Jun 28th, 4:00 PM - 6:00 PM

Investigating Supercritical Flows in Curved Open Channels with Three Dimensional Numerical Modeling

Kortney Brown
kbrown@schnabel-eng.com

Brian Crookston
Schnabel, bcrookston@gmail.com

Recommended Citation

Brown, K., Crookston, B. (2016). Investigating Supercritical Flows in Curved Open Channels with Three Dimensional Numerical Modeling. In B. Crookston & B. Tullis (Eds.), Hydraulic Structures and Water System Management. 6th IAHR International Symposium on Hydraulic Structures, Portland, OR, 27-30 June (pp. 230-239). doi:10.15142/T3580628160853 (ISBN 978-1-884575-75-4).

6th International Symposium on Hydraulic Structures Portland, Oregon, USA, 27-30 June 2016
Hydraulic Structures and Water System Management
ISBN 978-1-884575-75-4 DOI: 10.15142/T3580628160853

Investigating Supercritical Flows in Curved Open Channels with Three Dimensional Numerical Modeling

W.K. Brown, PE[1] and B.M. Crookston, PhD, PE[1]
[1]Schnabel Engineering
West Chester, PA 19382
USA
E-mail: KBrown@Schnabel-eng.com

ABSTRACT

The flow patterns observed in supercritical flows at bends in open channels are encountered in spillways, canals, and drainage works – this complex flow condition has been investigated more than 80 years ago, first with hydraulic models and more recently using numerical models. This paper presents a numerical investigation using a selection of experimental data from Dr. A. T. Ippen and a commercially available 3-dimensional CFD solver. A comparison of the numerical results to the physical data is presented, highlighting the ability of the numerical models to reproduce these complex water surface profiles, including the magnitude and location of standing waves. Modeling efforts were of a single fluid and used RNG and LES turbulence models. Furthermore, the observations, findings, and conclusions of this paper are discussed as they relate to open channel design.

Keywords: *Supercritical, Bends, Computational Fluid Dynamics, CFD, Numerical, Modelling*

1. INTRODUCTION

Complex flow behaviors in open channels have been of interest to researchers and practitioners for well over 100 years. One such behavior encountered in spillway and channel applications is the oblique wave patterns occurring in supercritical flow around channel bends, such as is presented in Figure 1 (Henderson 1966). The majority of the empirical relationships commonly used to address this complex behavior resulted from studies performed by researchers such as Ippen (1936), Knapp and Ippen (1938), Von Kármán (1938), Shurkey (1950), Ippen (1951), Knapp (1951), Poggi (1956), Sananes and Acatay (1962), Rakotoarivelo and Sananes (1967), Marchi (1988), and Reinauer and Hager (1997). The United States Army Corps of Engineers summarizes some of these works in their EM 1110-2-1601 (Hydraulic Design of Flood Control Structures, 1991). While experimental results and various calculations (empirical, method of characteristics, etc.) are commonly used to predict water surface profiles, wave amplitudes, and spatial locations around bends, uncertainties can exist when mapping standing waves or computing these parameters for specific channel geometries.

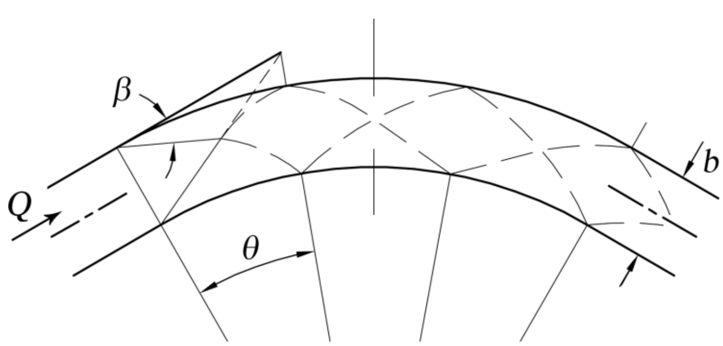

Figure 1. Supercritical Flow in Bends.

Physical hydraulic engineering models have been the preferred tool by practitioners for the design of hydraulic structures and channels. Published literature indicates that numerical modelers began studying flow in curved chutes in the late 1960s and early 1970s. Krause (1970) solved 2D shallow water equations for supercritical flow, although his results appear to be indefinite. As computational fluid dynamic (CFD) solvers evolve and computational power advances, so does the ease with which timely solutions are obtained for complex numerical models. By the early 1990s, works utilizing the 2D shallow water equations were published (Berger and Stockstill (1995), Causon et al. (1999), Elliot and Chaudhry (1992)), and some focused on vertical 2D models (Montes (1994), Valiani and Caleffi (2005), Bhajantri et al. (2007), Ghaeini-Hessaroeyeh et al. (2011), Montazeri-Namin et al. (2012)). The findings of Valiani and Caleffi (2005) indicate that while their shallow water equation (SWE) numerical solutions predict certain characteristics of the flow around bends reasonably well with a low Froude number, the maximum water surface level (i.e., wave extrema) is underestimated. Ghaeini-Hessaroeyeh et al. (2011) and Montazeri-Nemin et al. (2012) compliment Valiani and Caleffi (2005), where Montazeri-Nemin et al. (2012) conclude that the shallow water equations (SWE) using the Godunove type methods (same method as Valiani and Caleffi) can be used in "satisfactorily predicting the reflection pattern of shock waves and the wavelengths along the outer chute wall" yet "the 2D model based on the SWE is however limited to small relative curvature."

For many hydraulic structure applications, CFD models have proven to be a valuable design, analysis, and validation tool, securing a place in the engineer's toolbox with the ability to execute global modification quickly, provide extensive information throughout the flow field, supply qualitative and incremental information and quantities, and compliment physical hydraulic modeling efforts. Of course, it is preferred that a numerical modeler have a thorough understanding of both the actual hydraulics and the chosen numerical methods so as not to be deceived by virtual hydraulic quirks, shortcomings, and erroneous results (Knight 2013).

It is the intent of this paper to explore the complex hydraulics occurring in supercritical flow around bends by juxtaposing the earliest research efforts by Ippen (1936) with 3D numerical simulations via the CFD solver FLOW-3D developed by Flow Science. Results are followed by conclusions and discussion regarding the application of the CFD tool.

2. NUMERICAL MODEL DEVELOPMENT

2.1. Digital Models Based on Ippen (1936)

Ippen (1936) conducted a number of tests for three channel slopes (10%, 3.5%, and 1.5%), three radii of curvature (12.2 m, 6.1 m, and 3.0 m), and two angles of deviation (22.5° and 45°). Additionally, Ippen (1936) investigated means of reducing the magnitude of standing waves by installing longitudinal vanes along the channel bends. Each geometric configuration was tested for a range of discharges, resulting in 93 runs with various corresponding water surface contour maps, water surface profiles, and velocity distributions, though data for some runs appears to be missing. Of the data published, many runs have very limited information available, such as an outside bend profile. A portion of this dataset was selected for numerical modeling and is summarized in Table 1. These selections were based upon water surface data available in Ippen (1936) (for comparison) and also the range of discharges for a given bend geometry.

Development of the digital three dimensional models was accomplished with commercially available CAD software. Two assumptions were required as not all details of the experimental setup were documented by Ippen (1936). First, the model assumed that the radii cited by Ippen (1936) were in the plan view and not in the plane of the slope. Additionally, it was assumed that the flume floor perpendicular to the primary axis was horizontal, that is the floor was horizontal in any given channel cross section. Numerical point gauges were located in alignment with Ippen (1936) data points for experiments summarized in Table 1, as well as at 0.3048 m (1ft) intervals along the chute to provide additional resolution of the computed water surface profile.

Table 1. Investigation Summary

Run	Slope	Discharge	Channel Width (b)	Radius of Curvature (R_a)	Relative Radii of Curvature (p_a=b/R_a)	Angle of Deviation
13	0.0995	57.68 l/s	0.3048 m	3.048 m	1/10	45°
14	0.0995	47.86 l/s	0.3048 m	3.048 m	1/10	45°
15	0.0995	28.74 l/s	0.3048 m	3.048 m	1/10	45°
17	0.0995	9.57 l/s	0.3048 m	3.048 m	1/10	45°
38	0.0345	21.72 l/s	0.3048 m	3.048 m	1/10	45°
39	0.0345	56.07 l/s	0.3048 m	3.048 m	1/10	45°
41	0.0345	33.33 l/s	0.3048 m	3.048 m	1/10	45°
42	0.0345	42.48 l/s	0.3048 m	3.048 m	1/10	45°

2.2. CFD Model

Numerical simulations, as summarized in Table 1, were performed using the commercially available CFD solver FLOW-3D version 11.1. This solver uses a finite-volume approximation to solve continuity and momentum equations in three dimensions. Only one fluid was simulated. The domain was subdivided with a hexahedral single mesh. The 3D digital models representative of Ippen's setups were imported into the CFD software and embedded onto the mesh using the FAVOR method, which allows portions of a cell to be occupied by a solid or obstacle.

Two turbulent closure methods were evaluated: the Re-Normalization Group (RNG) turbulence model and the Large Eddy Simulation (LES) turbulence model, which have various method characteristics and limitations but were viewed as relevant to this case. Turbulence lengths were dynamically computed, and the Split Lagrangian method was used for volume-of-fluid advection and the second order monotonicity preserving solution for momentum advection. Viscous stresses were solved explicitly and pressures were solved implicitly.

Boundary conditions were defined according to the physical model tests using a defined flow rate as the upstream boundary, a free-overfall at the downstream boundary, and solid walls for the remaining boundaries except Z_{max}, which was defined as atmospheric pressure. All simulations were modeled with temporally fixed boundaries with the objective of reaching a steady-state solution of the system. Three cell sizes were examined for mesh convergence in Run 13 and solutions agreement – 30.5 mm, 15.2mm, and 7.6 mm; the results indicate that solution convergence was obtained with minor differences between the 15.2 and 7.6-mm meshes. Two cell sizes were examined for mesh convergence in Run 39 and solution agreement—30.5 mm and 15.2 mm. At minimum, a third additional refinement is needed to establish a mesh-independent solution for this run; however, this has not been performed to date due to time constraints. Figure 2 presents sample water surface profile (inside and outside bend) solution of the RNG model for Run 1. Figure 3 presents a sample water surface profile (outside bend) solution of the RNG and LES models for Run 39, and. Both figures include the experimental data reported by Ippen (1936).

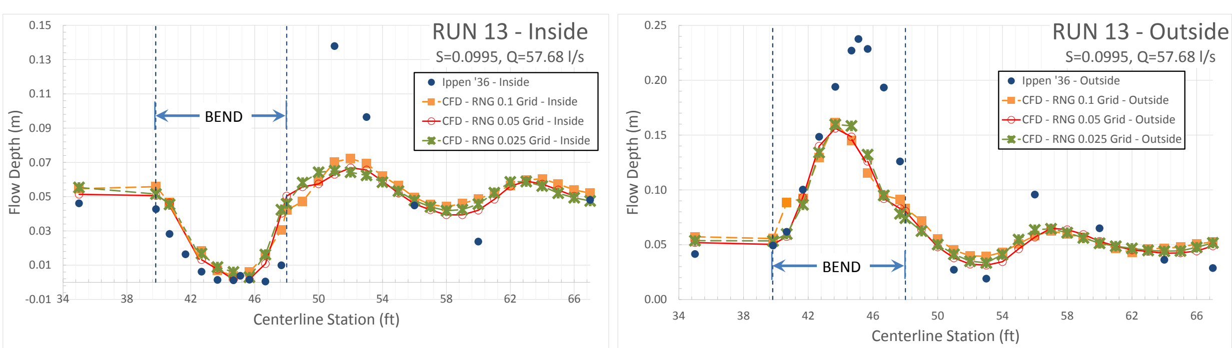

Figure 2. Run 13 Water Surface Profiles (inside and outside bend) for Multiple Mesh Sizes

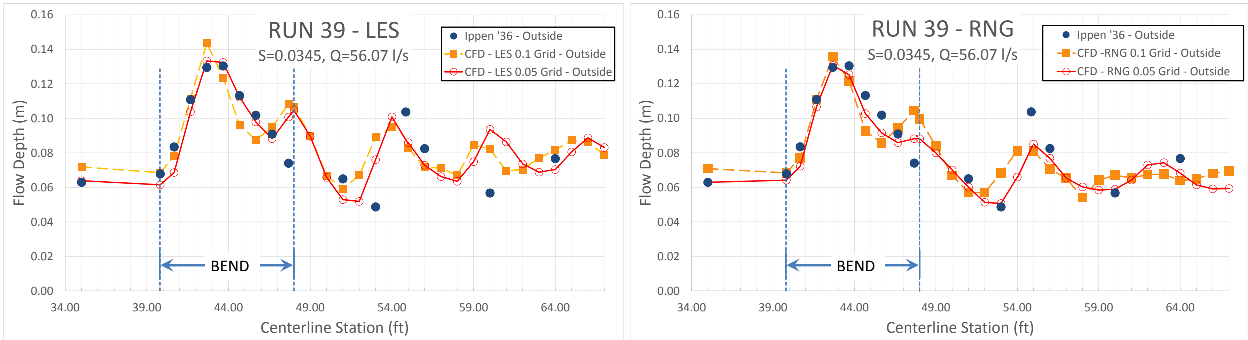

Figure 3. Run 39 Water Surface Profiles (outside bend) for Multiple Mesh Sizes

3. RESULTS

Water surface profiles for the runs identified in Table 1 are presented in Figures 4 through 11. Final numerical solutions from the RNG and LES models are plotted with the experimental results reported by Ippen (1936), with the abscissa maintaining the flume stationing during the experimental investigations. Please note that no profiles are provided for simulations where no physical data was reported in Ippen (1936).

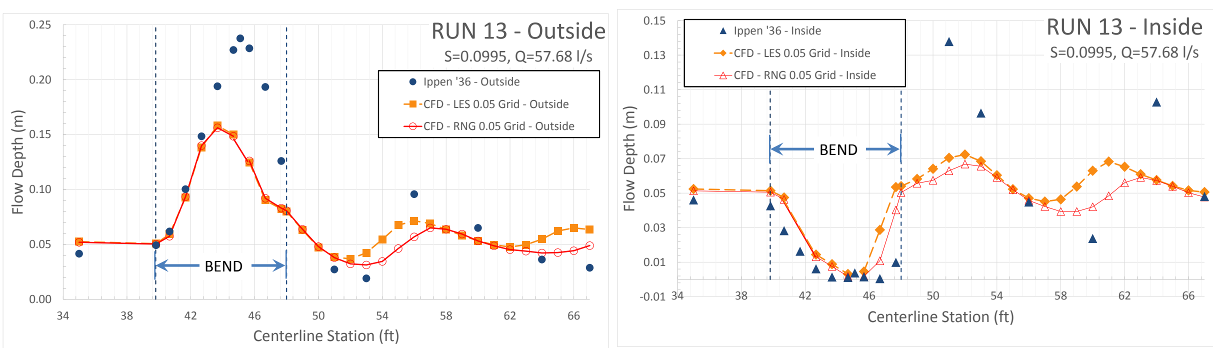

Figure 4. Run 13 CFD and Physical Model Bend Inside and Outside Profile Comparisons.

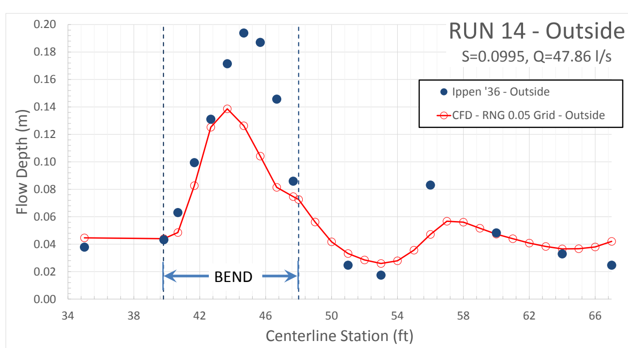

Figure 5. Run 14 CFD and Physical Model Bend Outside Profile Comparison.

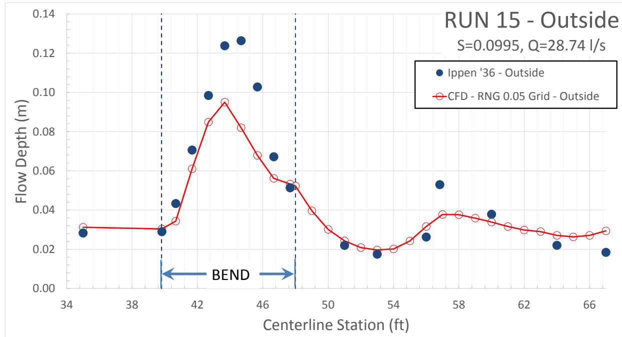

Figure 6. Run 15 CFD and Physical Model Bend Outside Profile Comparison.

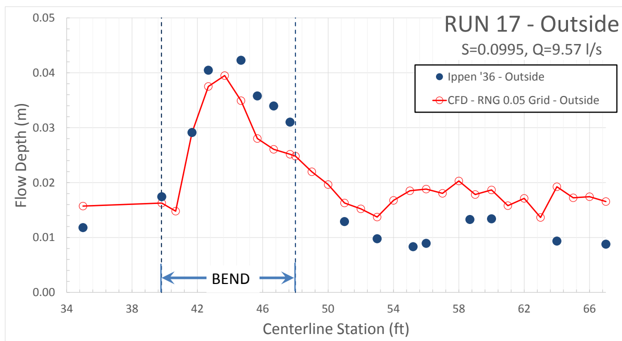

Figure 7. Run 17 CFD and Physical Model Bend Outside Profile Comparisons.

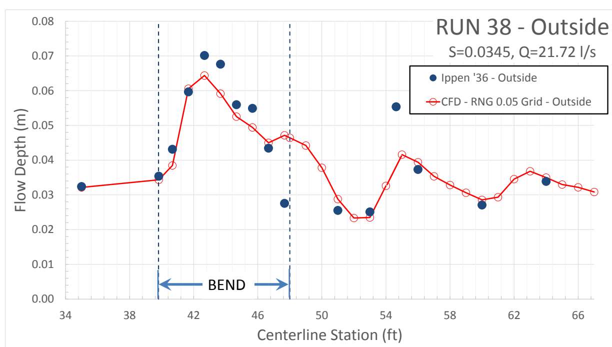

Figure 8. Run 38 CFD and Physical Model Bend Outside Profile Comparisons.

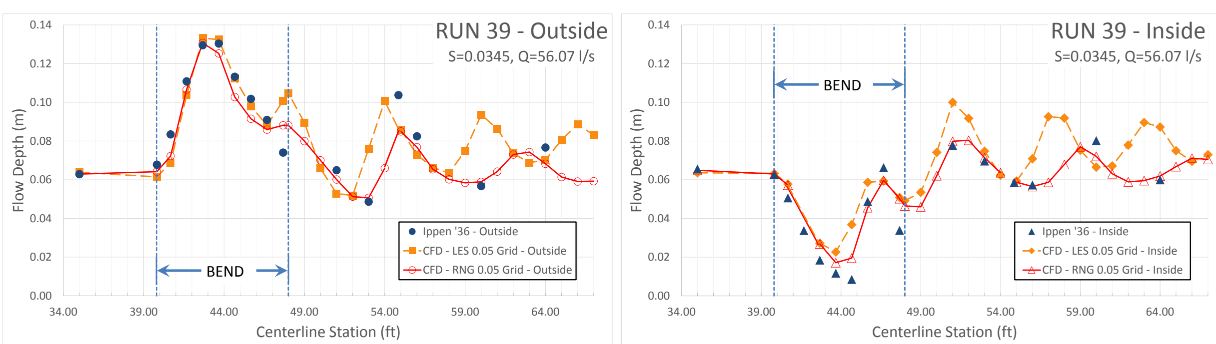

Figure 9. Run 39 CFD and Physical Model Bend Inside and Outside Profile Comparisons.

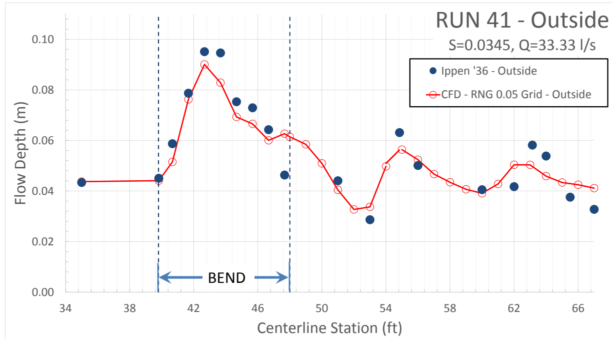

Figure 10. Run 41 CFD and Physical Model Bend Outside Profile Comparison.

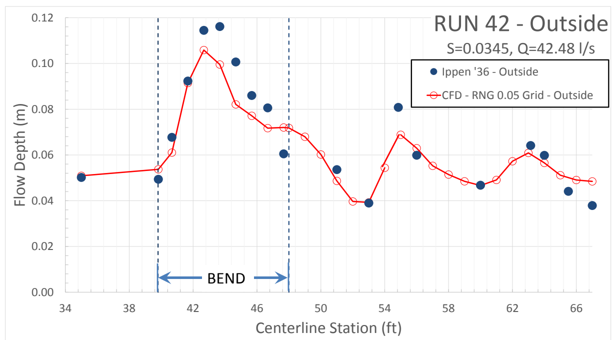

Figure 11. Run 42 CFD and Physical Model Bend Outside Profile Comparison.

Results indicate that the numerical models have good agreement along the outside wall within the bend for each run. No significant differentiator is observed between the LES and RNG turbulence models regarding the water surface along the outside of the channel bend except that, for the milder slope (Runs 38-42), the LES model predicted one additional standing wave in Run 39. In general, the LES model has fairly good agreement with the RNG model; however, the prediction of standing waves in the channel downstream of the bend appears to be phase shifted with a slightly lower frequency.

The numerical results compare most favorably with the experimental results of Runs 38-42, with an average deviation generally within +13% to -12% for Run 38, +5% to -10% for Run 39, +25% to -12% for Run 41, and +20% to -20% for Run 42, depending on the location of the wave in the channel bend. Figure 12 presents plots of the deviation of the RNG models from the experimental data of Ippen (1936) for each simulation, with results summarized in Fig. 13.

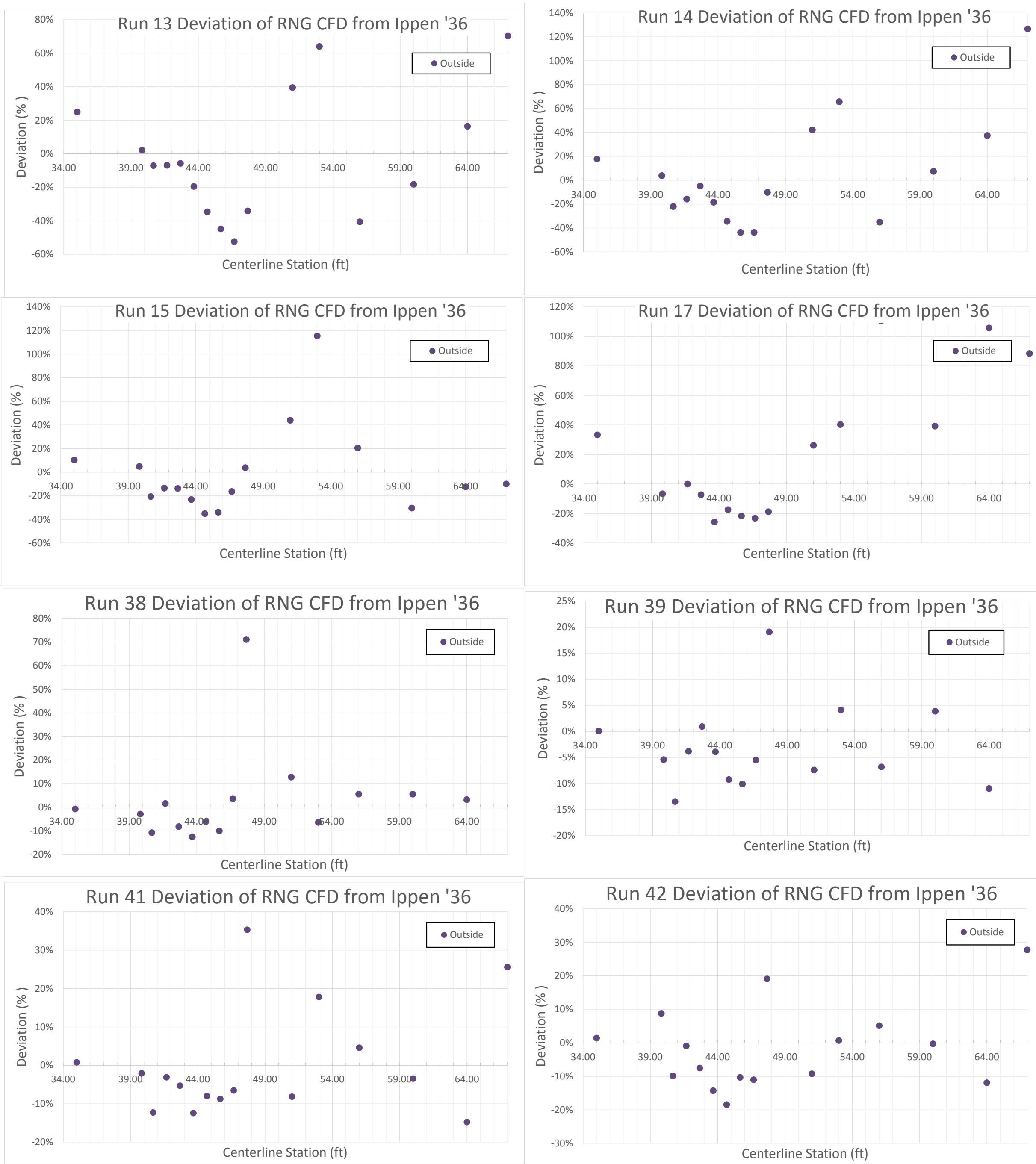

Figure 12. Deviation Plots - Outside of Bend.

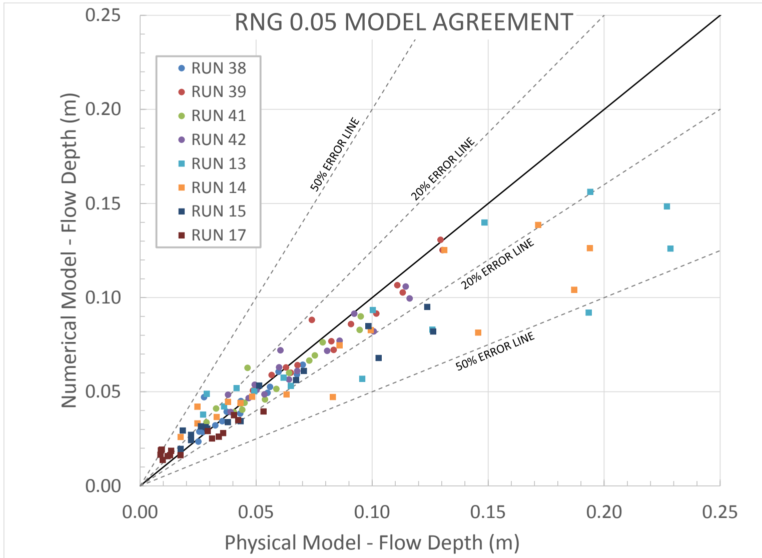

Figure 13. CFD and Ippen (1936) Model Agreement - Outside of Bend.

For a steeper slope and higher Froude number, the general trend of the water surface profile and waves are estimated numerically but differences arise for peak wave magnitudes; better agreement exists along the outside bend (often of more interest in design) with some disagreement along the inside channel bend. Generally, maximum wave heights appear to be underestimated around the channel bends when compared to Ippen (1936).

4. INTERPRETATION OF CFD RESULTS

Flow through a channel is governed by conservation of energy, momentum, and mass. These principles are of necessity approximated numerically by CFD solvers. Therefore, users should be aware of numerical model performance and limitations, including derivations of basic principles of fluid mechanics, closure methods (i.e., numerical methods, algorithms, etc.), coefficients, and any other details applicable to the situation of interest.

As discussed, relatively good agreement (see Figure 13) was generally found between the numerical results and the physical measurements of Ippen (1936). However, there were instances where the numerical point estimations for flow depth were in disagreement by more than 50%, particularly along the inside of the bend (Figures 4 and 9) and downstream of the bend. It is assumed that a small fraction of this error could be attributed to experimental uncertainties, such as measurement accuracy, technique, and temporal variations. Possible sources of numerical error could come from the aforementioned assumptions and approximations of the solver code related to estimation and tracking of the free surface (i.e., free boundary) that, in this case, is quite complex; Volume of Fluid (VOF) technique implemented in the selected solver code; changes in the pressure field; and shear forces. Therefore, it is good practice and strongly recommended that numerical modelers verify the computed results with experimental data to facilitate interpretation of the numerical results for design purposes. An appropriate physical model should be performed when determined to be a beneficial risk-reduction measure for a given project with its corresponding project goals and safety standards.

5. SUMMARY AND RECOMMENDATIONS

The results of this exploration indicate that it is possible to numerically simulate the complex flow patterns created by supercritical flows passing through a bend. In the case of a chute with S=0.0345 and R_a=1/10 (Fr=3.6), the numerical results were generally within approximately 10% to 20% of experimental measurements of Ippen (1936) and considered useful to designers. However, for a steeper slope, S=0.0995 and R_a=1/10 (Fr=6.6), there was general agreement between the physical and numerical results, but the simulations were not able to replicate the wave magnitudes or peak wave heights within an acceptable level of accuracy. Agreement was observed to decrease with increasing slope and discharge.

Hydraulic optimizations and modeling efforts are leveraged to meet project goals (safety, design, operation, economics, etc.) (Crookston et al 2013). Also, every project does not necessarily justify a quality physical hydraulic model to facilitate an efficient design. For smaller projects, a more conservative design may be preferable. It is also understood that numerical modeling is being used with increasing frequency both alone and in concert with empirical and/or physical modelling. CFD modelling can also be of great value in assessing a range of layouts and channel geometries for supercritical flow through a bend to arrive at one or more design concepts for physical modelling.

Based upon published literature and this investigative effort, it is recommended that a numerical model examining superelevation and standing waves in a supercritical curved channel be calibrated, ideally with experimental results from scaled hydraulic models. Calibration efforts should recognize the experimental accuracies of the physical model data. Quality data sets can also be a valuable source of information in calibration efforts. Until computational advances provide higher levels of consistency with physical model findings, caution should be exercised when interpreting numerical results, as the accuracy of findings will be uncertain.

6. ACKNOWLEDGEMENTS

This investigation was supported by Schnabel Engineering.

7. REFERENCES

Berger, R.C., and Stockstill, R.L. (1995). “Finite-Element Model for High-Velocity Channels.” *Journal of Hydraulic Engineering,* 121(10), 710-716.

Bhajantri, M.R., et al. (2007). “Numerical modelling of turbulent flow through spillway with gated operation.” *International Journal for Numerical Methods in Engineering,* 72, 221-243.

Brown, W.K., et al. (2012). “Revisiting Spillway Discharge Coefficients for Several Weir Shapes.” *Proc., 32nd Annual USSD Conference: Innovative Dam and Levee Design and Construction for Sustainable Water Management,* USSD, New Orleans, La., 1317-1334.

Causon, D.M., et al. (1999). “Advances in Calculation Methods for Supercritical Flow in Spillway Channels.” *Journal of Hydraulic Engineering,* 125(10), 1039-1050.

Crookston, B.M., Paxson, G.S., and Campbell, D.B. (2013). “Effective Spillways: Harmonizing labyrinth weir hydraulic efficiency and project requirements.” in Labyrinth and Piano Key Weirs II – PKW 2013. S. Erpicum, F. Laugier, M. Pfister, M. Pirotton, G Cicéro, A. Schleiss Editors. CRC Press, London, UK.

Elliot, R.C., and Chaudhry, M.H. (1992). “A wave propagation model for two-dimensional dam-break flows.” *Journal of Hydraulic Research,* 30(4), 467-483.

Ellis, J. (1985a). “Numerical analysis of Kielder dam spillway.” *Journal of the Institution of Water Engineers,* 39, 254-270.

Ellis, J. (1985b). “Numerical modelling of spillway flows.” *Proceedings of the 2nd International Conference on the Hydraulics of Floods and Flood Control,* BHRA, Cambridge, England, B(1), 73-90.

Ghaeini-Hessaroeyeh, M., et al. (2011). “Numerical modelling of supercritical flow in rectangular chute bends”. *Journal of Hydraulic Research,* 49(5), 685-688.

Ippen, A.T. (1936). *An analytical and experimental study of high velocity flow in curved sections of open channels.* Dissertation (Ph.D.), California Polytechnic University, Ca.

Ippen, A.T., and Knapp, R.T. (1936). “A study of high-velocity flow in curved channels of rectangular cross-section.” *Transaction, American Geophysical Union,* 17(2), 516-521.

Ippen, A.T. (1951). “Mechanics of Supercritical Flow.” *Transactions, ASCE, 116,* 268-295.

Knapp, R.T. (1951). “Design of Channel Curves for Supercritical Flow.” *Transaction, ASCE,* 116, 1318-1347.

Krause, E., and Hirchel, E.H. (1970). “Exact Numerical Solutions for Three-Dimensional Boundary Layers.” *Second International Conference on Numerical Methods in Fluid Dynamics,* University of California, Berkeley, Ca.

Marchi, E. (1988). “Correnti veloci in curve a 90° molto strete.” *Idrotecnica* 15(6), 439-455.
Montazeri-Namin, M., et al. (2012). “3D Numerical Simulation of Supercritical Flow in Bends of Channel.” *International Conference on Mechanical, Automotive and Materials Engineering (ICMAME’2012),* Dubai, United Arab Emirates, 167-171.
Montes, J.S., (1994). “Potential-Flow Solution to 2D Transition from Mild to Steep Slope.” *Journal of Hydraulic Engineering, ASCE,* 1(5), 601-621.
Poggi, B. (1956). “Correnti veloci nei canali in curva.” *L’Energia Elettrica,* 34, 465-480.
Rakotoarivelo, W., and Sananes, F. (1967). “Etude de l’ècoulement supercritique dans un canal dont la section est en forme de U.” *12 IAHR Congress,* Fort Collins 1(38), 1-7.
Reinauer, R., and Hager, W.H. (1997). “Supercritical Bend Flow.” *Journal of Hydraulic Engineering,* 123(3), 208-218.
Sananes, F., and Acatay, T. (1962). “Contribution à l’ètude des ècoulements supercritiques dans des canaux coudès.” *Comptes Rendus de L’Acadèmie des Sciences,* Paris, France, 255, 471-473.
Shukry, A. (1950). “Flow around bends in an open flume.” *Transactions, ASCE,* 115, 751-788.
Valiani, A., and Caleffi, V. (2005). “Brief Analysis of Shallow Water Equations Suitability to Numerically Simulate Supercritical Flow in Sharp Bends.” *ASCE, Journal of Hydraulic Engineering*, 131(10), 912-916.
Von Karman, T. (1938). “Eine praktische Anwendung der Analogie zwischen Überschallströmung in Gasen und überkritischer Strömung in offenen Gerinnen.” *Zeitschrift für Angewandte Mathematik und Mechanik,* 18(1), 49-56 .