Getting in the Flow of Hydraulic Solutions

By Andy McCoy, Ph.D., P.E. – Senior Water Resources Engineer, Des Moines, IA ; Adrian Strain, P.E. – Water Resources Project Engineer, Des Moines, IA; Hany Gerges, Ph.D., P.E. – Operation Assistance Director, Walnut Creek, CA

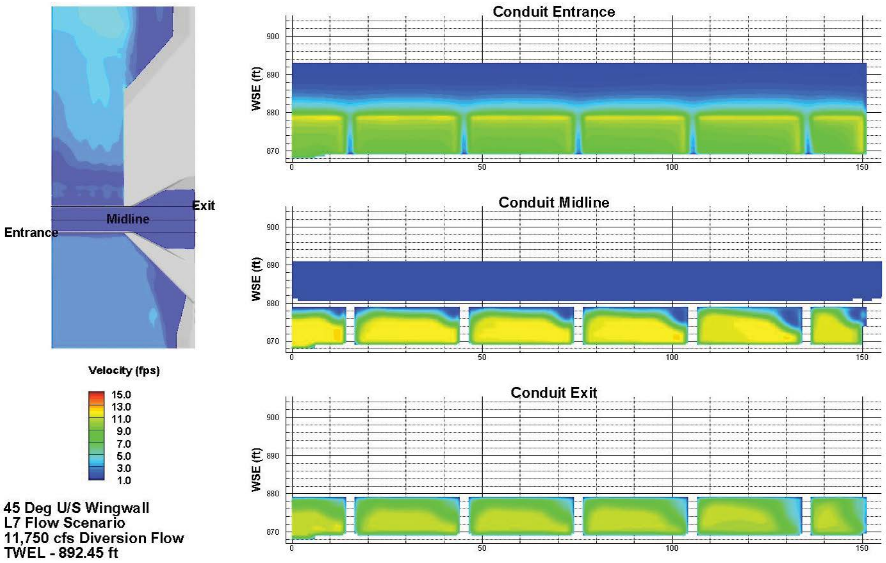

Figure 1: Velocity contours from an aqueduct design

Computational Fluid Dynamics (CFD) solves two- and three-dimensional equations of fluid motion to provide detailed solutions to a multitude of hydraulic issues. A design, permit application, or plant improvement often requires solving for and leveraging a detailed three-dimensional velocity field. We have developed CFD models that include a wide range of in-plant hydraulics projects (grit chambers, clarifiers, wet wells, contact tanks) and water resources projects (reservoirs, spillways, intakes, gates, fish collectors). Modelers choose the appropriate modeling methods, and the solutions are customized for the projects' needs, which provide valuable insight to the design and analysis teams. Exploring and designing around the fundamentals of how water moves through a system, whether in-plant, near a hydraulic structure, or in a riverine system, helps create solutions that are efficient and cost effective over the long run.

CFD models range in scale from several miles (riverine systems and reservoirs) to fractions of an inch (near bed turbulence, sediment transport, and screens). The trick is applying the right amount of detail in the appropriate area. Understanding the technical issues the designer is facing is paramount to creating a CFD model that satisfies the needs of the project.

CFD models are rarely a stand-alone component of the design process. Depending on the project type, the CFD model can be paired with 1D and/or 2D hydraulic models to enhance the overall understanding of a particular component of the hydraulic system and the issues facing the design team. 1D/2D models can be used to provide boundary conditions and serve as a calibration tool when field data is not available. Physical models are commonly paired with CFD models when analyzing hydraulic characteristics within facilities and natural riverine environments. This happens in projects where the length scales exceed practical limits or they are investigating flow phenomena that stretch to the limit of the underlying numerical model assumptions. In these cases, CFD models still can be used to inform the initial and preliminary design decisions followed by physical modeling. This approach limits the number of physical model iterations, saving time and money during the design process. The physical modeling can also be used to validate the CFD results.

CFD modeling can be incorporated at any point in the design process. However, it is recommended to engage the modeling team as early as possible. Good communication between the design and modeling teams leads to appropriate model complexity in the development stage and a proper understanding of model limitations associated with the results. The following examples and case studies illustrate the benefit of adding

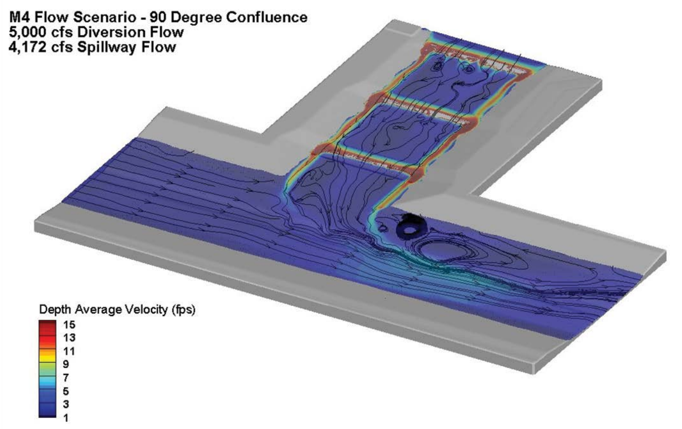

Figure 2: Depth-averaged velocity magnitudes and streamlines for this confluence design concept

CFD to the design process. In each of these examples, the modeling and design teams used CFD technology to evaluate and explore how water moves through a system (both engineered and natural). This level of understanding and resolution has enabled efficient and cost-effective solutions.

Aqueduct and Confluence Design Project

As part of a Flood Risk Management Project performed for the St. Paul District of the U.S. Army Corps of Engineers, CFD tools were used to evaluate an aqueduct that was part of a proposed solution to carry flow and to maintain aquatic and hydraulic connectivity in the Maple River over a flood diversion channel. The configuration included a spillway with a control weir located a short distance upstream from the proposed aqueduct. The spillway control weir was designed to divert high flows from the Maple River into the diversion channel, reducing flow in the Maple River channel downstream of the aqueduct during flood events.

Due to the complexity of the hydraulics associated with these structures, numerical and physical models were created to evaluate the interaction of the Maple River spillway and aqueduct structure. Numerical and physical modeling were conducted to verify assumptions made during feasibility design, to refine the final design, to assess potential ice problems associated with the structure, and ultimately to produce final design criteria.

As part of efficient pairing of multiple hydraulics tools, 1D and 2D models were initially used to determine the impacts of the aqueduct on the diversion channel flow during flood conditions. Head loss in the diversion channel due to pressure flow at the aqueduct became a concern after the initial modeling results.

The flows from the selected recurrence intervals in the diversion channel were large enough to create pressure flow at the aqueduct crossing. Due to the nature of the local topography constraints (very flat), minimizing head loss at the aqueduct was a requirement. Several modifications were analyzed, including the leading edge of the piers as well as the upstream and downstream wingwall configuration, to determine which physical aqueduct configuration was capable of lowering the overall head loss on the diversion channel flow. The final design struck a balance between hydraulic efficiency and constructability. Figure 1 shows velocity contours from one of the analyzed aqueduct designs.

A second model for the project was developed to better understand the hydraulic characteristics of the confluence between the spillway and the diversion channel. During flood events, a portion of the flows continues through while the rest takes an alternate path down the spillway and into the diversion channel. Multiple confluence alignments were modeled to minimize downstream impacts in the diversion channel, such as scour and unacceptable flow patterns. Figure 2 shows the depth-averaged velocity magnitudes and streamlines for one of the confluence design concepts. The recirculation zones and high-velocity areas helped identify locations that may be susceptible to erosion and deposition processes, resulting in appropriately designed scour counter measures and maintenance planning.

Non-Conventional Gate Application

The non-conventional use of a miter gate was proposed during the design of a flood control project. Miter gates are typically used in applications such as navigation locks. These systems operate under a static water condition. Gates open or close when the water surface is approximately equal on both sides of the gate. A miter gate that operates while flow is passing through the gate channel is considered a non-conventional application. The miter gate system experiences increased stresses and forces while in operation during a moving water condition as compared to a static water condition.

To gain confidence in the gate design, a CFD study was performed. The simulations combined the Large-Eddy Simulation (LES) turbulence model with the General Moving Objects (GMO) module. The closing motion of the gate was prescribed and key design factors affecting the operational parameters of the miter gate were identified. Upstream and downstream boundary conditions were prescribed from 2D model results, and several different water levels were simulated.

Data from the CFD simulations were applied to an uncoupled structural model to determine deflections and mechanical loads. Several analysis points were identified and data was extracted from the results using arbitrary surfaces to seed the structural model.

Since the gate will operate during moving water conditions, additional information was necessary to produce the mechanical and structural analysis required, and more detailed information was needed to design around the additional hydrodynamic loading.

(continued on next page)

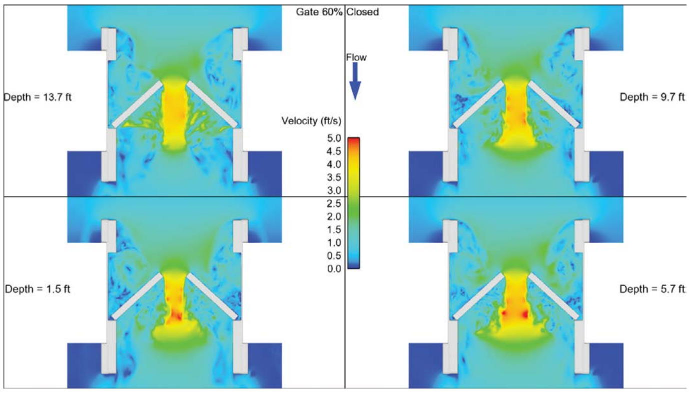

Figure 3: Velocity at multiple depths as the gate closes

The end product of the CFD model was hydrostatic and hydrodynamic forces acting on the gate surfaces. CFD model results were coupled with a numerical structural model to guide the mechanical and structural design. Figure 3 shows the velocity at multiple depths while the gate is closing.

Municipal Water Intake

A CFD model was used to qualitatively determine the capture efficiency and potential sedimentation for a municipal water intake within the Yellowstone River for Great West Engineering and the City of Laurel, Montana. Coupled with the results of a 1D model, the CFD model was developed using a hybrid 2D/3D approach. To expand the model domain but maintain a high degree of resolution near the intake, a 3D fully hydrodynamic mesh was inserted into a 2D model.

Results from the initial modeling showed the intake was not parallel with the flow in the channel, lowering the capture efficiency. The intake was reoriented to parallel with the flow for the analyzed operation range. The second concern for intake design was the potential for sediment deposition within the intake. Since the intake would reside in a river with a highly mobile bed, the velocities inside the intake were evaluated. The results showed the velocities within the intake would remain high enough to allow sediment to pass through while maintaining the desired water withdrawal. Figure 4 shows the streamlines inside and near the intake.

Flow Distribution for Secondary Clarifiers

CFD models are often used to study in-plant hydraulics; one example of this is flow distribution within a closed system. A symmetrical CFD model was created to determine the approximate flow distribution for a series of secondary clarifiers. The existing design used a series of butterfly valves located at each clarifier to create the desired flow distribution. The process of adjusting the multiple valves at each of the eight clarifiers was time consuming and difficult for operations staff to manage. As part of the facility upgrade process, an option for a less laborious process to distribute the flow was requested.

The CFD model was set up to show symmetry along the center axis of the mixed liquor channel that supplied eight of the clarifiers. The major assumption of the model was the flow would be evenly distributed on either side of the axis of symmetry. The axis of symmetry allows for shorter model run time and/or increased

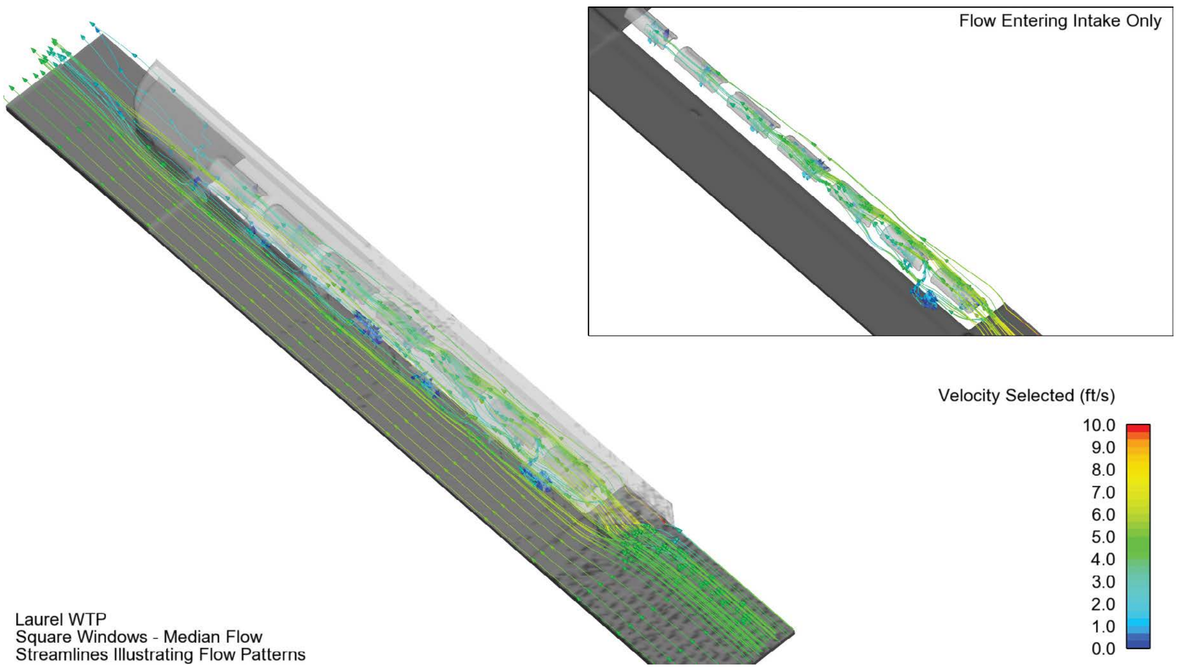

Figure 4: Streamlines illustrating flow patterns

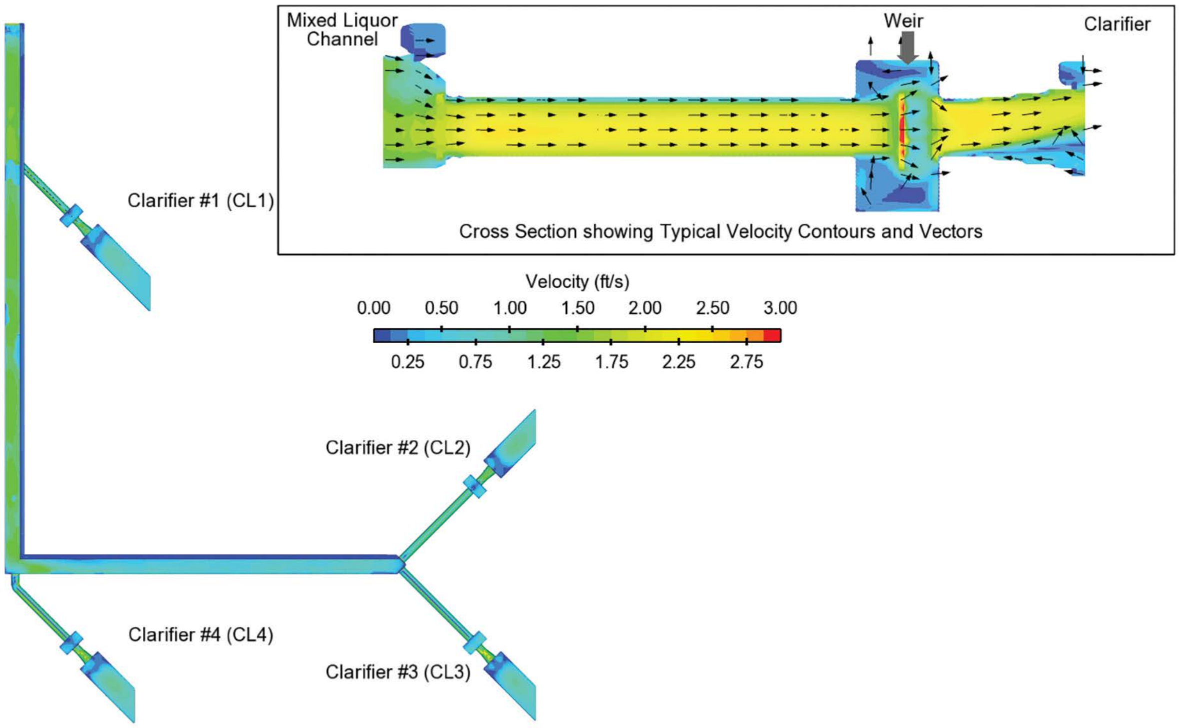

Figure 5: typical velocity profile across the weir and the velocity contours in the mixed liquor channel

level of detail without running into hardware constraints. Using an axis of symmetry is common practice in CFD models and a valid assumption based on known hydraulic behaviors.

The CFD model started with examining the flow distributions for the existing facility. High-flow and low-flow cases were run with the valves fully open to determine the flow split for the existing conditions. The existing conditions showed an imbalance of flow across the system when allowed to operate with the butterfly valves fully open. One possible solution to eliminate the task of adjusting multiple valves for each clarifier was a weir concept that would provide passive flow distribution. A series of weir iterations were run in the CFD model to determine weir dimensions that would allow for approximately equal flow distributions for the system. Figure 5 shows typical velocity profile across the weir and the velocity contours in the mixed liquor channel.

The final proposed design featured a unique weir elevation for each of the modeled clarifiers. Since the CFD model represents half of the clarifiers, the weir characteristics can be applied to their opposite counterpart to complete the design.

Customized CFD Solver for In-Facility Processes

CFD can be used to analyze two phase flows for in-facility processes. A customized CFD model was developed in collaboration with the software developer to properly simulate multiphase flow. The customized CFD model has been used to determine the removal efficiencies of the clarifiers by adding settling velocity equations for primary clarifiers to the source codes. By using such a customized model, the project team was able to investigate the effect of different structural modifications and identify the most cost-effective ones. Other applications of customized code have included the addition of sludge rheology equations to simulate flow and mixing patterns in sludge-blending tanks and digesters.

For more information about this article, please contact Andy McCoy at Andrew.McCoy@hdrinc.com, Adrian Strain at Adrian.Strain@hdrinc.com, or Hany Gerges at Hany.Gerges@hdrinc.com