3D CFD modeling with FLOW-3D HYDRO

Brian Fox, Senior Applications Engineer,
Flow Science Inc., Denver, CO, brian.fox@flow3d.com

Abstract

FLOW-3D HYDRO is an advanced 3D CFD modeling software that has been used extensively for 40+ years in applications related to the design and analysis of dams/spillways, conveyance infrastructure, water treatment, coastal and river hydraulics. Core capabilities include a fully 3D non-hydrostatic solver with specialization in free surface flow applications. Additional multi-physics capabilities include a 3D sediment transport model, non-Newtonian flow rheology, air entrainment, heat transfer, and combined 2D/3D simulation domains.

Fully 3D CFD simulation software can provide valuable insights for testing design options, helping to reduce complexity and focus efforts on optimized solutions in situations where simplified numerical and physical modeling approaches are limited. This computer modeling demonstration will first provide a general overview of 3D CFD fundamentals and how it is different from other types of simulation and analysis tools. We will then discuss key technical features of the numerical model and demonstrate how it is used to go from problem statement to solution for a case study of the Garrison Dam, Missouri River, North Dakota. This will include an analysis of modeling approaches, model setup workflows, post-processing, and results validation. We will also explore the use of workflow automation and optimization tools.

Introduction

Over the past decade, 3D CFD has emerged as a commonly used and practical simulation tool for design and analysis in hydraulic engineering applications. These simulation approaches solve the fully 3D Navier-Stokes equations and can be applied to a broad range of complex scenarios where simplified, depth-averaged 1D and 2D hydraulic modeling tools may not be applicable. Common applications of 3D CFD simulation include the design and analysis of dams, spillways, fish passages, rivers, convenance systems, coastal defenses, and fluid/structure interaction among many others. The potential benefits of using 3D CFD simulation tools include improved design performance, risk reduction, decrease in maintenance and construction costs, stakeholder communication and enhanced environmental or regulatory compliance.

While 3D CFD simulation tools are widely used across many different engineering disciplines (mechanical, aerospace, etc.), applications within civil engineering can pose unique challenges for engineers. These can include requirements for resolving complex and irregular free surfaces; the need to resolve length scales that vary by orders of magnitude; and the effects of additional physics such as air entrainment or sediment transport. Compared to other simplified simulation approaches, practical usage of 3D models has additional considerations regarding computational resources and software usability.

FLOW-3D HYDRO is a commercial 3D CFD software that is commonly used within civil engineering. With advanced capabilities for simulating complex free surfaces, it can address many of the unique challenges faced in the design and analysis of hydraulic structures (Figure 1). Examples of early use cases within for civil engineering are summarized in Burnham (2011). These include a discharge analysis of Parshall flumes (Hirt and Williams, 1994); evaluation of hydraulic patterns within an already developed equilibrium scour hole at the base of a bridge pier (Richardson and Panchang, 1998); and the hydraulic performance of ogee weirs (Savage and Johnson, 2001; Johnson and Savage, 2006). More recent use cases include evaluation of sharp crested weirs (Sinclair, et al. ,2022); analysis of bendway weirs (Siefken et al, 2021); and investigation of cavitation potential (Yusuf and Micovic, 2020). An extensive list of publications using FLOW-3D HYDRO can be found at *https://www.flow3d.com/resources/bibliography/water-environmental-bibliography/.*

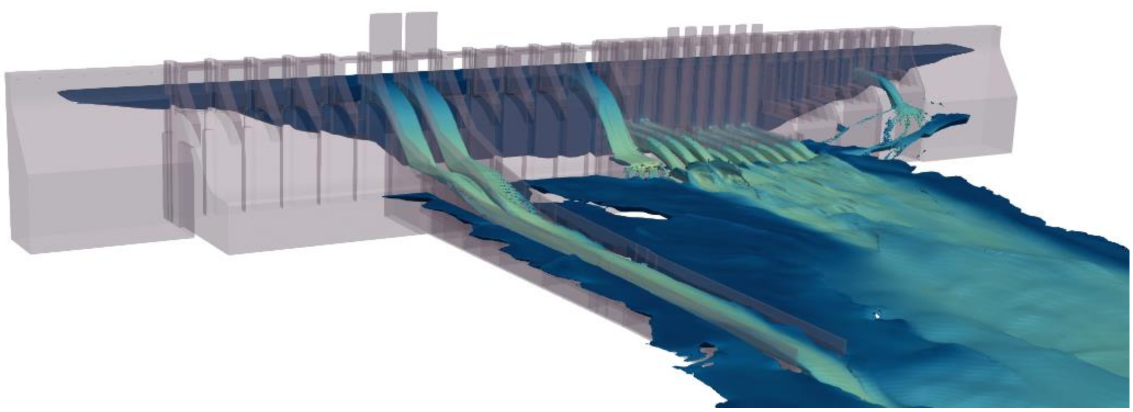

Figure 1. Example of general free surface flow simulations in FLOW-3D HYDRO.

The goal of this computer modeling demonstration is to highlight the key technical capabilities of FLOW-3D HYDRO and the types of applications where it can provide unique value. We also seek to demonstrate that 3D modeling tools can be used quickly and efficiently for a wide range of applications. This includes not only applications of highly complex and uniquely challenging hydraulic structures, but also for applications of relatively simple culverts, fish passages or other scenarios where additional 3D detail may be valuable. In all subsequent sections FLOW-3D HYDRO will be referred to as the "numerical model".

Model Description

The core of the technical capabilities of the numerical model include a solver for the fully 3D, non-hydrostatic, Navier-Stokes equations. Additional key technical features include options for RANS and LES turbulence modeling; the implementation of the Volume of Fluid (VOF) method for the tracking of complex free surface; and a free meshing approach for fast and efficient simulation of complex geometry. Other core capabilities include linkages with multi-physics models to simulate the coupled interaction of physical phenomena such as air entrainment, sediment transport, multi-species flows, etc.

Beyond the core technical capabilities, other key features include a fully parallelized solver for simulations on standard desktop workstations, high performance computing (HPC) clusters or cloud services. The model also includes a fully integrated GUI designed specifically for hydraulic engineering applications, post-processing for results analysis, and workflow automation and optimization tools.

Hydrodynamic Model

A derivation of the numerical model's governing equations and model capabilities can be found in Flow Science (2019). A key feature of the numerical model is the implementation of the VOF method for simulating free surfaces (Hirt and Nichols, 1981). The VOF method is a numerical technique used to track the location and movement of complex free surfaces and apply proper dynamic boundary conditions to those free surfaces. The current version of the numerical model incorporates major improvements beyond the original VOF method to increase the accuracy of boundary conditions and improve interface tracking (Barkhudarov, 2004). The numerical model has been used and validated extensively for a wide range of free surface hydraulic engineering applications (Burnham, 2011).

Another important feature of the numerical model is the use of a structured computational mesh that is composed of rectangular elements defined by a set of planes perpendicular to each of the coordinate axes. The numerical model uses a Fractional Area-Volume Obstacle Representation (FAVOR) method to incorporate the effects of geometry directly into the governing equations (Hirt and Sicilian, 1985). This approach calculates the open volume fraction and open area fractions to define obstacles in each cell and offers a simple and accurate method to represent complex surfaces without requiring a body fitted mesh (Figure 2).

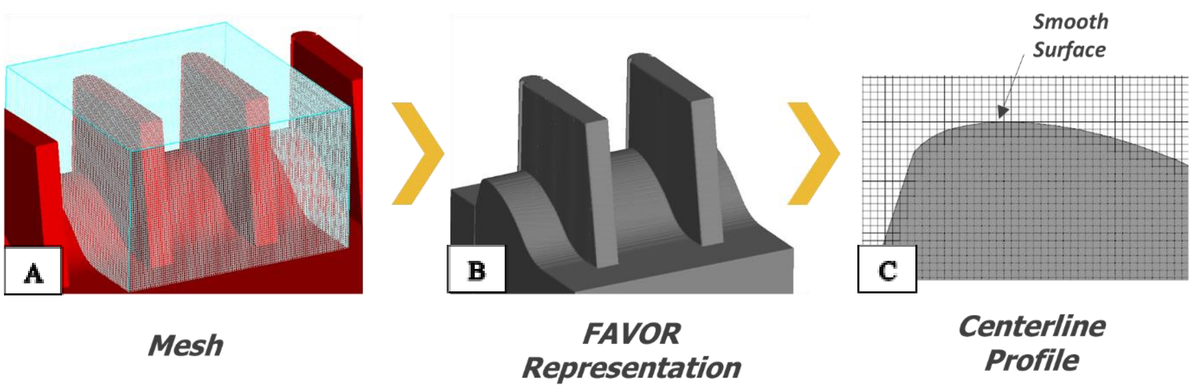

Figure 2. Example of meshing and irregular geometry handling in the numerical model using a structured Cartesian mesh. (A) Structured mesh is overlain with 3D CAD geometry of any arbitrary complexity. (B) FAVOR method interprets geometry as plane within each cell. (C) Profile of weir crest illustrating the results of the FAVOR implementation and representation of solid boundaries as flat surfaces.

The numerical model provides flexibility to modify and fine tune many of the numerical options within the solver. This includes defining the type and order-of-accuracy of numerical approximations, pressure solver convergence settings, implicit/explicit time stepping, and boundary layer treatments. Code customization and user-defined functionality is also available for a select number of sub-routines. This is a valuable option for research applications or to provide flexibility for other non-standard use cases.

Multiphysics Models

The numerical model is packaged with a suite of additional physics models that link the hydrodynamic solver with other physical phenomena such as air entrainment, sediment transport, heat transfer, particles, shallow water models, chemical reactions, multi-species flows, etc.

The Air Entrainment model predicts the entrainment of undissolved gas bubbles at free surfaces due to turbulent energy at the free surface and entrainment observed by impinging jets (Figure 3). Once air is entrained, it is treated as a dispersed multiphase flow to simulate the transport, diffusion, and escape of the entrained air bubbles, while also accounting for the mixture density and buoyancy effects of the entrained air within the fluid. It is used to simulate the effects of entrained air on flow bulking, overtopping, cavitation, and variable density hydrodynamics.

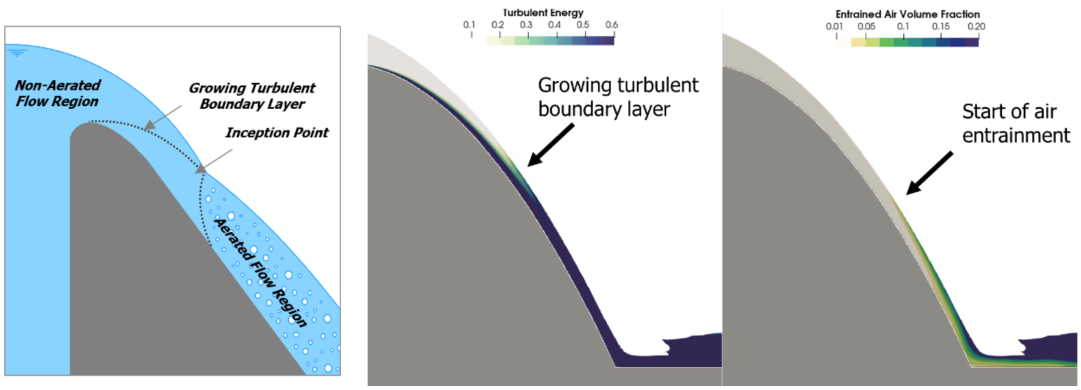

Figure 3. Conceptual example of air entrainment and transport multi-physics model.

The numerical model's sediment transport model simulates a 3D transient mobile bed. It is fully coupled with the 3D hydrodynamic solver to simulate the morphological changes to an erodible solid boundary. The model is capable of simulating bedload transport and suspended sediment transport. This allows for the exchange of material between the two transport mechanisms, erosion, and sedimentation. The effect of the bedform evolution, by scour and deposition, on the flow hydrodynamics are captured at each timestep. The model includes the capability to simulate up to 10 different sediment classes, where each defines a unique combination of grain size and material density. A non-uniform grain size distribution or variable sediment density can be simulated by defining multiple sediment species. A full description of the numerical model's sediment transport capabilities and a validation of pier scour (Figure 4) can be found in Fox and Feurich (2019).

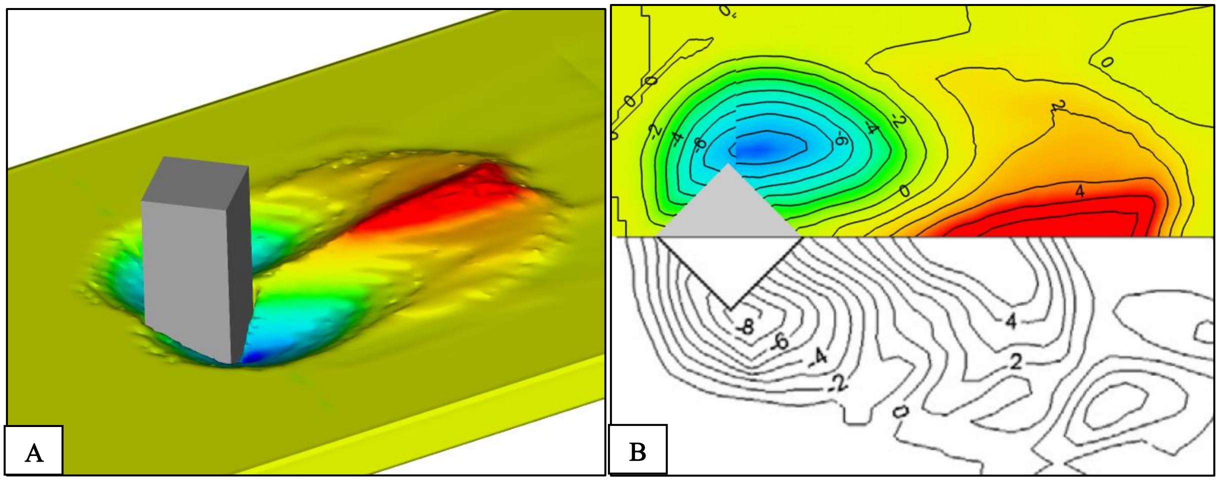

Figure 4. Equilibrium bed elevation changes predicted by the numerical model for the diamond pier. (A) Isometric view of scour and deposition adjacent to the pier. (B) Comparison between numerical results (top) vs physical model measurements (bottom). (Fox and Feurich, 2019)

The numerical model has the ability to run as a standalone 2D shallow water solver, which is also coupled with the available multiphysics models. A unique feature of the numerical model is the ability to link the shallow water model with fully 3D regions for hybrid 2D/3D modeling. This approach allows for dynamically linked simulation domain to balance the efficiency of a 2D shallow water solver with localized accuracy of a fully 3D solution (Figure 5).

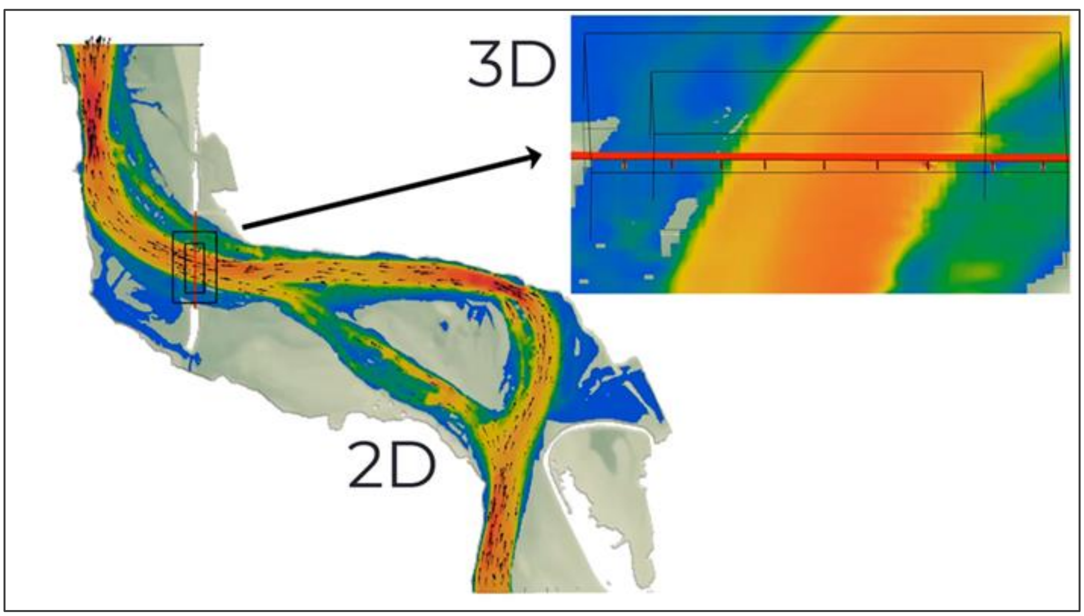

Figure 5. Example of dynamically linked 2D shallow water and 3D modeling domains. The area in the vicinity of the bridge was simulated using a fully 3D solver, while the approach and downstream regions were simulated using a 2D shallow water solver.

The Scalar model can be used to simulate the advection and diffusion of a passive or active scalar within the 3D flow field (Figure 6). Common applications for scalars include flow visualization, contaminant transport modeling and reaction kinetics. The Heat Transfer model is used to simulate the mixing and dispersion of variable temperature fluids, and accounts for the variable density effects of thermal stratification. Applications include the simulation of thermal plumes in rivers, lakes and estuaries.

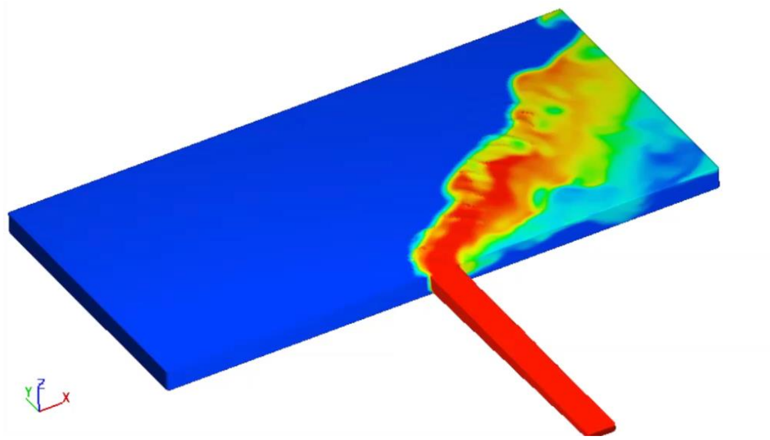

Figure 6. Example of passive scalar physics model, commonly used for simulation of jets, plumes and other flow mixing applications.

Additional key multi-physics models include the General Moving Object (GMO) model, which provides capabilities to simulate the motion of any solid object coupled with fluid flow with up to six degrees of freedom. This is often used to study the motion of sluice gates, flowing debris, waterwheels, and avalanches. The Tailings model allows for defining non-Newtonian flow properties as a function of spatially varying local sediment concentration. The Particle model tracks the motion of individual particles in the fluid domain. Marker particles are used for flow visualization, while physical properties can be defined for mass, gas, and fluid particles to simulate complex fluid-particle interactions. The Cavitation Potential model offers an efficient way of assessing possible cavitation by calculating the cavitation potential. Additional details on multiphysics capabilities can be found in Flow Science (2019).

High Performance Computing Capabilities

3D CFD simulation often requires a solution to computational domains with cell counts in the order of 1 M – 100 M. Because of the high computational demand compared with 1D and 2D simulation tools, computing performance and solver efficiency are critical aspects for practical applications. The numerical model is fully parallelized using a hybrid MPI/OpenMP approach, that allows for fast and efficient simulations on a broad range of computing hardware (Figure 7). Common desktop configurations, ranging from 4 core laptops to 32 core desktop workstations, are sufficient for a wide range of applications. For simulations requiring large cell counts and fast run times, the numerical model can also be used with in-house high-performance computing (HPC) clusters or commercial cloud services.

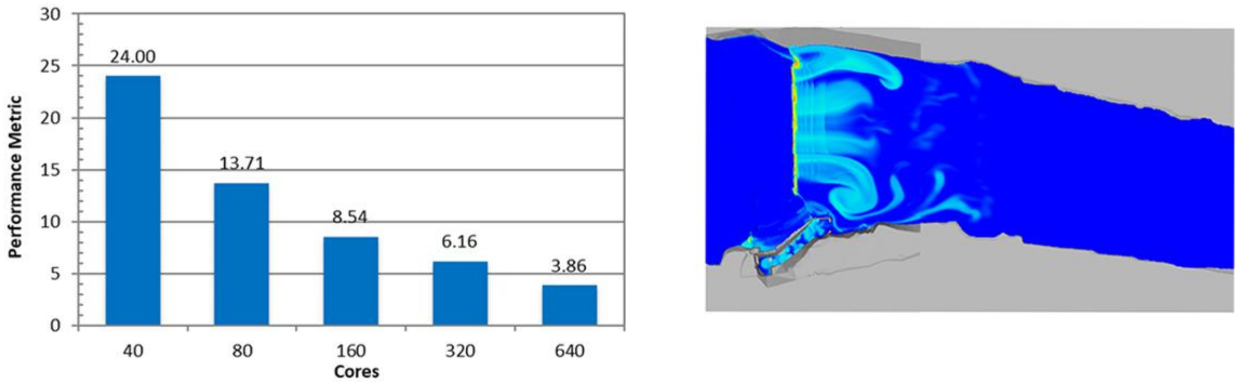

Figure 7. Hydraulics benchmarking case demonstrating scaling performance and 8x speed up in simulation run times. Performance metric is a normalized run time, where decreasing values are indicative of simulation speed up.

Garrison Dam Case Study

The Garrison Dam is located on the mainstem of the Missouri River in west central North Dakota. The original design report (WES, 1956) describes the physical modeling efforts to evaluate the hydraulic performance of the spillway and outlet works. Drawing on data from this report, we will demonstrate the use of the numerical model to repeat this analysis with the goal of predicting spillway flow capacity and pressures under a range of reservoir elevations and gate openings. The demonstration will include an analysis of simulation strategy, model setup workflows, and validation of simulation results with measured data. We will also demonstrate the use of optimization and automation tools to accelerate repetitive simulation tasks.

A description of the Garrison Dam spillway is provided in the 1956 Technical Memorandum No 2-431 (WES, 1956). Key features of the spillway include an arced ogee weir with a gross length of 1,444ft; 28 radial gate bays 40ft wide x 29 ft high; and a non-uniform approach channel (Figure 8).

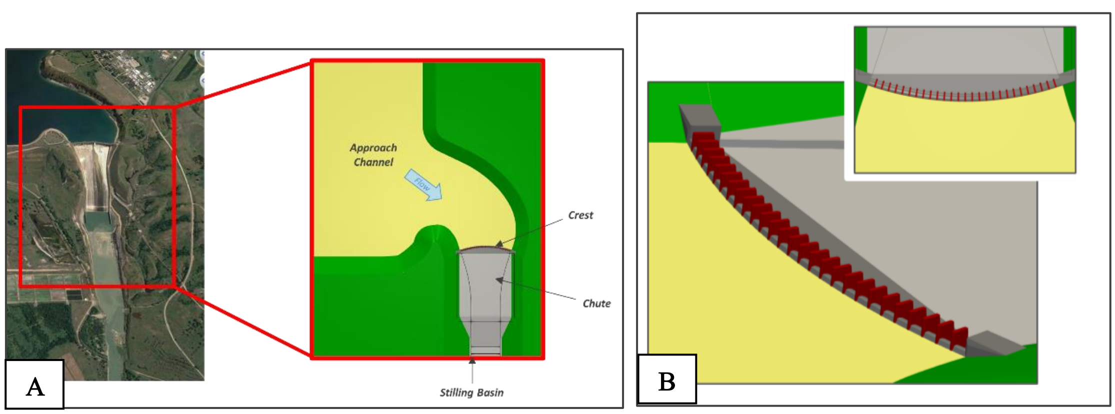

Figure 8. (A) Overview of Garrison Dam spillway study area. (B) The primary area of interest includes an arced ogee weir with a gross length of 1,444ft; 28 radial gate bays 40ft wide x 29 ft high; and a non-uniform approach channel.

In general, the numerical model has been extensively validated for similar applications of discharge prediction at hydraulic controls. However, the unique challenge of this specific case is range of applicable length scales. This includes a relatively large domain to accurately represent the non-uniform approach flow conditions, while simultaneously requiring a relatively fine resolution at the weir crest to ensure accurate discharge predictions. To satisfy these requirements, a primary practical challenge will be to define a simulation strategy and workflow to manage the cell count that will balance simulation run time and accuracy.

To efficiently address these challenges, we take an incremental approach in the model setup by starting with a simple, 2D vertical slice simulation of the ogee weir crest; then transition to a 3D simulation for half of a single bay; and finish with a simulation of the full domain using the 2D shallow water model to simulate approach flow conditions (Figure 9). This gradual increase in model setup complexity provides an efficient method for quickly testing a broad range of setup options before applying them to larger domains and longer running simulations. This method further allows for validation of the numerical model at matching stages relative to the physical modeling effort described in the report.

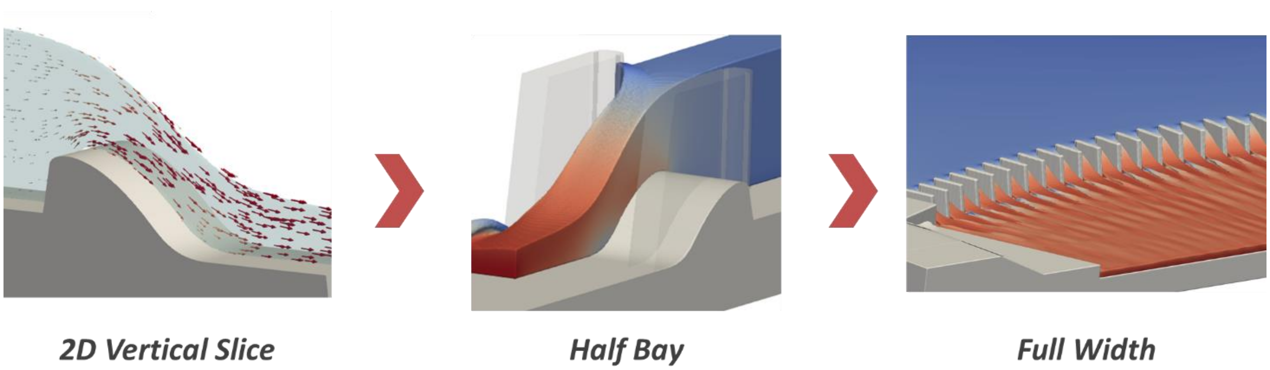

Figure 9. Conceptual modeling approaches of incremental increase in setup complexity.

The first step of this workflow is setup the simulation of the 2D vertical slice of the weir profile. This is used to isolate the vertical contraction through the weir and determine the required mesh size and numerical settings to achieve a desired level of accuracy. After creating an initial simulation template, the automation extension, FLOW-3D(X), was used to visually construct a workflow to setup, run and post-process simulation results from 3 different weir crest profiles, 5 mesh sizes, and 2 advection schemes (Figure 10). These were all tested for 6 different headwater elevations and resulted in 180 individual simulation runs. Results were compared with validation data from WES (1956). Simulation results confirmed a mesh cell size equal to 1/10 of the flow depth at the weir crest provides for discharge predictions within 2.5% of measured data (Figure 11).

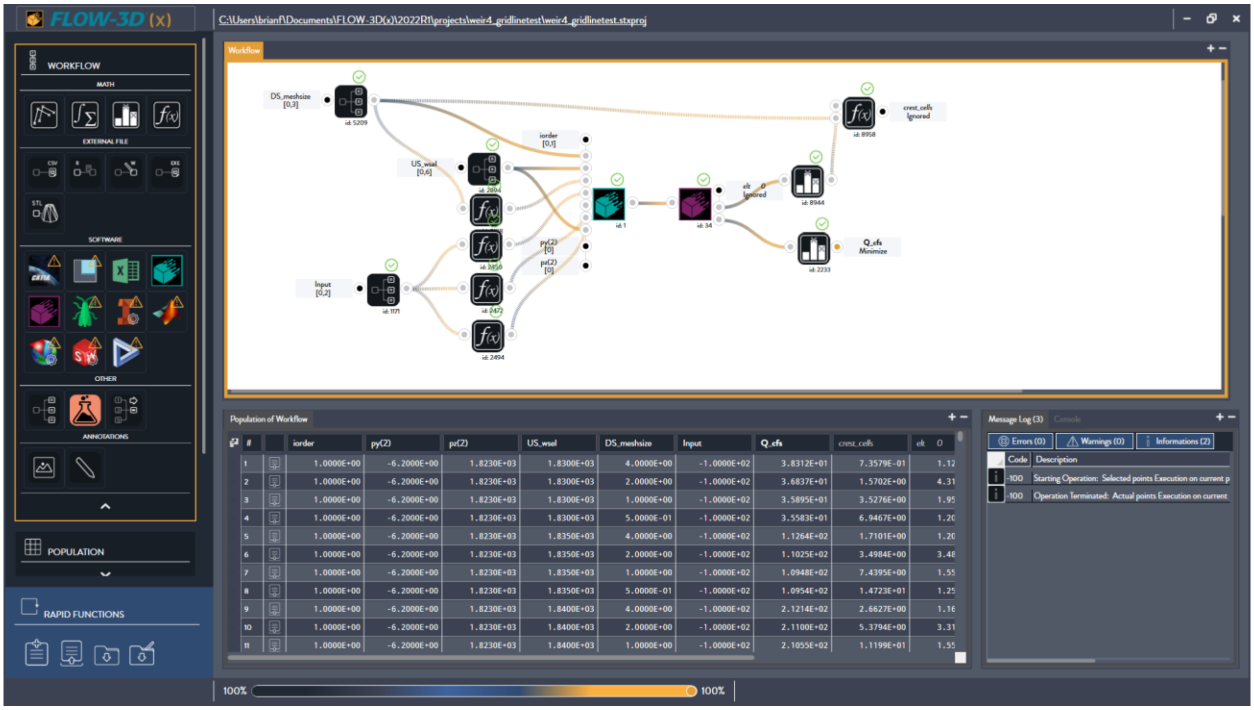

Figure 10. FLOW-3D (x) is a workflow automation/optimization extension of the numerical model. In this example it was used to automatically iterate through 180 individual simulations with different combinations of crest geometry, mesh size, momentum advection, and headwater elevation.

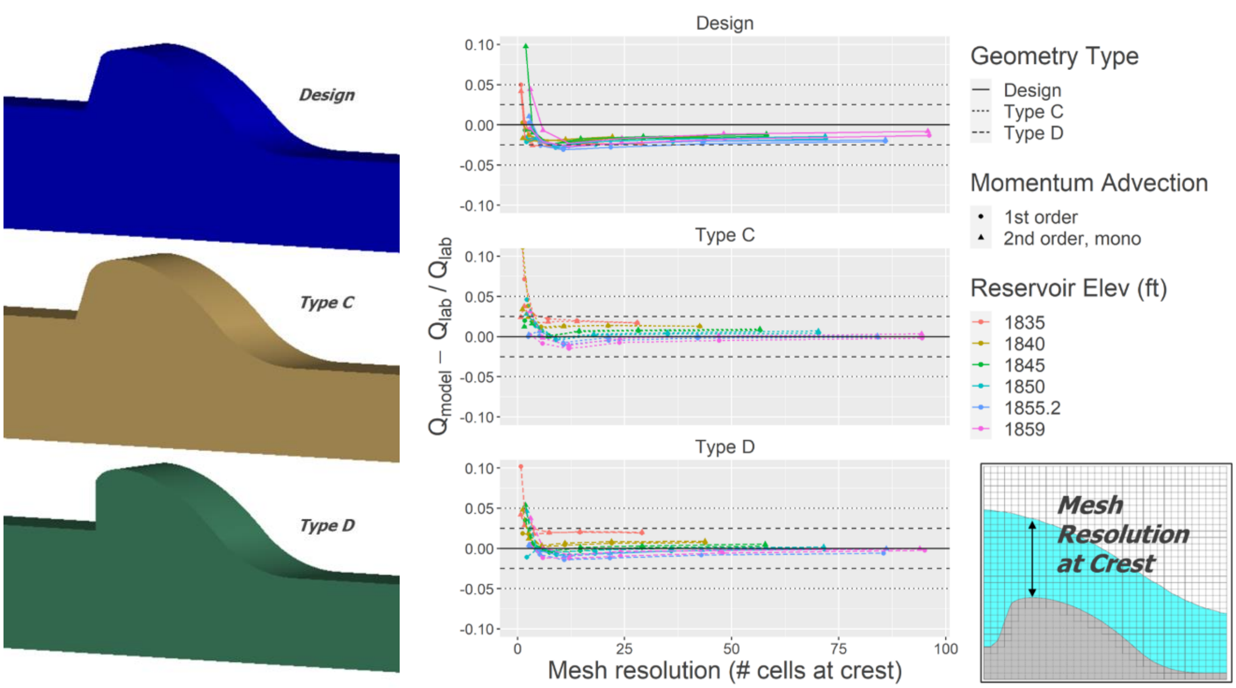

Figure 11. Model validation results for 2D vertical slice simulations. Results compare the predicted vs measured discharge for a defined headwater elevation.

The next phase of our project workflow was to convert the 2D slice model to a fully 3D simulation for half of a single bay. This is easily accomplished within the numerical model's GUI by adjusting the mesh dimensions. Mesh size and numerical options identified in the 2D slice tests were then validated with data from WES (1956), for both gated and ungated flow conditions. Spillway pressures were extracted from the numerical model results and compared against pressure measurements available in the physical modeling report and showed close comparison for all cases, including the identifying conditions with negative pressures (Figure 12).

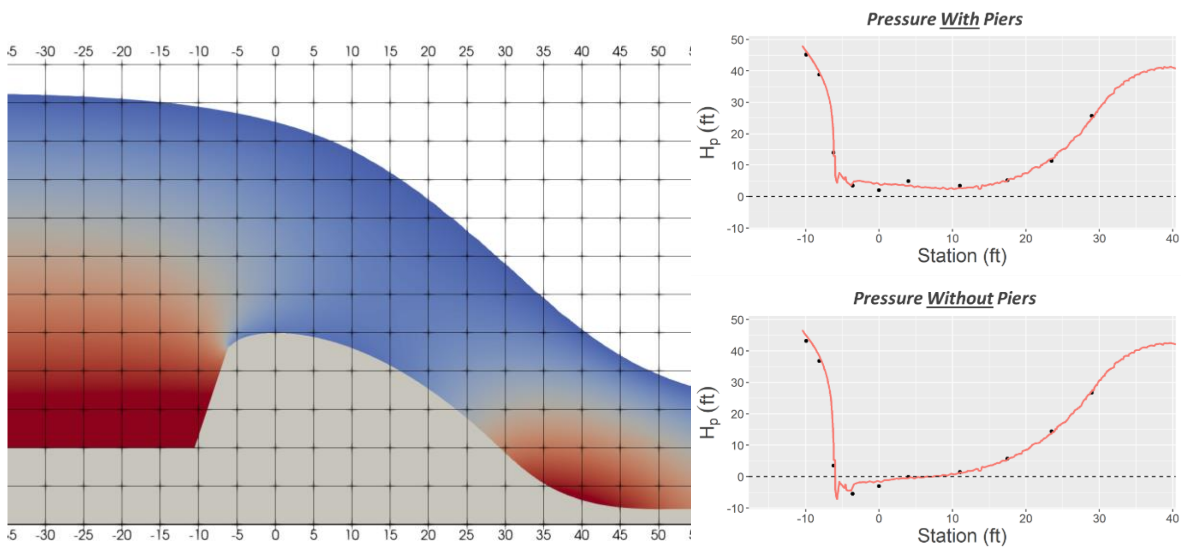

Figure 12. Validation comparing predicted (red line) vs measured (black point) spillway surface pressure.

The final step in the project workflow is simulation of the full domain to include the full weir width and upstream approach flow. Within the numerical model's GUI, the 3D mesh was modified to include the entire spillway width, and nested 3D meshes were added for higher resolution at the weir. Conforming mesh options were selected to restrict the most refined 3D mesh within a specified distance of the weir. Upstream of the weir, a 2D shallow water mesh block was created to allow for the development of non-uniform approach flow conditions (Figure 13).

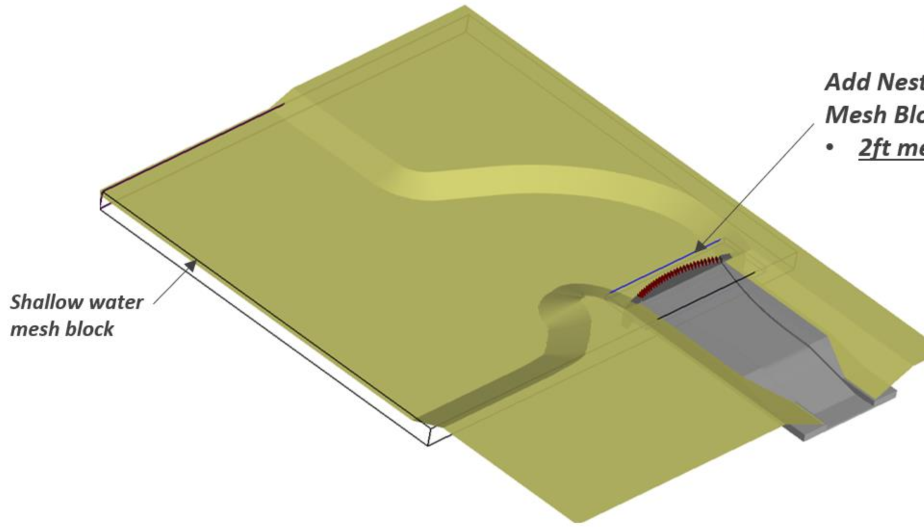

Figure 13. The area in the vicinity of the weir is simulated using a fully 3D mesh, while the approach flow is simulated using a 3D shallow water mesh. The combined 2D/3D simulation approach allows for efficient balancing of accuracy and runtime for practical applications.

The combination of the 2D and 3D allows for a balance of run time efficiency and accuracy by reducing cell count in the approach channel where a fully 3D solution is not required (Figure 14/15). The simulation results for the full domain were compared with data from WES (1956), and validated within 5% of measured data.

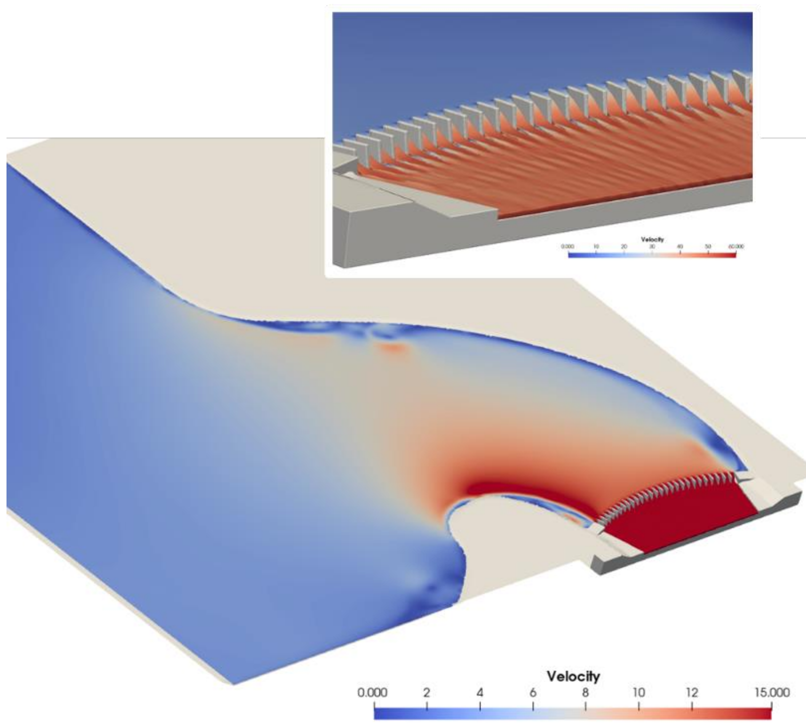

Figure 14. Simulation results of the full domain model. The combined 2D and 3D modeling domains provide an efficient approach to balance run time efficiency and accuracy.

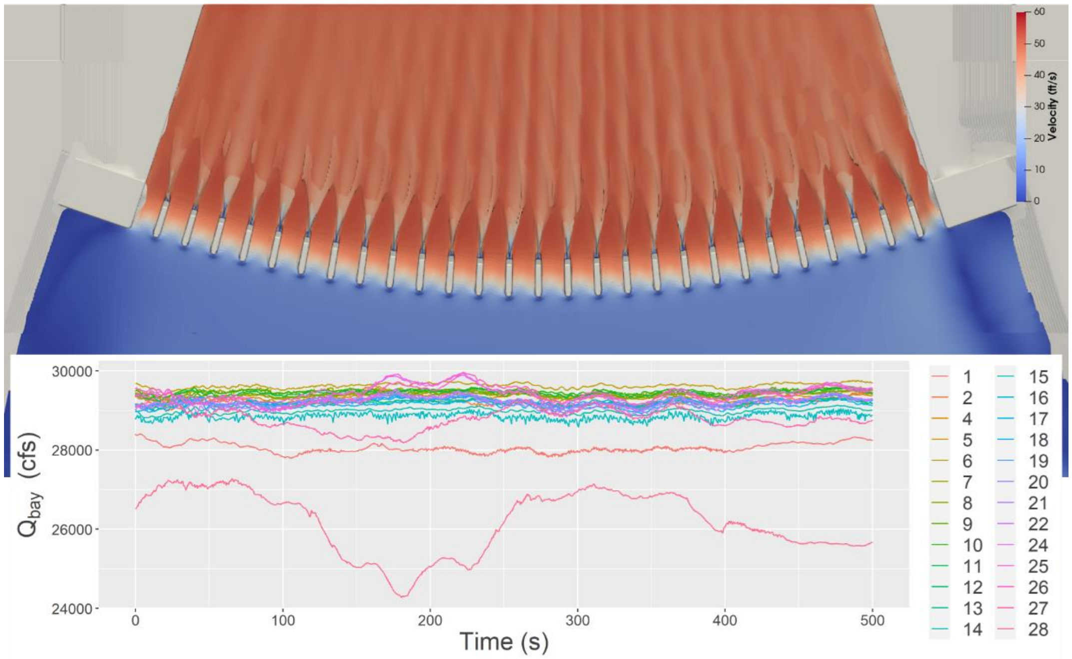

Figure 15. Plan view of full domain model within the vicinity of the weir. Computed flow rates through each of the 28 individual gate bays show a reduction in discharge in both of the outer bays.

Summary

FLOW-3D HYDRO is a commercial 3D CFD simulation software that provides advanced modeling capabilities where simplified modeling approaches may not be suitable or full 3D resolution is required. The numerical model includes technical solver capabilities uniquely suited to civil hydraulic engineering applications. These include an advanced VOF implementation for free surface flows; fully coupled multiphysics; and fast, simple and robust meshing. Additional features such as a fully integrated GUI, custom workflows, and automation tools with FLOW-3D (X) allow for fast and efficient practical usage.

This numerical model's capability is demonstrated through the Garrison Dam case study to evaluate spillway capacity. We demonstrate the use of the numerical model starting from a small-scale evaluation of flow through a simple hydraulic control. This includes sensitivity testing and validation using FLOW-3D(X). The simplified modeling approach is easily extended to a fully 3D simulation, and a coupled 2D/3D shallow model of the full domain. In all cases, the model predicted discharge validates within 2.5% of measured data.

References

Barkhudarov, M. R. 2004. "Lagrangian VOF Advection method for FLOW-3D". Flow Science Inc, 1(10).

Burnham, J. 2011, "Modeling Dams with Computational Fluid Dynamics-Past Success and New Directions", Dam Safety 2011, National Harbor, MD.

Flow Science. 2022. FLOW-3D HYDRO® Version 2022r2 Users Manual. Santa Fe, NM: Flow Science, Inc. https://www.flow3d.com

Fox, B. and Feurich, R. 2019. "CFD analysis of local scour at bridge piers". In Proc. Federal Interagency Sedimentation and Hydrologic Modeling SEDHYD Conference (pp. 24-28).

Hirt, C. W., & Nichols, B. D. 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of computational physics, 39(1), 201-225.

Hirt, C.W. and Sicilian, J.M. 1985. "A porosity technique for the definition of obstacles in rectangular cell meshes". 4th International Conference on Numerical Ship Hydrodynamics, 4th.

Hirt, C. W., and K. A. Williams. 1994. "FLOW-3D predictions for free discharge and submerged Parshall flumes." Flow Science Technical Note.

Johnson, M.C. and Savage, B.M. 2006. "Physical and numerical comparison of flow over ogee spillway in the presence of tailwater". Journal of hydraulic engineering, 132(12), pp.1353-1357.

Richardson, J. E., & Panchang, V. G. 1998. "Three-dimensional simulation of scour-inducing flow at bridge piers". Journal of Hydraulic Engineering, 124(5), 530-540.

Savage, B.M. and Johnson, M.C. 2001." Flow over ogee spillway: Physical and numerical model case study". Journal of hydraulic engineering, 127(8), pp.640-649.

Siefken, S., Ettema, R., Posner, A., Baird, D., Holste, N., Dombroski, D.E. and Padilla, R.S. 2021. "Optimal configuration of rock vanes and bendway weirs for river bends: Numerical-model insights". Journal of Hydraulic Engineering, 147(5), p.04021013.

Sinclair, J.M., Venayagamoorthy, S.K. and Gates, T.K. 2022. "Some Insights on Flow over Sharp-Crested Weirs Using Computational Fluid Dynamics: Implications for Enhanced Flow Measurement". Journal of Irrigation and Drainage Engineering, 148(6), p.04022011.

Yusuf, F. and Micovic, Z., 2020. "Prototype-scale investigation of spillway cavitation damage and numerical modeling of mitigation options". Journal of Hydraulic Engineering, 146(2), p.04019057.

Waterways Experiment Station (WES). 1956. "Outlet works and spillway for Garrison Dam, Missouri River, North Dakota". Technical Memorandum No. 2-431.