VALIDATION OF A HYBRID 3-DIMENSIONAL AND 2-DIMENSIONAL NUMERICAL FLOW MODELING TECHNIQUE FOR AN INSTANTANEOUS DAM-BREAK

Authors

David Souders, Vice President of Sales, Marketing, and Customer Support, Flow Science, Inc., Santa Fe, New Mexico

Jayesh Kariya, Testing & Validation CFD Engineer, Flow Science Software Pvt Ltd., Pune

Jeff Burnham, PE, LEED AP, Hydraulics Applications Engineer, Flow Science, Inc., Santa Fe, New Mexico

Abstract

While physical models remain the mainstay of the water and environmental industry, numerical simulations using Computational Fluid Dynamics (CFD) are increasingly common and provide a comparatively inexpensive and rapid way to study catastrophic conditions and validate designs. However, available computational resources have often restricted the use of CFD to certain classes of problems, such as the modeling of spillways and intakes. A novel addition to the commercial CFD package ***FLOW-3D*** bypasses this restriction by solving the full 3-D Navier-Stokes equations in parts of the computational domain and solving the simpler 2-D depth-averaged flow equations in other parts of the computational domain. This unique approach reduces the computational overhead required for many simulations and consequently allows the simulation of previously intractable configurations, such as 3-D dam breaks and the subsequent downstream flooding. To validate the hybrid approach, ***FLOW-3D*** was used to replicate an experimental study of an instantaneous dam break. The resulting mean relative error in the numerical solution is less than 6% when compared to the physical experiment.

Introduction

Flooding due to dam failure can have catastrophic consequences and is of significant concern to dam safety professionals. A number of methods have been developed for predicting the outcome of a dam breach including flow behavior, inundation locations and depths, and possible damage and loss of life. Computational Fluid Dynamics (CFD) is one of the most cost-effective options in the water and environmental industry for modeling catastrophic conditions and validating designs for safety and environmental criteria. CFD proves especially useful wherever flows are complex and difficult to approximate with physical models; some common examples of CFD models for water resources include spillway flows under probable maximum flood (PMF) conditions, hydraulic performance of fish passages and river intakes, and models of large scale flooding due to dam breakage.

CFD models can be limited by the relevant scales of the computational domain: finely-detailed 3-D solutions over large domains require tremendous computational resources. 2-D (depth-averaged) solutions are more efficient but neglect vertical flow components, thus making them less general. The present work tests a unique hybrid approach that combines 3-D and 2-D modeling to capitalize on the strengths of each. The results of the simulation, performed with the commercial software ***FLOW-3D***, are compared to physical experiment. References [5-9] document additional dam-break modeling validation tests of ***FLOW-3D***.

Experimental Setup

The numerical accuracy of ***FLOW-3D*** was tested against a benchmark experiment of a near-instantaneous dam break measured at the University of Leuven (UCL Belgium) hydraulics lab [2]. The experimental facility is illustrated in Figure 1.

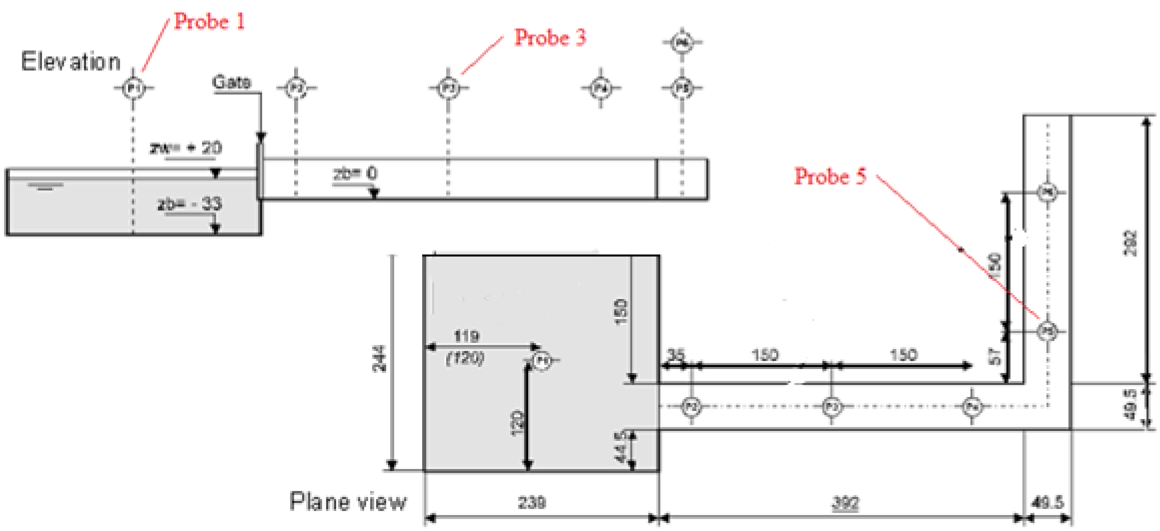

Figure 1: Experimental facility for the L-channel test. Units are in centimeters. [2]

The setup consists of an upstream square reservoir connected to a rectangular channel with a 90 degree bend, steel bottom, and glass side walls. The effective Manning's coefficient was tested and found to be about 0.006 $s/m^{1/3}$. A guillotine-type gate separates the reservoir from the channel and is lifted rapidly to cause a wave of water to propagate into the channel.

Gauges are used to measure water depth with respect to time. The gauges collect data at a sampling rate of 0.1 seconds. Data from three probe points (probes 1, 3 and 5) are available in the referenced paper and are used for comparison with the numerical model. Gauge 1 is located in the reservoir. Gauge 3 is located at 1.85 meters from the reservoir in the channel upstream of the bend. Gauge 5 is located 0.57 meters from the inner corner channel downstream of the bend.

Numerical Model

FLOW-3D version 10.1 was used for the numerical simulation. The CFD package applies the original volume-of-fluid (VOF) method of Hirt and Nichols [3] for tracking fluid motion with transient free surfaces. 3-D turbulent effects and shear stresses are modeled with the renormalized group (RNG) k-ε turbulence model with mean surface roughness height estimated from the experimental Manning's n. 2-D turbulence effects and shear stresses are modeled by shallow water equations with mean drag coefficient also estimated from the experimental Manning's n. The computational domain consists of a 3-D grid defining the reservoir with 4-cm cubic cells adjacent to a 2-D grid representing the channel with 2-cm square cells. Measurement probe locations match those in the physical experiment. The grids, channel geometry, and probe locations are shown in Figure 2.

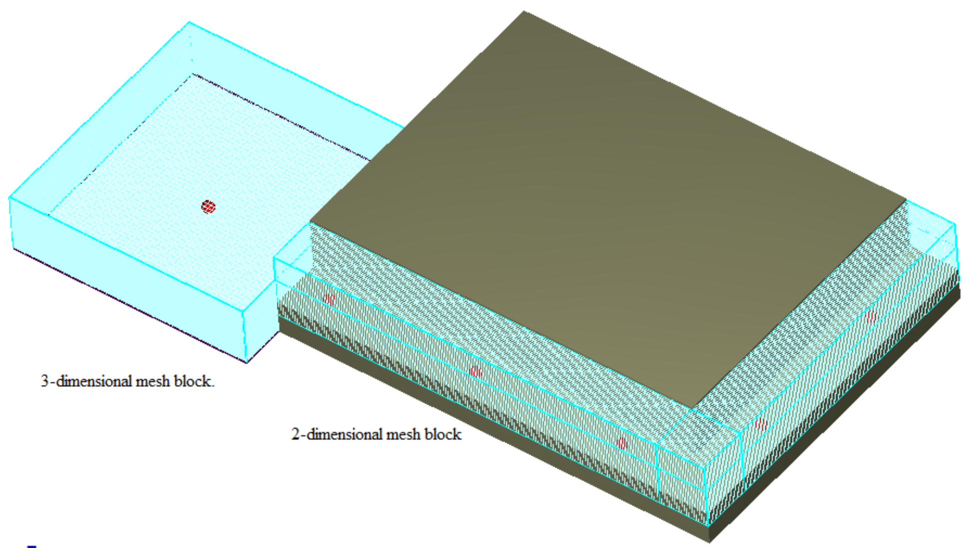

Figure 2: Numerical model domain, solid geometry, and measurement probes.

Results

Physical and numerical water depths are compared at the three monitoring locations described in [2]. While the uncertainties related to the measurements of the physical probes are not provided in the source paper, CFD modeling includes inherent uncertainty related to iterative solution methods, convergence tolerances, discretization in space and time, and rounding error, as well as uncertainty related to the accuracy of the assumptions made in the mathematical models [2]. These are difficult to quantify individually but their cumulative effects commonly result in approximately 5% mean relative error for free surface hydraulic models [1].

Probe 1, shown in Figure 4, measures the reservoir free surface elevation during 40 seconds of draining. The reservoir draining curves match closely: the root-mean-square error (RMSE) is less than 3.8 mm and the mean relative error is less than 0.7%.

The numerical and physical depths in the channels are measured over 20 seconds at probes 3 and 5. Results are shown in Figures 5 and 6, and also match very closely. RMSE is less than 10 mm and 6.3 mm, and the mean relative error is less than 6.0% and 2.4%, for probes 3 and 5 respectively. The larger error measures at probe 3 are related to the 1-second difference in arrival time of the reflected wave front there; this may be due to inaccuracy in the estimated roughness height and drag coefficient used in the numerical model.

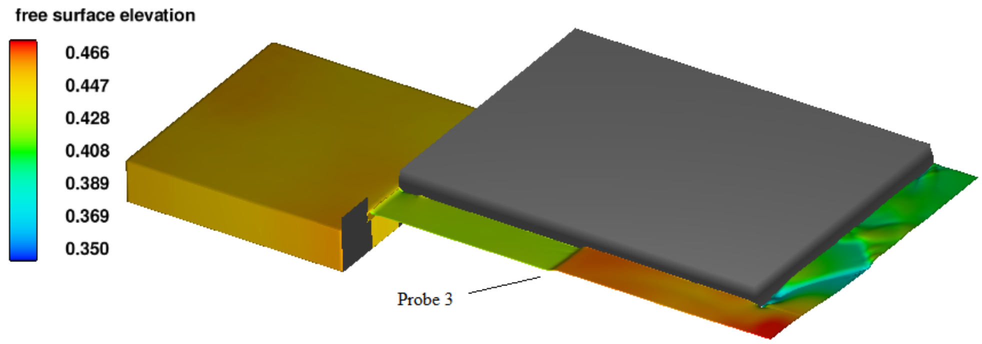

Figure 3: Numerical surface elevation at 12 seconds. Image is colored by surface elevation.

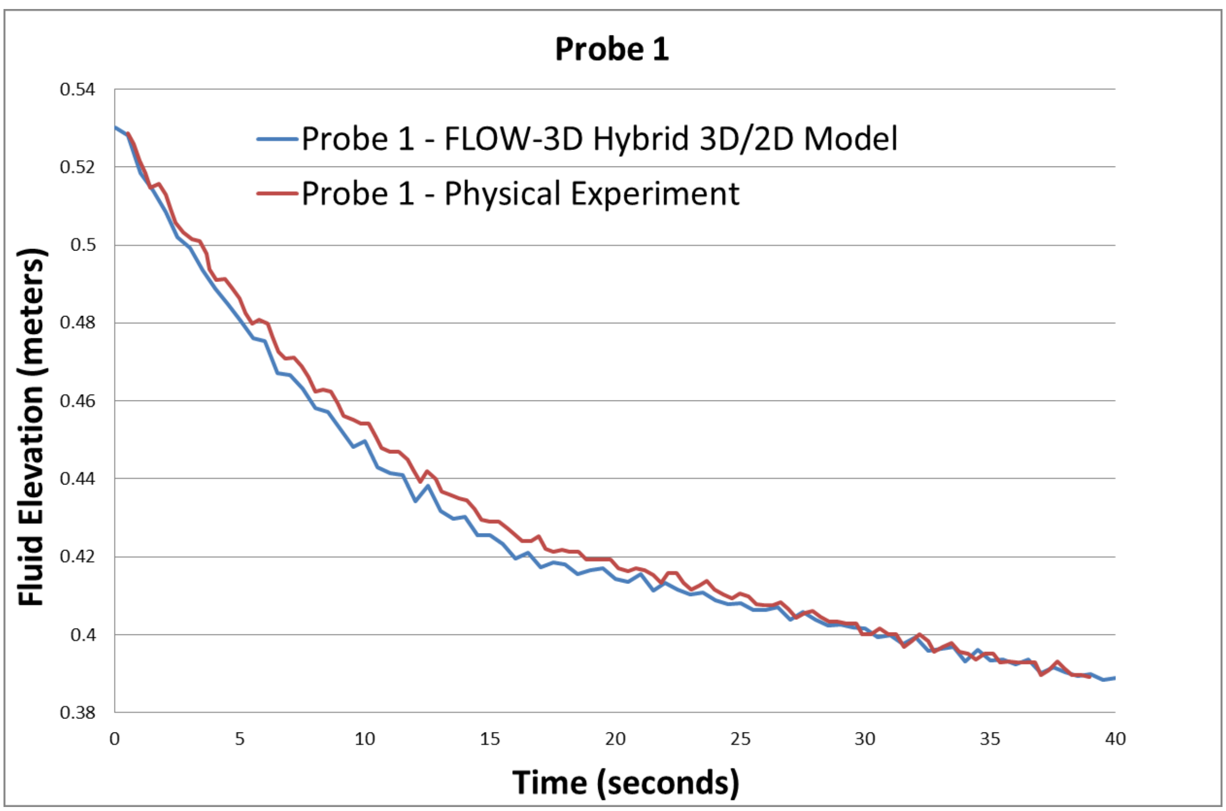

Figure 4: Fluid elevation in the reservoir with respect to time.

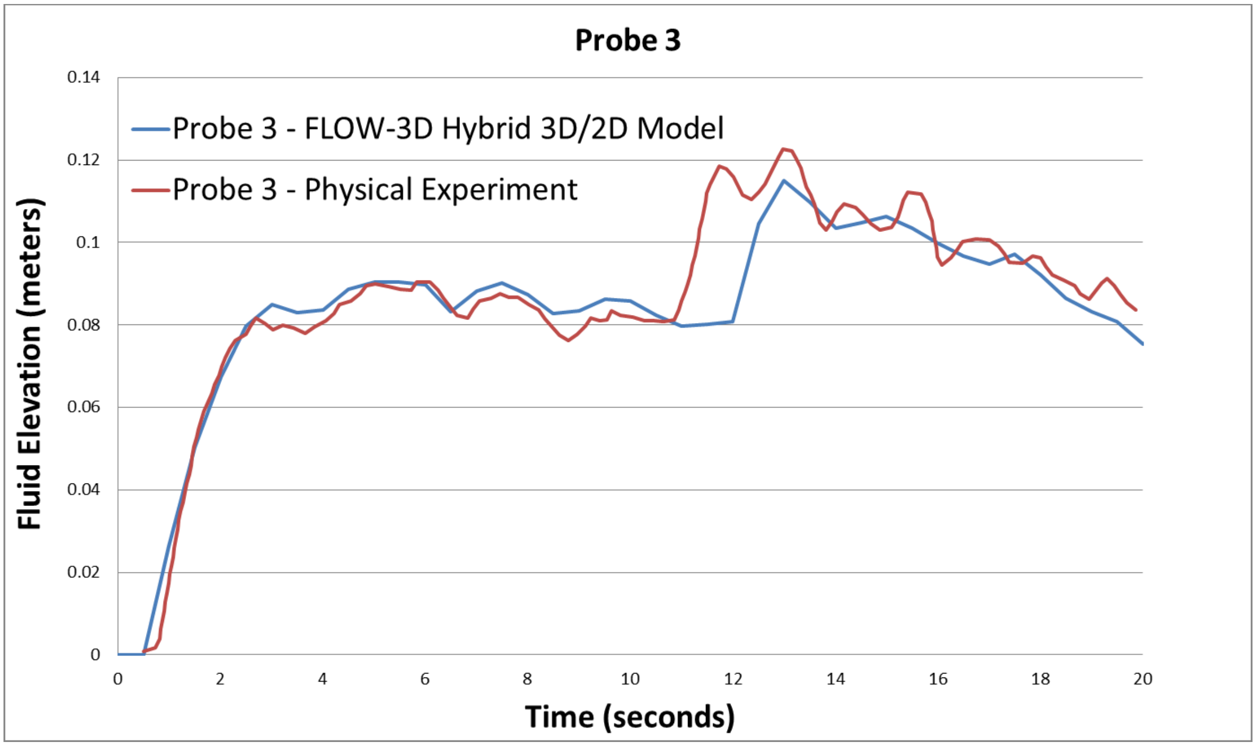

Figure 5: Fluid elevation in the first channel with respect to time.

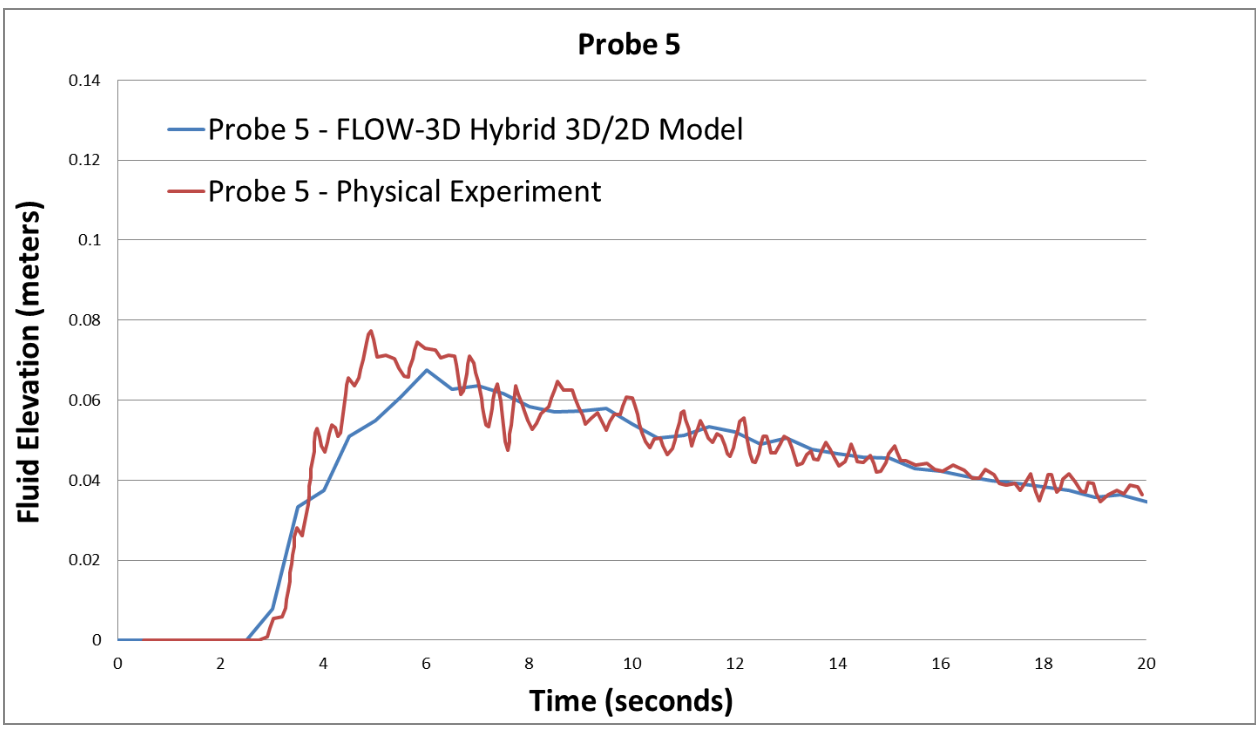

Figure 6: Fluid elevation in the second channel with respect to time.

Conclusion

A novel hybrid 3-D/2-D numerical model (commercial code ***FLOW-3D*** Version 10.1) was compared to a benchmark physical experiment. Root-mean-square error and mean relative error for primary and reflected wave heights were reported and found to be between 3.8 and 10 mm (0.7% and 6.0%) at three measurement locations. The accuracy of the tested commercial code is in keeping with the findings of other authors [5], [6], [7], [8], [9]. Additional testing will build confidence in the hybrid 3-D/2-D model for dam-break and inundation modeling.

References

[1] Burnham, J., *Minimizing Error Between Model and Reality: Guidelines and Trend Review*, 2012 FLOW-3D World Users Conference, September 13-14, 2012, San Francisco, CA. Flow Science, Inc., Santa Fe, NM.

[2] Erpicum, S., Dewals, B.J., Archambeau, P., Pirotton, M., *Dam-break Flow Computation Based on an Efficient Flux-Vector Splitting*, Journal of Computational and Applied Mathematics, Volume 234, Issue 7, 1 August 2010, 2143-2151

[3] Hirsch, C., "Numerical Computation of Internal and External Flows", 2nd Ed., 2007, Butterworth-Heinemann, Burlington, MA.

[4] Hirt, C.W., Nichols, B.D., *Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*, J. Computational Physics 39(1), 201-225, 1981.

[5] Kocaman, S. *Experimental and Theoretical Investigation of Dam Break Problem*, 2007, thesis: Department of Civil Engineering, Institute of Natural and Applied Sciences, University of Çukurova. In Turkish.

[6] Oertel, M. and Bung, D.B., *Initial Stage of Two-Dimensional Dam-Break Waves: Laboratory versus VOF*, Journal of Hydraulic Research, 2011, Taylor and Francis (online 08 Dec 2011).

[7] Ozmen-Cagatay, H. and Kocaman, S., *Dam-break flows during initial stage using SWE and RANS approaches*, Journal of Hydraulic Research, Vol 48, No. 5 (2010), pp. 603-611, doi: 10.108/00221686.2010.507342, © 2010 International Association for Hydro-Environment Engineering and Research. Available for purchase online at: informaworld.com.

[8] Ozmen-Cagatay, H. and Kocaman, S., *Investigation of Dam-Break Flow Over Abruptly Contracting Channel With Trapezoidal-Shaped Lateral Obstacles*, Journal of Fluids Engineering, 2012, ASME, Vol. 134 / 081204-1.

[9] Vasquez, J.A. and Roncal, J.J., Testing River2D and FLOW-3D for Sudden Dam-Break Flow Simulations, CDA 2009 Annual Conference, Congres annuel 2009 de l'A CB, Whistler, BC, Canada, 2009 Oct 3-8, pdf pages: 44-55.

Authors

David Souders is the Vice President of Sales, Marketing and Customer Support at Flow Science, Inc. His primary responsibility is to manage the development and execution of near-term and long-term marketing and sales campaigns and to ensure top quality customer support and training on use of Flow Science's software products.

Jayesh Kariya is a Testing & Validation CFD Engineer at Flow Science Software Pvt. Ltd. Pune. His holds a Master of Technology degree in Computational Fluid Dynamics, and is currently working on developing the user interface for the ***FLOW-3D*** software package and validating the software against experimental results.

Jeff Burnham, PE, LEED AP is a Hydraulics Applications Engineer at Flow Science, Inc. He conducts CFD training worldwide and assists ***FLOW-3D*** users in the water and environmental industry.