CFD software applications for transcritical free surface flow. ASME fluids engineering division summer meeting, Vol 1: Symposia: Parts A, B and C. 2009 Aug 2–6; Vail, CO. ISBN: 978-0-7918-4372-7.

[23] Jowhar RM, Jihan M. Qasim. comparison of 1D HEC-RAS with 2D ADH for flow over trapezoidal profile weirs. Caspian J Appl Sci Res. 2012;1:1–12.

[24] Hirsh C. Numerical computation of internal and external flows. New York (NY): John Wiley & Sons; 1990.

[25] Garcia R, Kahawita R. Numerical solution of the St. Venant equations with the MacCormack finite difference scheme. Int J Numerical Methods Fluids. 1986;6:259–274.

[26] McNoldy B. Flow properties of horizontal rectangular channel hydraulic jumps. 2015. Available from: http://einstein.atmos.colostate.edu/~mcnoldy/HydraulicJump.html

[27] Ponce VM. Sequent depth in a hydraulic jump. 2015. Available from: http://onlinecalc.sdsu.edu/onlinechannel11.php