HYBRID HYDRAULIC MODELING OF RIVER-TRAINING STRUCTURES IN SINUOUS CHANNELS

S. Michael Scurlock, Ph.D, Post-Doctoral Fellow, Colorado State University, scurlock@engr.colostate.edu; Amanda L. Cox, Ph.D, Assistant Professor, Saint Louis University, coxal@slu.edu; Drew C. Baird, Ph.D, Hydraulic Engineer, United States Bureau of Reclamation, dbaird@usbr.gov; Christopher I. Thornton, Ph.D; Associate Professor, Director Engineering Research Center, Colorado State University, thornton@engr.colostate.edu; Steven R. Abt, Ph.D; Senior Research Scientist/Scholar, Colorado State University, sabt@engr.colostate.edu

Abstract: Hybrid hydraulic modeling research, which integrates physical and numerical modeling, is currently being conducted to develop empirical design procedures for river-training structures. Transverse features are rock structures, usually installed in series around a river bend, which can control near-bank flow velocities, increase bank stability, decrease the effect of secondary currents, and promote habitat. An extensive database was developed from physical modeling of transverse features in a native-topography channel. The laboratory database is being used to calibrate and validate computational fluid dynamic (CFD) models, which will be used to approximate flow fields for varying structure designs. Empirical design procedures will then be developed from the resulting numerical-modeling database. The hybrid hydraulic modeling approach is detailed and results are presented from the CFD model calibration and validation to laboratory data.

INTRODUCTION

Hydraulic modeling has historically been grouped into the paradigms of either theoretical or empirical derivation. In essence, theoretical models are derived from conservation fundamentals while purely empirical methods rely on statistical methods to fit observed data. Due to the complex nature of fluid flow, the majority of theoretical models have a degree of associated empiricism; e.g. Manning *n*, viscosity, energy loss coefficients, which must be calibrated using observed hydraulic data. However, the core of the theoretical models is grounded in the Navier-Stokes equations. Empirical models used for hydraulics, such as stage-discharge relationships, scour-depth studies, rip-rap sizing, or stilling basin design, typically implement pertinent design parameters grouped into physically meaningful groups with numerical weights tailored to a collected dataset.

With increasing computational power and efficiency, theoretical models have evolved into robust, three-dimensional computational fluid dynamics (CFD) simulations. Such simulations have been shown effective in the evaluation of complex hydraulic conditions. While validated, the widespread use of CFD has not been realized in the current applied engineering and scientific realms. Theoretical models incorporating assumptions of the behavior of flow are widespread for practitioners, including HEC-RAS one-dimensional and two-dimensional code; however, the assumptions of such models significantly limit accuracy of results in complex flow environments where they are violated. In one-dimensional and two-dimensional theoretical flow simulations, assumptions break down when there are spatially rapidly-varied flow conditions containing a significant vertical flow component. Codified and widely implemented theoretical modeling codes such as HEC-RAS utilize empirical equations to account for the instances where assumptions break down, such as in the instance of a hydraulic jump or encountering an instream structure. The research presented in this proceeding illustrates a novel approach for hydraulic modeling methods by examining the feasibility of the combination of both theoretical and empirical models to develop design procedures for riverine structures. Proposed methods for the development of empirical design guidelines for transverse instream structures using data gathered from three-dimensional CFD simulations are expounded and detailed.

The Middle Rio Grande River between Cochiti Dam and Elephant Butte Reservoir in New Mexico has been the focus of extensive river restoration work since the upstream Cochiti Dam installation in 1975. The dam effectively disconnected the sediment continuity to the downstream reach, resulting in a geomorphic shift from a historically braiding channel to a slightly sinuous, incising system. The United States Bureau of Reclamation (Reclamation), as the responsible party for management of the river, launched an investigative study on the performance of transverse in-stream structures jointly with Colorado State University (CSU). Research performed by CSU and Reclamation has provided a wealth of physical model hydraulic data surrounding transverse in-stream structures and quantitative design guidelines for structure installation.

Transverse in-stream structures are a type of river-training structure that has the primary goal of halting bank migration in a meandering system. Structures extend from the outer-bank of the channel into the center, diverting flow from the outer-bank to the relocated channel thalweg at the structure tips. Nomenclature for transverse in-stream structures varies dependent upon the crest height and intended flow pattern. Bendway-weirs, spur-dikes, and bank-attached vanes are types of in-stream structures, planimetrically identical, yet different in their cross-sectional geometries and intended hydraulic effects. Planimetric and cross-section schematics of the three identified in-stream structures are provided in Figure 1, which expounds differences between structure classifications in the cross-sectional view. In a general hydraulic sense, bendway weirs redirect conveyance perpendicularly and over the top of the structure crests, spur-dikes shift flows around the structure tip, and bank-attached vanes combine both crest overtopping and shifted flow to redirect conveyance to the channel center.

Design recommendations for transverse in-stream structures are typically anecdotal and do not provide specifics of hydraulic performance based upon alteration of design parameters within recommended ranges. Examples of guidelines include NRCS (2005) for bank-attached vanes, Lagasse et al. (2009) for spur-dikes, and McCullah and Gray (2005) for bendway weirs. Scurlock et al. (2014) presented a quantitative model for estimation of normalized maximum and average velocities at various locations within a channel resulting from structure installations. The general mathematical model presented by Scurlock et al. (2014) is given as Equation 1.

$$MVR, AVR = a_1 + a_2 \left(A^*\right)^{a_3} \left(\frac{L_{ARC}}{T_W}\right)^{a_4} \left(\frac{R_C}{T_W}\right)^{a_5} \left(\frac{L_{W-PROJ}}{T_W}\right)^{a_6} \left(\frac{D_B}{D_B - \Delta z}\right)^{a_7} \left(\frac{2\theta}{\pi}\right)^{a_8} \qquad (1)$$

where:

MVR = maximum velocity ratio;

AVR = average velocity ratio;

A^* = percentage of baseline cross-sectional area blocked by structure;

$L_{W\text{-}PROJ}$ = projected length of structure into channel [L];

L_{ARC} = arc length between centerline of structures [L];

R_C = radius of curvature of channel bend centerline [L];

T_W = averaged top width of channel measured at baseline in bend [L];

D_B = averaged maximum cross-section baseline flow depth in bend [L];

Δz = elevation difference between water surface and structure crest at the tip [L];

θ = structure plan angle [radians];and

$a_1,\ldots,a_6$ = regression coefficients.

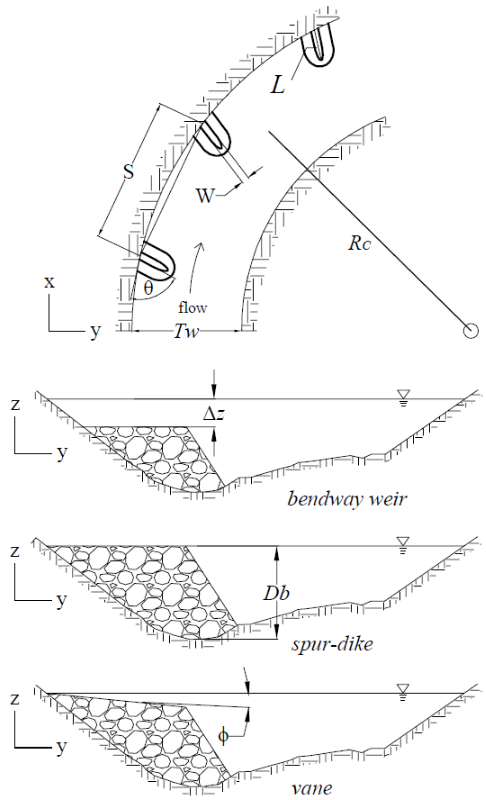

Figure 1 In-stream structure geometric parameter definitions

Heintz (2002), Darrow (2004), and Schmidt (2005) evaluated hydraulic performance of transverse in-stream structures in a physical model at CSU. Using physical model data collected from these studies from a 1:12 Froude scale model of two trapezoidal representations of the prototype Middle Rio Grande River, Equation 1 was optimized to predict normalized velocities at the outer-bank, centerline, and inner-bank of the channel for spur-dike and bank-attached vane installations. A total of 130 independent data points were used for regression analysis, representing a statistically large database for equation development.

Further research was conducted at CSU by Thornton et al. (2011), Scurlock et al. (2014a), and Scurlock et al. (2014b) on spur-dikes, bendway-weirs, and bank-attached vanes installed in a physical model representation of natural channel geometry. Survey data were obtained from two channel bends in the Middle Rio Grande and were constructed within the spatial constraints of the trapezoidal physical model. A total of four bendway-weir configurations, one spur-dike, and one bank-attached vane configuration were evaluated in the natural channel with comprehensive velocity fields realized through data collection with an acoustic Doppler velocimeter (ADV). Data collected in the natural topography physical model elucidated important hydraulic characteristics regarding transverse in-stream structures; however, evaluated configurations do not substantiate regression analysis using the model of Equation 1.

The primary goal of the current research is to investigate methods of compiling a large dataset for the bendway-weir structure type in attempt to develop quantitative design guidelines similar to the model of Equation 1. To facilitate this development, a dataset of significant size would be required, at least one order of magnitude higher than the four physically modeled natural-topography bendway-weir configurations. Physical modeling can be time consuming and resource intensive; factors which can inhibit the compilation of the requisite dataset for design guideline development. Utilizing CFD, it may be possible to take numerically simulated results from modeled bendway-weir configurations and compile a dataset large enough for empirical development. The feasibility of developing bendway-weir design guidelines using CFD is examined using calibration and validation procedures with the physical model data of Scurlock et al. (2014a) and Scurlock et al. (2014b). It is shown that CFD methodologies present a viable alternative to physical modeling or field data collection for the development of empirical equations, representing a hybrid approach to hydraulic modeling.

NUMERICAL MODEL SELECTION

A proven, reliable, and reproducible model was desired for the completion of numerical evaluation of laboratory data. A balance between desired accuracy and required computational resources must be achieved during numerical modeling. While large-eddy simulations and direct-numerical solutions outperform Reynolds-averaged Navier-Stokes (RANS) models in resolving turbulent and separating flows (Kang and Sotiropoulos, 2012; Constantinescu et al., 2011), the required computational times are substantially greater and unfeasible for the proposed research objectives. A numerical code using a RANS approach with an appropriate turbulence model was investigated. FLOW-3D is a commercial numerical package created by Flow Science which has been proven in open-channel flow applications and in-stream structures (Rodriguez, 2004; Abad et al., 2008; Plymesser, 2014; Kolden, 2013). The model incorporates a Fractional-area-volume-obstacle-representation (FAVOR) method for solid object interfaces and volume-of-fluid (VOF) method for free-surface fluid interfaces. FLOW-3D uses a rectilinear, orthogonal grid system in conjunction with FAVOR and VOF to rapidly develop discretized meshes around complex objects, which is preferable when investigating multiple geometries.

The RNG k-ε turbulence mode, an option incorporated into FLOW-3D model, was found to be an appropriate numerical method of representing overall hydraulic trends while accounting for smaller scales of turbulence than other RANS turbulence closure models. The RNG k-ε model presented by Yakhot et al. (1992) accounts for different scales of turbulent motion influencing the transport of k and ε and may improve RANS model resolution of rotating flows. All simulations were conducted using the RNG k-ε RANS model with the standard turbulent mixing length coefficient of 0.09. Pressure was solved for implicitly and momentum advection was set as a second-order monotonicity preserving explicit scheme akin to the PRIME method of Maliska and Raithby (1983).

Meshing proficiency, graphical display capabilities, ease of user interaction, numerical method flexibility, and application track record led to the selection of FLOW-3D as the model of choice to meet project objectives. With the numerical package selected and specific code aspects of the model determined, the spatial domain of the simulation was then defined in order for the mesh to be generated and the numerical code executed. Representations of the physical model channel and in-stream structures were created to serve as the boundaries for the numerical simulations.

GEOMETRY REPRESENTATION AND MODEL SETUP

Representation of a physical surface as a boundary in a numerical mesh requires surveyed data and a method for interpolation between known data points. The nature of the FAVOR model in FLOW-3D allows for rapid integration of new components to surfaces. This concept works well in the context of the current research. A baseline model was created as an individual surface and in-stream structure configurations were represented as independent surfaces and brought into the model separately.

The creation of the numerical domains utilized high-resolution LiDAR data of approximately 15 million individual points parsed to approximately 6 thousand points for surface generation. Parsed data were imported to AutoCAD Civil 3D and developed into a surface using a triangular-irregular network (TIN) which was then exported to FLOW-3D. The same process was followed with bendway-weir configurations. Figure 2 illustrates a schematic of the bendway-weir configurations installed with a topographic representation interpolated from LiDAR data. FLOW-3D requires that the flume outlet be oriented on an orthogonal axis. The model was rotated and linearly translated to ensure that the outlet was oriented parallel to x = 0 ft. Initial and boundary conditions within FLOW-3D are specified once the surface has been imported to the program. The developed baseline surface was added, the mass-flow outlet was designated, and the domain inlet was defined as a constant mass input. Constant mass input boundary conditions designate the full cross section with a uniform deliverance of volumetric flow and do not initially contain information regarding developed velocity profiles expected in a physical laboratory or field. This entrance effect was mitigated by extending the model input section approximately 10 channel widths upstream allowing uniform cross-section inflow to develop along the channel before it encountered the test area. Downstream boundary conditions were specified at water-surface elevations observed during laboratory testing. Figure 3 provides an example of the numerical model topography depicting the baseline model (in grey) along with a bendway-weir structure installation (in red).

MODEL CALIBRATION

Numerical models may be calibrated to specific applications through adjustment of spatial grid resolution, maximum permitted time step, surface roughness, turbulence parameters, mass inflow, fluid properties, initial conditions, and boundary conditions. In the case of modeling bendway-weirs using FLOW-3D, grid independence, time step, surface roughness, and mass inflow was adjusted during the calibration process. Initial grid, time step, and mass inflow calibrations were performed on the baseline model and then applied to the bendway-weir configurations. Approximately 3,000 ADV data were used for the baseline calibrations as reported in Scurlock et al. (2012). Total velocity measurement variability from instrumentation thresholds as reported from Scurlock et al. (2012) amounted to ± 4.5% of the measured value. Surface roughness was calibrated to baseline and structure installation configurations.

Numerical grid independence is an evaluation of the level of mesh resolution required for efficient and accurate representation of fluid dynamics throughout the solution domain. Both spatial precision and solution accuracy increase with finer grid resolution up to a threshold at which further reduction of grid size does not warrant computational expense. To prove grid independence, an original Cartesian mesh of size {x,y,z} = {1.5 ft, 1.5 ft, 0.5 ft} was evaluated then split by a factor of two for reevaluation. Global mean absolute difference was computed between the flow velocities for each mesh and a mean absolute percent difference (MAPD) tolerance was established at 5%. *MAPD* was calculated using the grid points from the coarser mesh as comparison locations and used the finer mesh for deviation normalization. Five grid sizes were evaluated and grid independence for the baseline model was established at a spacing {0.1875 ft, 0.1875 ft, 0.0625 ft}. The *MAPD* between the last grid iteration was 2.6% for depth-averaged velocity points and 3.7% for all velocity data, largely centered in small, localized areas at the flow boundaries.

Time-step discretization is another dimensional grid in addition to the spatial mesh which must be specified for numerical model execution. Time-step sizing is not as important as spatial grid resolution for data precision; a smaller time step will generally not provide more information than a larger one. However, the explicit nature of the momentum advection numerical scheme and Courant-Friedrichs-Lewy (CFL) limitations makes the time-step size fundamental for local numerical stability as well as for attenuation of global numerical oscillation perturbations (Courant et al., 1928). Time-step independence was proven by iterating maximum time step size. CFL limitations dictate that the time step must be small enough to reconcile all advective motion within a given grid cell, or that the speed at which the mass travels through a certain distance cannot exceed the numerical computation speed. As such, the time step was automatically reduced if CFL requirements were violated.

Surface roughness affects the frictional resistance on the flow and all hydraulic properties within the channel. Five roughness iterations were carried out to bracket the minimum deviation between numerical and physical results. Flow depths, 60% depth laboratory velocity locations, and the full set of laboratory velocity data collection locations were used for comparison between numerical and physical models. A surface roughness value of 0.07 ft produced the most accurate results at a precision level of ± 0.01 ft.

Using the calibrated mesh size, time step, and roughness value, a full-domain simulation was performed on the baseline model. It was observed that flow depths as well as velocities were greater than those recorded in the physical model, indicating that the volumetric flow rate in the numerical model was in excess of that in the physical model. The physical model surface was not impermeable and an amount of seepage was present during operation. A calibration adjustment of the numerical flow rate to 11.5 ft^3/s produced more accurate results and the discharge reduction of 0.5 ft^3/s, representing 4.7% of the bankfull flow rate, was applied to all subsequent numerical simulations.

The calibrated numerical baseline model was run at 11.5 ft^3/s until a steady-state condition was achieved. Numerical model data were extracted at the physical data-collection locations for comparison. Flow depths throughout the solution domain matched physical values well with mean-absolute deviation of 0.026 ft and *MAPD* of 3.78%. Velocity magnitudes at 60% flow depth were represented well with important regions of flow for the project objectives resolved. Regions of high and low flow at the respective outer-bank and inner-bank were nearly identical in both models and velocity magnitudes were of approximately the same. The *MAPD* was calculated at 11.67%, the median was 9.00%, and the standard deviation was 12.33%, indicating a strong right-skew. Approximately 55% of the data were represented with less than 10% difference and 84% of data were predicted with 20% difference or less.

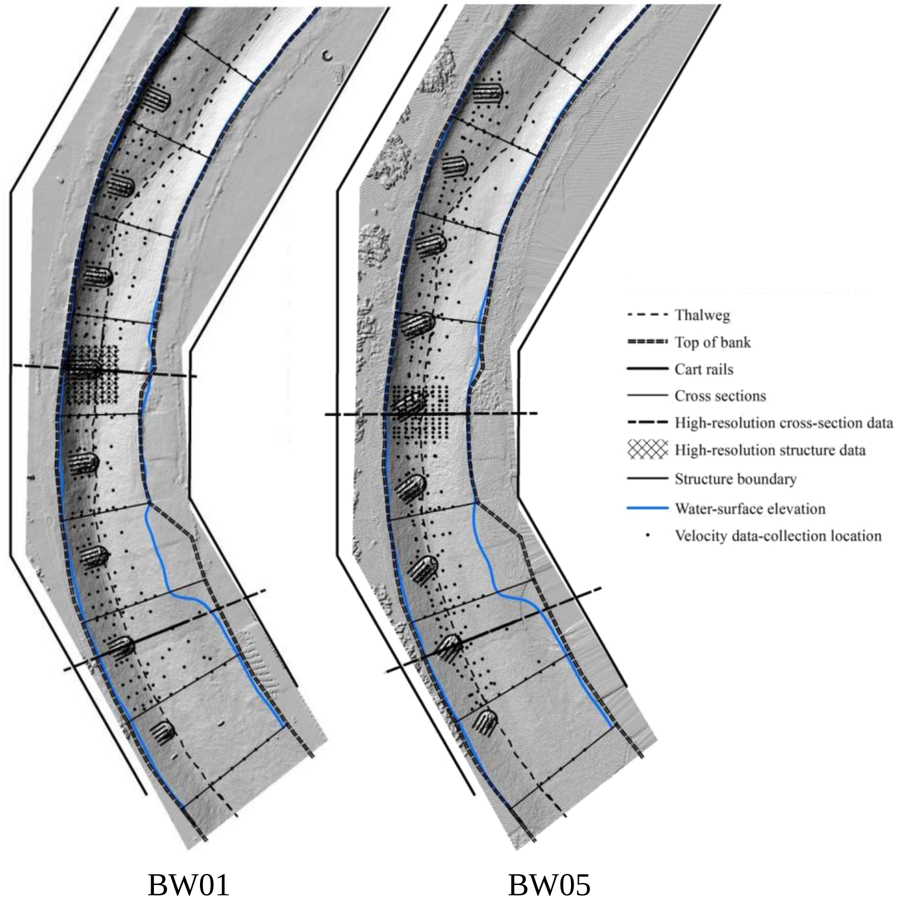

Figure 2 Bendway-weir configurations, flume schematic, and data-collection locations

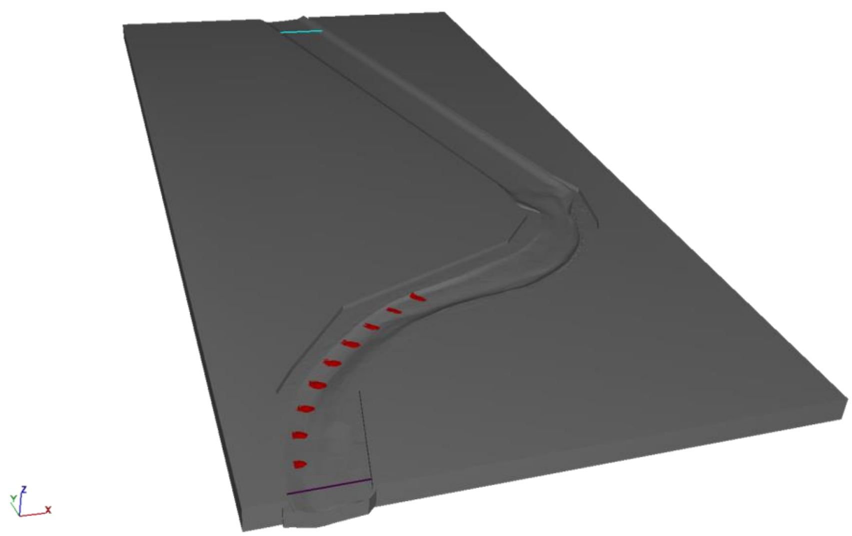

Figure 3 Geometric representation of baseline model (grey) and installed structures (red)

A calibration simulation at 11.5 ft^3/s was performed on the BW05 bendway-weir configuration from Scurlock et al. (2014b) with uniform surface roughness set at R = 0.07 ft and compared to the physical model dataset. Figure 3 illustrates the BW05 structure configuration and velocity data-collection locations. Roughness values for the structures were iterated and deviations between numerical and physical results increased with roughness greater than the bed surface. The structures were then set uniformly at 0.07 ft of surface roughness. Mean-average difference for flow depth was calculated as 0.04 ft and the *MAPD* was found to be 4.61%. Distributions of velocity in the physical and numerical model of the BW05 configuration are presented in Figure 4. The numerical model velocity distribution adheres closely to magnitudes at 60% flow depth in the physical model with shifted conveyance to the channel center, acceleration over structure crests, acceleration around structure tips, and outer-bank velocity increases captured. A notable area of accuracy of the numerical and physical model occurs at the shear layer between the reduced outer-bank velocity zone and increased channel-center velocity zone. The gradient zone was well resolved by the numerical model especially in the downstream regions of the channel. Discrepancies in the velocity distributions were mainly concentrated in the leeward areas of the bendway weirs and at the downstream extent of the model where a planimetric expansion occurred. The RNG κ-ε turbulence model used in the numerical model has limitations in application to highly turbulent flows with strong vorticity components and flow separation (Menter, 1994), such as observed in the areas of higher difference. Overall, 60% velocity magnitudes were predicted with a right-skewed difference distribution with a *MAPD* of 14.30%, median of 9.21%, and standard deviation of 16.53%. Seventy-nine percent of the data were predicted at better than 20% relative difference and 52% were predicted at 10% difference or less.

A high-density data collection cross section for BW05 transected behind the structure crest and passed through the zone of reduced outer-bank velocity. Approximately 300 ADV data collection points were spaced across this cross section. The distributions of velocity magnitudes between the physical and numerical models were similar as illustrated in Figure 5. A high velocity core was centered in the channel due to bendway-weir flow redirection, high velocity was noted above the structure crest, and low velocity zones were located behind the structure and at the inner-bank boundary. The gradient between the high velocity in the channel center and low velocity in the leeward shadow of the structure was represented well by the numerical model. The zone of high velocity in the channel center was slightly larger in size and of higher magnitude for the physical model. The majority of the channel cross-section was predicted with relatively low differences; 54% of the data were below 10% difference and 81% were below 20% difference. The calculated cross-sectional *MAPD* was 12.80%, median difference was 8.99%, and standard deviation was 13.67%

MODELVALIDATION

Numerical model validation consists of application of a calibrated numerical algorithm to an independent, yet similar simulation to which the numerical model was tailored to apply. If a numerical model performs well in describing a validation situation when no parameters are adjusted, then confidence is warranted for interpretation of further simulation extrapolations. The downstream minimum, BW01 configuration from Scurlock et al. (2014a) was used for the validation dataset for the bendway weir structure type. Figure 3 details the structure schematic and data-collection locations overlaid on a LiDAR topographic survey.

Numerical simulations for BW01 were performed at a steady 11.5 ft^3/s flow rate with calibrated parameters and boundary condition water-surface elevations observed in the physical model. Numerical data were extracted at the physical data-collection locations for comparable analysis. Flow depths for BW01 were predicted with an average difference of 0.034 ft and *MAPD* of 4.34%. Planimetric velocity magnitude distributions at 60% flow depth for the physical and numerical model are displayed in Figure 6. The distribution of velocity within the BW01 structure field was complex and varied rapidly between high and low velocity magnitudes. Velocity contours between models matched closely, with high velocities centered off the structure tips, structure-crest acceleration, outer-bank acceleration over the structure crests, and reduced velocity zones in the leeward zone of the structures. Transition gradient zones between regions of high and low velocity were similar between the two datasets and the numerical model represented the overall flow conditions of the physical model with high accuracy. The most flagrant discrepancies in the velocity magnitude distributions occurred at the leeward side of the fifth and sixth structures moving downstream in the structure configuration series. In this leeward zone, the numerical model simulated high velocities on the order of the outer-bank increased zone while the physical model contained data which indicated a region of reduced velocity. The distribution of difference percentage was strongly right-skewed with a *MAPD* of

44.62%, a median of 14.83%, and a standard deviation of 111.47%. The majority of data in the solution domain contained differences much lower than the regions near the fifth and sixth structure. Sixty-two percent of the data had relative difference of less than 20% and 35% of the data were below 10% relative difference. Median difference of 14.83% exceeded the BW05 calibration median difference of 9.21%.

An upstream high-resolution data cross-section transected the majority of the fourth structure crest in the BW01 configuration series. The cross-sectional topography and numerical and physical velocity magnitude distributions are presented in Figure 7. Velocity magnitude distributions between the two models share similar overall patterns. The largest velocities were centered over the tip of the structure crest, a low velocity region existed at the inner-bank, and acceleration over the structure crest was noted. Numerical results also produced higher velocities and less boundary interference near the channel thalweg than observed in the physical model. However, on a cross-sectional scale, results of the two models are visually equivalent. The majority of the cross section was well predicted, with a *MAPD* of 8.92%, median difference of 7.85%, and standard deviation of 6.02%. The median difference for the validation configuration was less than that of the cross-sectional calibration median difference of 8.99%.

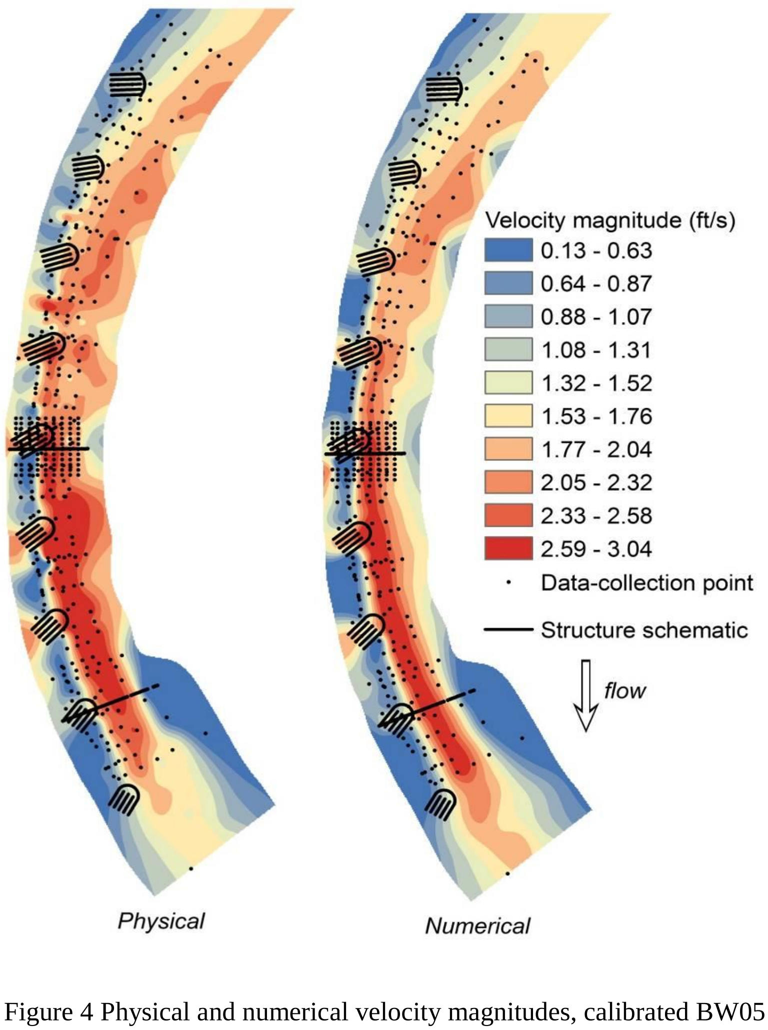

Figure 4 Physical and numerical velocity magnitudes, calibrated BW05

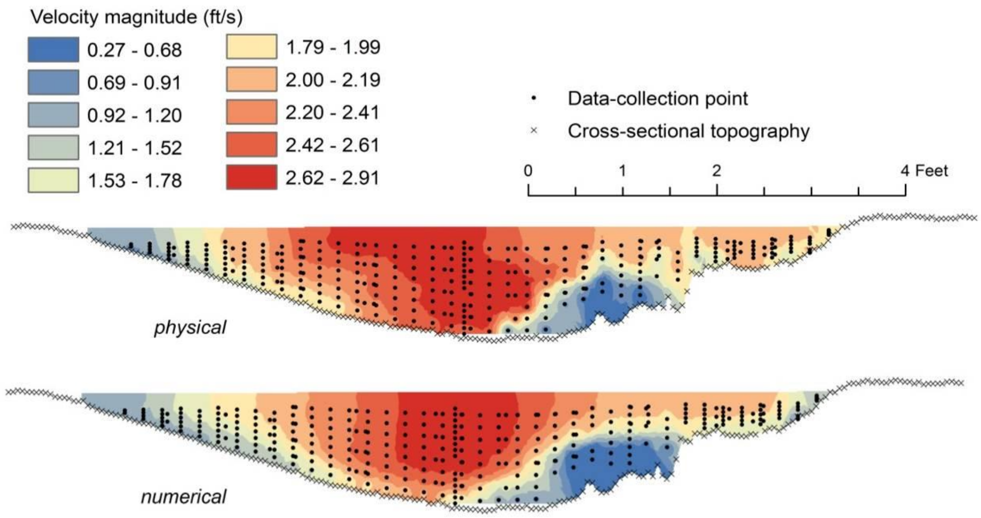

Figure 5 Velocity magnitude distributions, BW05XSA; downstream perspective

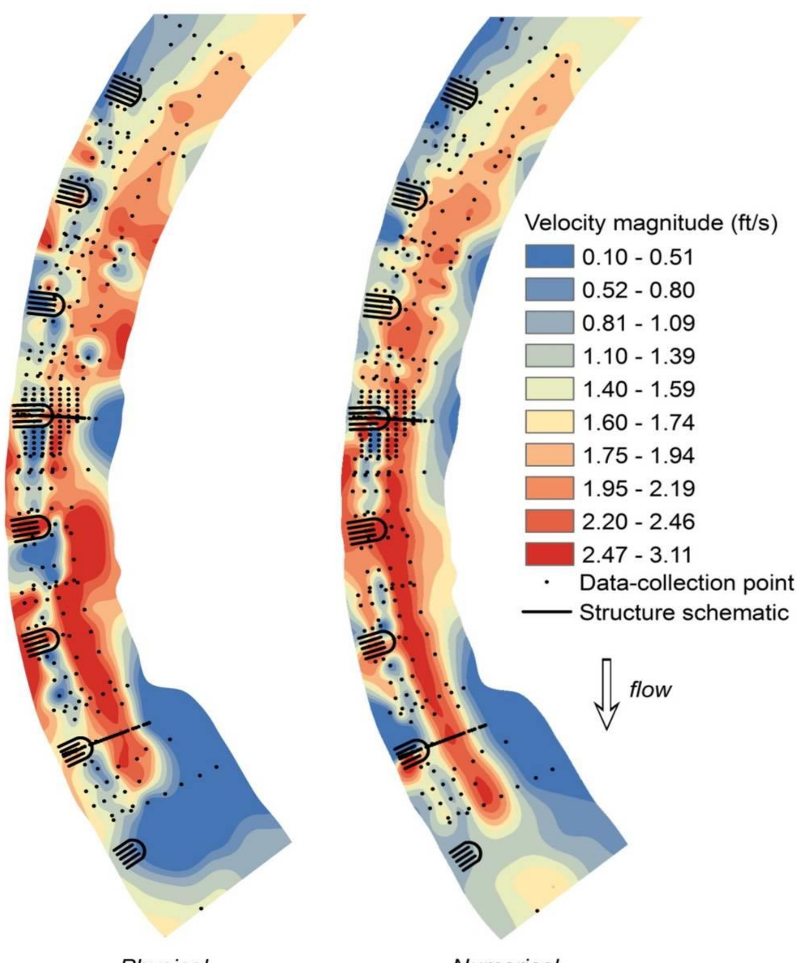

Figure 6 Velocity-magnitude distribution, BW01 bendway-weir validation simulation

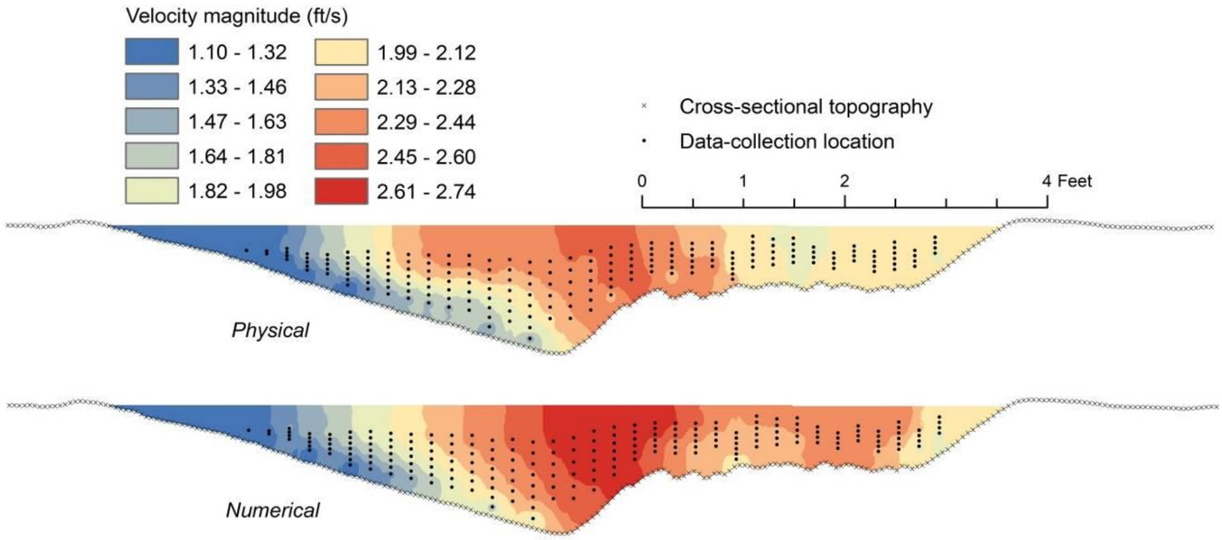

Figure 7 BW01XSA velocity magnitude distribution, bendway-weir validation simulation

SUMMARY AND CONCLUSIONS

Approximation of open-channel hydraulics may be achieved through the theoretical or empirical solutions. Prediction of flow conditions using theoretically grounded CFD methods has been proven to be accurate for in-stream structure hydraulics; however, widespread implementation of three-dimensional numerical modeling has yet to be fully integrated into typical engineering design. Designers typically employ empirical equations to account for scenarios in which typical one-dimensional and two-dimensional models are inaccurate. This study investigated the feasibility of using CFD to compile a large enough database for the development of an empirical model from numerical results, representing a hybrid modeling approach.

A CFD model was chosen and physical models of bendway weirs in a meandering river channel were represented numerically. Grid independence, maximum time-step independence, surface roughness, and seepage losses were calibrated using baseline flow data and one bendway-weir configuration and then validated using a separate bendway-weir configuration. Pursuant to the goals of empirical model development, the velocity magnitudes between the observed physical model data and predicted numerical results were compared. It was found that the numerical model represented observed velocity trends well and the general flow patterns were captured. Median percent differences in velocity magnitudes of comparable datasets were on the order of 10% and right-skewed, indicating a global representation of flow patterns with localized regions where results did not match.

Calibration and validation results accentuate the potential of CFD as a method of compiling a large enough dataset for empirical model development. A series of structure configurations at various geometry parameters will be created and placed within the baseline numerical model and simulations will be performed. Results of this process will provide substantial information about the effects of bendway-weir geometries on resulting flow fields and aid in the development of structure design procedures.

REFERENCES

Abad, J.D., Rhoads, B.L., Guneralp, I., and Garcia, M.H. (2008). "Flow structure at different stages in a meander-bend with bendway weirs." *Journal of Hydraulic Engineering.* 134:8. 1052-1063.

Constantinescu, G., Koken, M., and Zeng, J. (2011). "The structure of turbulent flow in an open channel bend of strong curvature with deformed bed: Insight provided by detached eddy simulation." *Water Resources Research.* 47, 1-17.

Courant, R.; Friedrichs, K.; Lewy, H. (1928), "Über die partiellen Differenzengleichungen der mathematischen Physik", Mathematische Annalen (in German) 100 (1): 32–74, Bibcode:1928MatAn.100..32C, doi:10.1007/BF01448839, JFM 54.0486.01, MR 1512478

Darrow, J.D. (2004). "Effects of bendway weir characteristics on resulting flow conditions." M. S. Thesis. Colorado State University, Department of Civil Engineering. Fort Collins, Colorado.

Heintz, M.L. (2002). "Investigation of bendway-weir spacing." M. S. Thesis. Colorado State University, Department of Civil Engineering. Fort Collins, Colorado.

Kang, S. and Sotiropoulos, F. (2012). "Numerical modeling of 3D turbulent free surface flow in natural waterways." *Adv. in Water Resources,* 40 (2012) 23-36.

Kolden, E. (2013). "Modeling in a three-dimensional world: Whitewater park hydraulics and their impact on aquatic habitat in Colorado." M. S. Thesis. Colorado State University, Department of Civil Engineering. Fort Collins, Colorado.

Lagasse, P.F., Clopper, P.E., Pagan-Ortiz, J.E., Zevenbergen, L.W., Arneson, L.A., Schall, J.D., and Girard, L.G. (2009). "Bridge scour and stream instability countermeasures: experience, selection, and design guidance." 3rd ed. HEC-23. Federal Highway Administration. FHWA-NHI-09-111.

Maliska, C.R. and Raithby, G.D. (1983). "Calculating 3-D fluid flows using non-orthogonal grid." *Proc. Third Intl. Conf. on Numerical Methods in Laminar and Turbulent Flows*, 656-666.

McCullah, J.A. and Gray, D., 2005, "Environmentally Sensitive Channel- and Bank-Protection Measures," NCHRP Report 544, Transportation Research Board, National Academies of Science, Washington, D.C.

Menter, F.R. (1994). "Two-equation eddy viscosity turbulence models for engineering applications." *AIAA Journal* 32(8). 1598-1605

Natural Resources Conservation Service (2005). "Design of stream barbs." *Tech. note 23.* Department of Agriculture. Portland, OR.

Plymesser, K.E. (2014). "Modeling fish passage and energy expenditure for American Shad in a steppass fishway using a computation fluid dynamics model." Ph.D Dissertation.Montana State University, Dept. of Civil Engineering.

Rodriguez, J.F., Bombardelli, F.A., Garcia, M.H., Frothingham, K.M., Rhoads, B.L., Abad, J.D. (2004). "High-resolution numerical simulation of flow through a highly sinuous river reach." *Water Resources Management,* 18: 177-199.

Schmidt, P.G. (2005). "Effects of bendway weir field geometry characteristics on channel flow conditions." M. S. Thesis. Colorado State University, Department of Civil Engineering. Fort Collins, Colorado.

Scurlock, S.M., Cox, A.L., Baird, D.C., Thornton, C.I., Parker, T.R., and Abt, S.R. (2012) "Middle Rio Grande Physical Modeling - Native topography: Construction and evaluation of baseline hydraulic conditions." Colorado State University, Department of Civil Engineering, Fort Collins, CO.

Scurlock, S.M., Thornton, C., Baird, D., and Abt, S. (2014). "Quantification of transverse in-stream structure hydraulics." *Journal of Hydraulic Engineering*. DOI: 10.1061/(ASCE)HY.1943-7900.0000952.

Scurlock, S.M., Thornton, C.I., and Abt, S.R. (2014a). "Middle Rio Grande physical modeling – Evaluation of bendway-weir structures with the native-topography channel." Technical report prepared for the United States Bureau of Reclamation, Albuquerque Area Office. October, 2014.

Scurlock, S.M., Thornton, C.I., and Abt, S.R. (2014b). "Middle Rio Grande physical modeling – Reevaluations and comparison of instream structures within the native-topography channel." Technical report prepared for the United States Bureau of Reclamation, Albuquerque Area Office. In Peer Review.

Thornton, C.I., Cox, A.L, Ursic, M.E., and Youngblood, N.A. (2011). "Data report for completed bendway-weir configurations within the native topography model." Colorado State University, Department of Civil and Enviromental Engineering. Fort Collins, CO.

Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. & Speziale, C.G. (1992), "Development of turbulence models for shear flows by a double expansion technique." *Physics of Fluids A*, 4(7): 1510-1520.