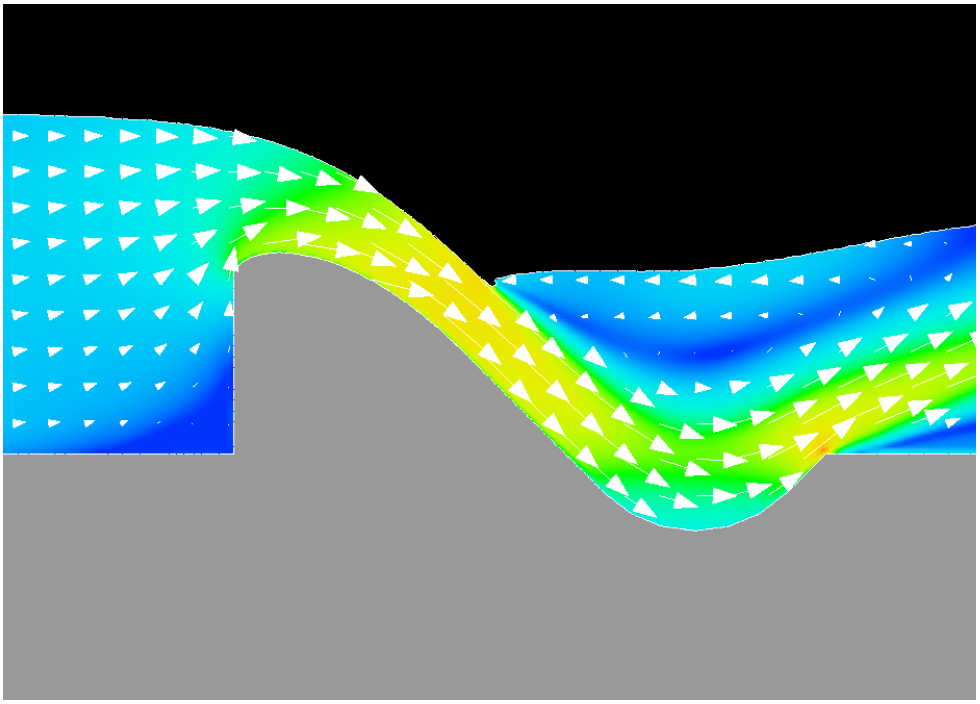
RECLAMATION

Managing Water in the West

Hydraulic Laboratory Technical Memorandum PAP-1070

Composite Modeling of the Halfway Wash Fish Barrier



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Hydraulic Investigations and Laboratory Services Group
Denver, Colorado

January 2013

Hydraulic Laboratory Technical Memorandum PAP-1070

Composite Modeling of the Halfway Wash Fish Barrier

Heiner

Prepared: Bryan J. Heiner
Hydraulic Engineer, Hydraulic Investigations and Laboratory Services Group, 86-68460

Robert F. Einhellig

Technical Approval: Robert F. Einhellig, P.E.
Manager, Hydraulic Investigations and Laboratory Services Group, 86-68460

Leslie J Hanna

1/3/13

Peer Review: Leslie J. Hanna
Date
Hydraulic Engineer, Hydraulic Investigations and Laboratory Services Group, 86-68460

U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Hydraulic Investigations and Laboratory Services Group
Denver, Colorado

January 2013

Background

Reclamation's Hydraulic Investigations and Laboratory Services group in Denver, CO was asked to develop an effective fish barrier design for the Virgin River at Halfway Wash (approximately 16 miles upstream from Lake Mead). The barrier is intended to protect native listed fish species by preventing upstream passage of invasive species entering the Virgin River from Lake Mead. Reclamation's Provo Area office was overseeing this project and asked the Hydraulics Laboratory to construct a physical model of the fish barrier to optimize its use as a deterrent while minimizing erosion immediately downstream of the barrier (Hanna and Lentz, 2012).

A 1:5 scale physical model was used to optimize the fish barrier design to prevent upstream passage of invasive species and minimize erosion caused by the barrier. After testing three different designs, an ogee crest with a downstream roller bucket, as shown in Figure 1, was determined as the most effective method for meeting all design criteria.

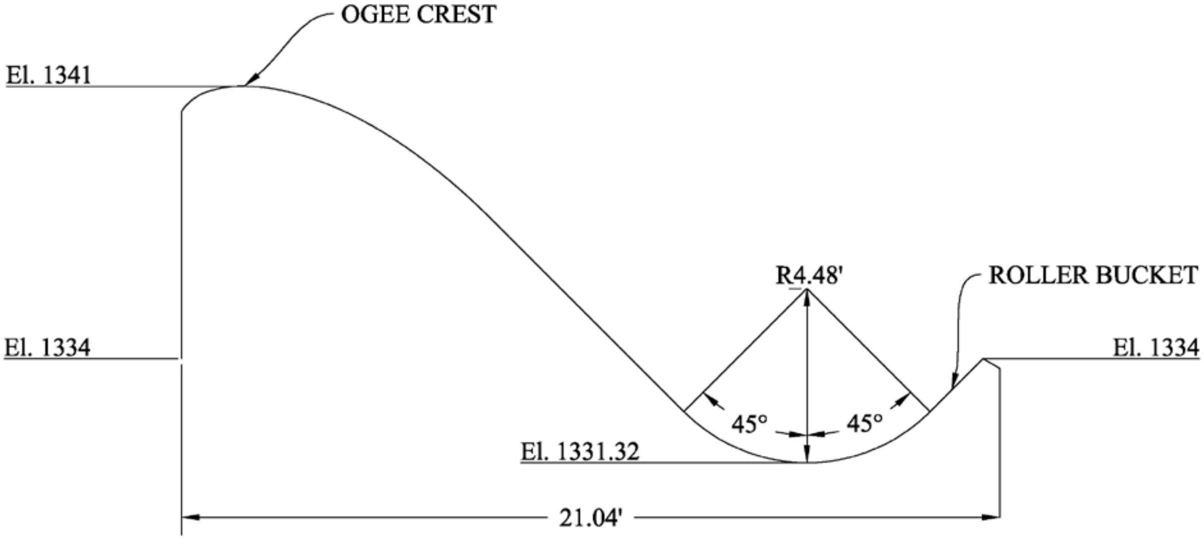


Figure 1 - Outline for the final design of the Virgin River fish barrier at Halfway Wash (prototype dimensions and elevations).

Due to limitations of the physical model, it was difficult to accurately measure the depth and velocity at specific locations along the ogee crest. Therefore, depths along the surface of the barrier were measured conservatively to ensure fish criteria were met for the full range of flow conditions expected in the prototype. It was also difficult to quantify the amount of turbulence in the roller bucket. Instead, the physical model relied more on a qualitative assessment based on visual comparisons between designs. The issue of collecting all the necessary data when conducting a scaled physical model can sometimes be a concern.

Introduction

Composite modeling was investigated as a more efficient and economical alternative to conducting only physical modeling for the Virgin River fish barrier case study. Composite modeling is when computational fluid dynamics (CFD) or other numerical models are used in conjunction with physical modeling to obtain more detailed information (Rahmeyer, et al. 2011, Gerritsen & Sutherland, 2011). For select studies, CFD modeling can reduce the number of iterations needed in the physical model and provide more detailed hydraulic information that may be difficult to obtain in physical modeling. Using the Halfway Wash fish barrier as a case study, this reports presents both physical and numerical model results and a simple cost-benefit analysis.

The ability to efficiently design and construct fish barriers is vital for the Bureau of Reclamation to meet its goal to manage, develop, and protect water and related resources in an environmentally and economically sound manner in the interest of the American public. As Reclamation continues to explore ways to mitigate the spread and impact of non-native aquatic species, more fish barrier installations may be required to limit potential damage to native fish species and their habitats. Composite modeling is an evolving tool that has the potential to improve current methods that are used to design fish barriers and other hydraulic structures.

Model Setups

Physical Model

A physical hydraulic model was constructed at Reclamation's Hydraulic Laboratory in Denver, CO. Due to the extreme width of the river channel and the wide range of flow conditions to be tested, a 1:5 geometric scale sectional model was used to represent the structure. Approximately 19.75-ft of the prototype structure was modeled in a 4-ft wide permanent testing flume. Tailwater in the physical model was set to match depths determined from a one dimensional HEC-RAS model of the Virgin River. Upstream depths were measured 4 model-feet upstream from the barrier crest. Downstream depths were measured 12.5 model-feet downstream from the roller bucket. Other water surface measurements and velocities were taken along the length of the structure to determine if design criteria were met properly. Model flow rates were measured using calibrated venturi meters. For a complete description of the physical model refer to the complete laboratory report (Hanna and Lentz, 2012).

Computational Fluid Dynamics Model

FLOW-3D, a commercially available CFD software package by Flow Science Inc. was used for all CFD simulations due to its ability to accurately track free surfaces. FLOW-3D utilizes the Reynolds-averaged Navier-Stokes (RANS) equations to solve for fluid flow. Modifications to the standard RANS equations include algorithms to track the water surface and flow around geometric objects (Hirt and Nichols, 1981; Flow Science, 2012; Hirt and Sicilian, 1985; Hirt, 1992).

The CFD model was configured using prototype dimensions to avoid any scale effects that may occur at the model scale and to simplify comparison with the physical model. The model was configured in two dimensions having the approximate boundaries in relation to the physical model as shown in Figure 2. A unit width of 1 was used in all CFD modeling to reduce the simulation times and allow for a unit discharge to be introduced at the inflow boundary.

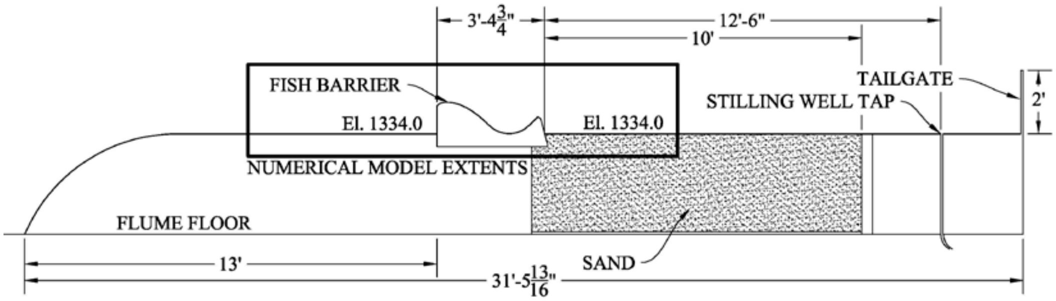


Figure 2 - Approximate boundaries of the numerical model (dimensions in model scale, elevations in prototype scale).

A 3D geometry file of the fish barrier was imported into FLOW3D as a stereo lithography file. The geometry was overlaid with a computational grid having cells 0.5-ft square in the x (streamwise) & z (vertical) direction. When flows over the barrier were shallow, a nested mesh block with smaller cell size was included to resolve the flow over the fish barrier with around ten cells. Repeat runs with smaller cell sizes produced little difference in results but unnecessarily increased the simulation time.

The inflow boundary (negative x) was set to match the physical model flow rates by specifying an approximate fluid height (hydrostatic in the z direction) and discharge for each simulation. A symmetry boundary condition was applied along both sides of the numerical model (±y). The floor (negative z) was set as a no-slip boundary condition. The top (positive z) boundary was set as a pressure boundary with gauge pressure equal to zero. The fluid exited the simulation through the positive x boundary which was set as a pressure boundary with fluid height equal to the HEC-RAS calculated tailwater elevations (Hanna and Lentz, 2012).

Turbulence was modeled using the Renormalized Group theory (RNG) because it more accurately describes low intensity turbulent flows and flows with strong shear regions using fewer computations than other methods (Flow Science, 2012).

Each test simulation was initialized and allowed to run until it reached steady state. After reaching steady state, data was collected for comparison to the physical model.

Results

Physical model depth measurements were taken along the barrier looking through the glass sidewall of the flume. Depths were determined perpendicular to the barrier surface every 0.5 prototype feet in elevation (Figure 3). The average velocity at each section was calculated from the measured depths and inflow. When depths at a section were adequate, point velocities were also collected using a Swoffer propeller meter raised 1.2 model-inches from the crest surface.

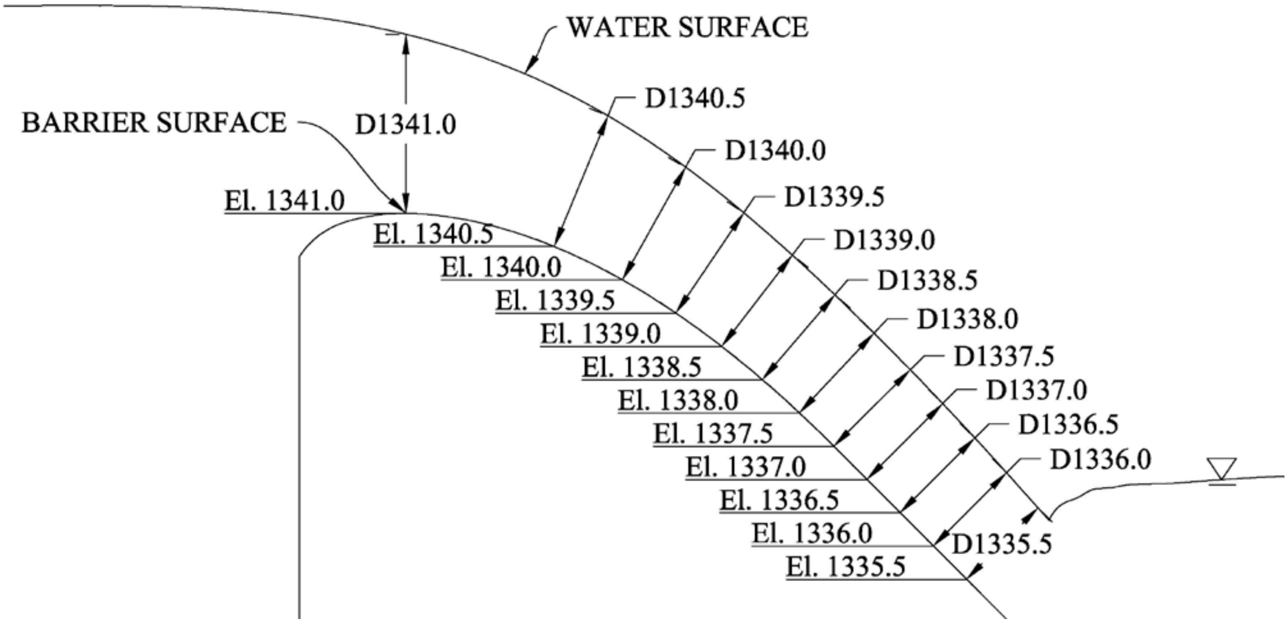


Figure 3 - Depth measurements taken perpendicular to crest.

Numerical model results were output at the same locations as in the physical model with the exception of the Swoffer point velocity measurements. Flow conditions along with upstream, downstream and crest water surface elevations including the absolute difference ("Diff.") for both the physical and numerical models are presented in Table 1 (all values are presented in prototype scale).

Table 1 - Upstream, downstream and crest water surface elevations (WSE) for various discharges evaluated in both models.

	Upstream WSE			Downstream WSE			Crest WSE		
Discharge	Physical	CFD	Diff.	Physical	CFD	Diff.	Physical	CFD	Diff.
(ft^3/sec)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)
200	1341.18	1341.16	0.02	1335.11	1335.11	0.00	1341.18	1341.10	0.08
600	1341.28	1341.33	-0.05	1335.53	1335.53	0.00	1341.26	1341.22	0.04
1,000	1341.43	1341.46	-0.03	1335.80	1335.81	-0.01	1341.34	1341.31	0.03
5,000	1342.33	1342.30	0.03	1337.48	1337.48	0.00	1342.02	1341.91	0.11
10,000	1343.05	1343.00	0.05	1338.68	1338.68	0.00	1342.60	1342.46	0.15
20,000	1344.08	1344.02	0.05	1339.93	1339.93	0.00	1343.47	1343.26	0.21
30,000	1344.93	1344.84	0.08	1341.08	1341.07	0.01	1344.00	1343.93	0.07
45,000	1345.88	1345.87	0.00	1342.48	1342.46	0.02	1344.17	1344.76	-0.59

Depth and Velocity

Tables 2-6 compare the depths and velocities obtained from both the physical and numerical models. The tables are presented in prototype measurements with the difference in values provided in both feet or ft/sec and percent difference. Any values in the table that are crossed out were not included in further analysis because they appear to be erroneous physical model readings.

Table 2 - Prototype depth and velocity comparison for 5,000 ft^3/sec

5,000 ft^3/sec							
	Prototype Depth			Prototype Velocity			
Elevation (ft)	Model (ft)	CFD (ft)	Difference (ft & (%))	Model Calc. Avg. (ft/sec)	Model Point (ft/sec)	CFD Depth Averaged (ft/sec)	Difference (ft/sec & (%))
1341.00	1.02	0.91	-0.11 (-11%)	4.90	6.00	5.38	0.48 (10%)
1340.50	0.56	0.50	-0.06 (-11%)	8.90	9.50	9.68	0.78 (9%)
1340.00	0.50	0.37	-0.13 (-26%)	10.10	11.90	11.83	1.73 (17%)
1339.50	0.44	0.34	-0.1 (-22%)	11.30	13.10	13.12	1.82 (16%)
1339.00	0.42	0.30	-0.12 (-28%)	12.00	N/A	14.75	2.75 (23%)
1338.50	0.39	0.31	-0.08 (-22%)	12.80	N/A	15.73	2.93 (23%)
1338.00	0.35	0.24	-0.11 (-32%)	14.30	N/A	16.90	2.6 (18%)
1337.50	0.34	0.24	-0.09 (-28%)	14.90	N/A	17.91	3.01 (20%)
1337.00	0.31	0.23	-0.09 (-28%)	16.00	N/A	18.95	2.95 (18%)

Table 3 - Prototype depth and velocity comparison for 10,000 ft^3/sec

10,000 ft^3/sec							
	Prototype Depth			Prototype Velocity			
Elevation (ft)	Model (ft)	CFD (ft)	Difference (ft & (%))	Model Calc. Avg. (ft/sec)	Model Point (ft/sec)	CFD Depth Averaged (ft/sec)	Difference (ft/sec & (%))
1341.00	1.60	1.46	-0.15 (-9%)	6.20	7.80	6.81	0.61 (10%)
1340.50	1.04	0.89	-0.15 (-14%)	9.60	11.20	11.02	1.42 (15%)
1340.00	0.86	0.73	-0.13 (-15%)	11.63	13.10	12.93	1.3 (11%)
1339.50	0.81	0.67	-0.14 (-17%)	12.40	13.90	14.30	1.9 (15%)
1339.00	0.76	0.59	-0.16 (-21%)	13.20	15.30	15.74	2.54 (19%)
1338.50	0.68	0.58	-0.1 (-15%)	14.80	16.30	16.63	1.83 (12%)
1338.00	0.65	0.53	-0.12 (-18%)	15.40	16.30	17.63	2.23 (14%)
1337.50	0.65	0.51	-0.14 (-22%)	15.40	N/A	18.43	3.03 (20%)

Table 4 - Prototype depth and velocity comparison for 20,000 ft^3/sec

20,000 ft^3/sec							
	Prototype Depth			Prototype Velocity			
Elevation (ft)	Model (ft)	CFD (ft)	Difference (ft & (%))	Model Calc. Avg. (ft/sec)	Model Point (ft/sec)	CFD Depth Averaged (ft/sec)	Difference (ft/sec & (%))
1341.00	2.47	2.26	-0.21 (-9%)	8.10	10.30	8.82	0.72 (9%)
1340.50	1.77	1.46	-0.31 (-18%)	11.30	13.30	12.92	1.62 (14%)
1340.00	1.55	1.27	-0.28 (-18%)	12.90	14.10	14.70	1.8 (14%)
1339.50	1.46	1.20	-0.26 (-18%)	13.70	17.10	16.13	2.43 (18%)
1339.00	1.35	1.05	-0.3 (-22%)	14.80	18.10	17.28	2.48 (17%)
1338.50	1.29	1.07	-0.22 (-17%)	15.50	19.20	18.30	2.8 (18%)
1338.00	1.25	1.02	-0.23 (-18%)	16.00	19.40	18.98	2.98 (19%)

Table 5 - Prototype depth and velocity comparison for 30,000 ft^3/sec

30,000 ft^3/sec							
	Prototype Depth			Prototype Velocity			
Elevation (ft)	Model (ft)	CFD (ft)	Difference (ft & (%))	Model Calc. Avg. (ft/sec)	Model Point (ft/sec)	CFD Depth Averaged (ft/sec)	Difference (ft/sec & (%))
1341.00	3.00	2.93	-0.07 (-2%)	10.00	13.20	10.28	0.28 (3%)
1340.50	2.32	2.02	-0.29 (-13%)	13.00	15.10	14.23	1.23 (9%)
1340.00	2.08	1.81	-0.27 (-13%)	14.40	17.50	15.92	1.52 (11%)
1339.50	1.95	1.69	-0.26 (-13%)	15.40	18.20	17.25	1.85 (12%)
1339.00	1.80	1.58	-0.22 (-12%)	16.70	19.20	18.21	1.51 (9%)

Table 6 - Prototype depth and velocity comparison for 45,000 ft^3/sec

45,000 ft^3/sec							
	Prototype Depth			Prototype Velocity			
Elevation (ft)	Model (ft)	CFD (ft)	Difference (ft & (%))	Model Calc. Avg. (ft/sec)	Model Point (ft/sec)	CFD Depth Averaged (ft/sec)	Difference (ft/sec & (%))
1341.00	~~3.17~~	3.76	~~0.59 (19%)~~	~~14.20~~	14.30	12.02	~~-2.18 (-15%)~~
1340.50	2.97	2.77	-0.2 (-7%)	15.20	17.30	15.82	0.62 (4%)
1340.00	2.75	2.52	-0.23 (-8%)	16.40	19.20	17.33	0.93 (6%)
1339.50	2.54	2.37	-0.17 (-7%)	17.70	19.30	18.43	0.73 (4%)
1339.00	2.47	2.26	-0.22 (-9%)	18.20	19.20	19.18	0.98 (5%)

Figures 4 and 5 plot the depth and velocity data from Tables 2-6. The black dashed line represents a perfect fit (physical model and numerical model results are identical). The data includes a linear fit trend line of all the data.

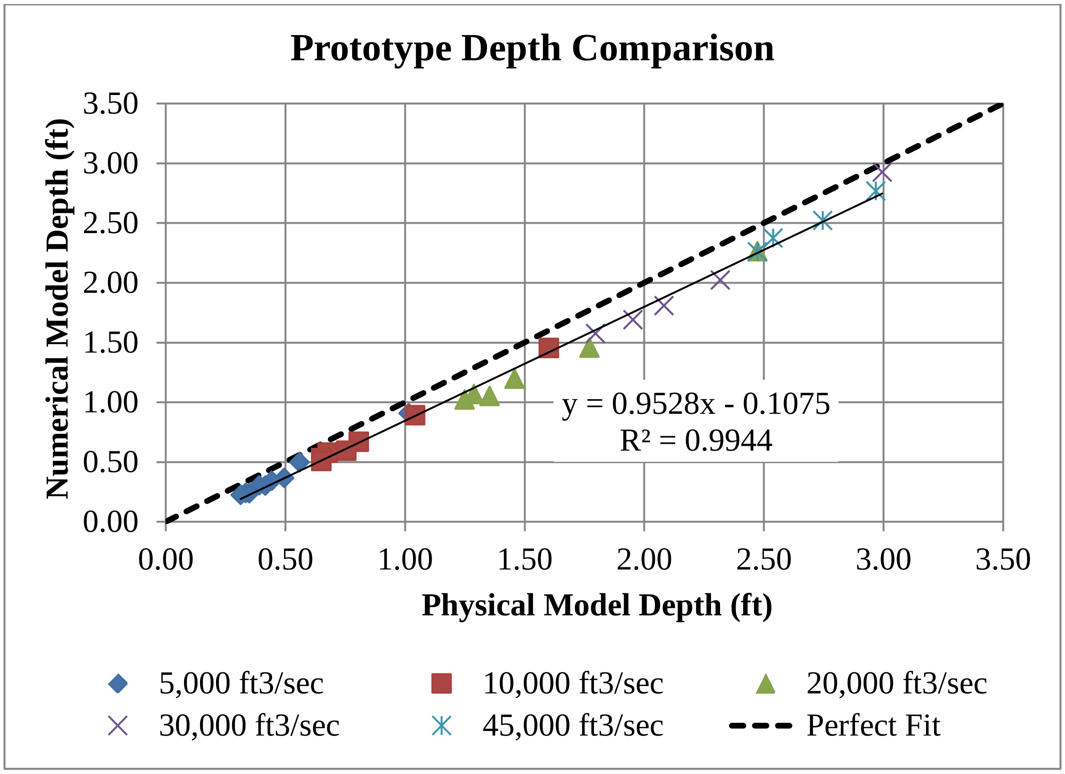


Figure 4 - Physical and numerical model depth comparisons (prototype scale)

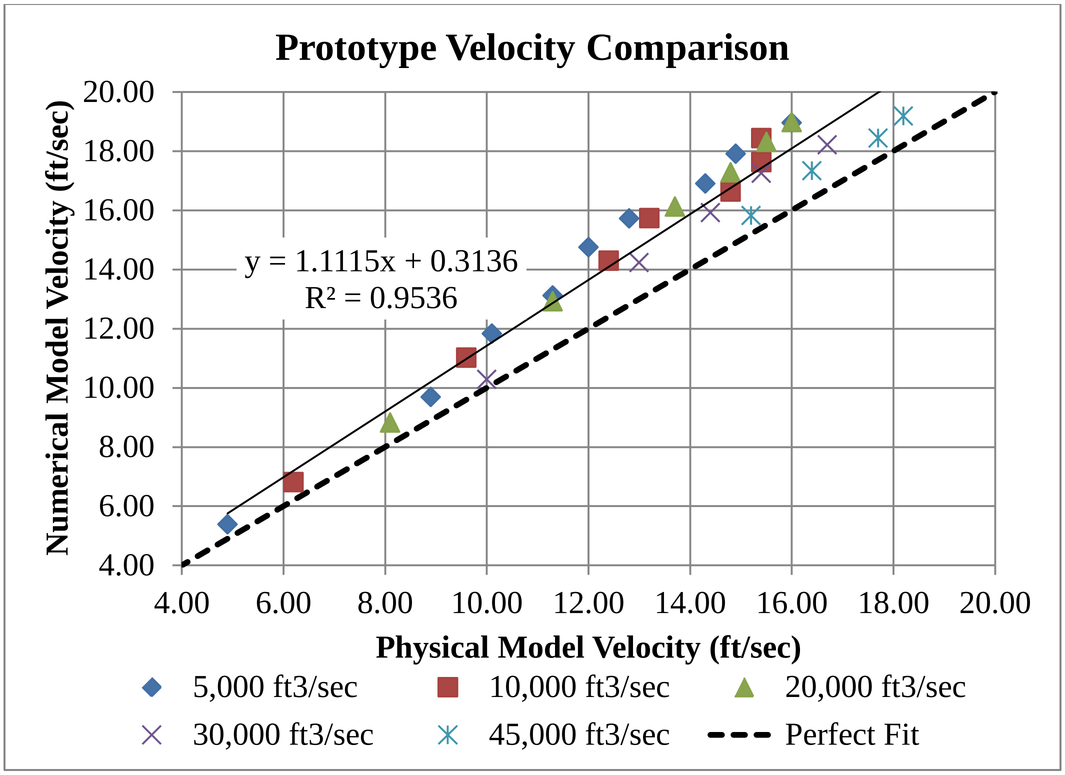


Figure 5 - Physical and numerical model velocity comparisons (prototype scale)

Turbulent Energy

Numerical modeling allowed the turbulent kinetic energy in the roller bucket to be determined. Table 7 gives the maximum turbulent kinetic energy from the numerical model for each flow rate. The turbulent energy for the 200 and 600 ft^3/sec discharges were not determined due to limitations of the numerical model.

Table 7 - Maximum turbulent kinetic energy determined with the numerical model

Prototype Discharge	Max Turbulent Energy
(ft^3/sec)	(ft^2/sec^2)
200	NA
600	NA
1,000	4.27
5,000	8.57
10,000	11.16
20,000	14.22
30,000	15.54
45,000	16.87

Discussion

Depth and Velocity

Visually comparing the water surface profiles created from both the numerical and physical models provides important information regarding any outliers in the data. Figures 6-10 provide the velocity contour plots from the numerical model. Included in each figure are plots of the velocity profile across each of the sections (Figure 3) that were not submerged with the tailwater. The white “+” signs plot the physical model data in the same coordinate system as the numerical model. As shown in Figure 10, the white “+” with a circle around it does not align well with the other physical model data. Further investigations showed that some of the physical model data were difficult to obtain where structural cross members in the physical model were blocking a line of sight view. The location where this measurement was taken was adjusted slightly downstream from the originally intended location. This measurement was excluded from further analysis.

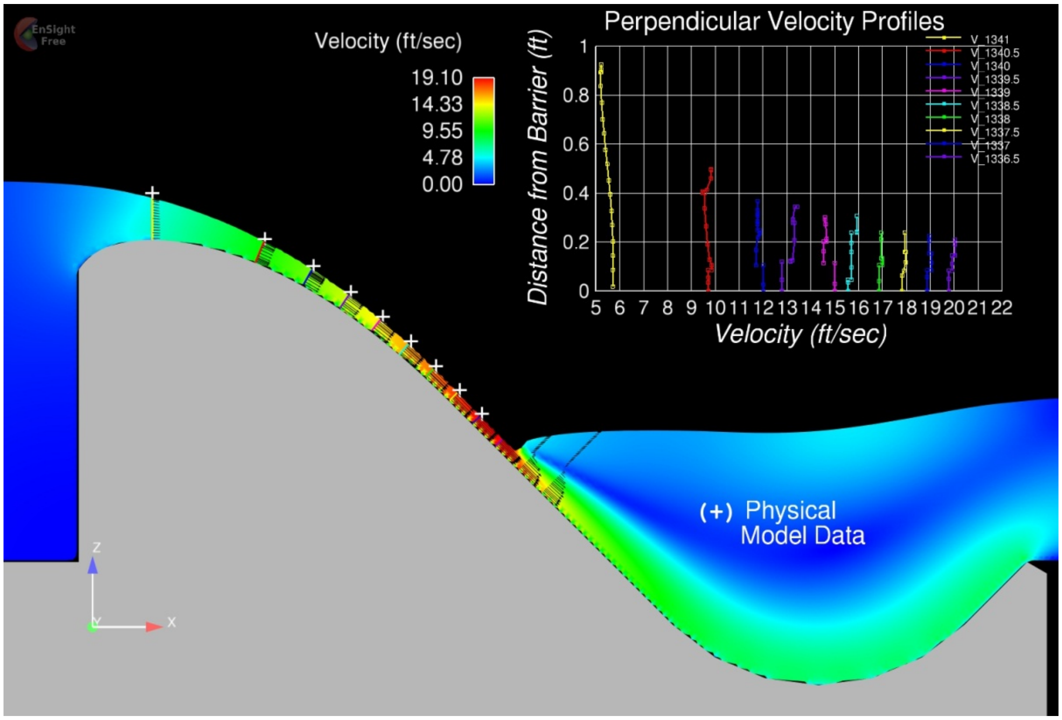


Figure 6 - Numerical model velocity contour and profiles with physical model overlaid at 5,000 ft^3/sec

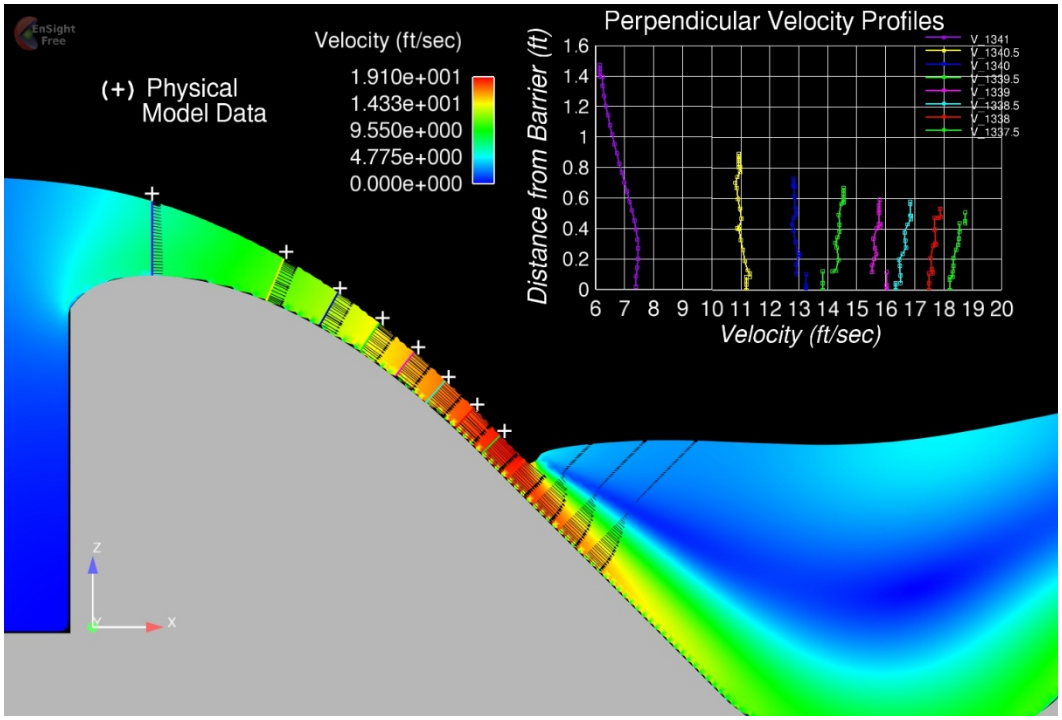


Figure 7 - Numerical model velocity contour and profiles with physical model overlaid at 10,000 ft^3/sec

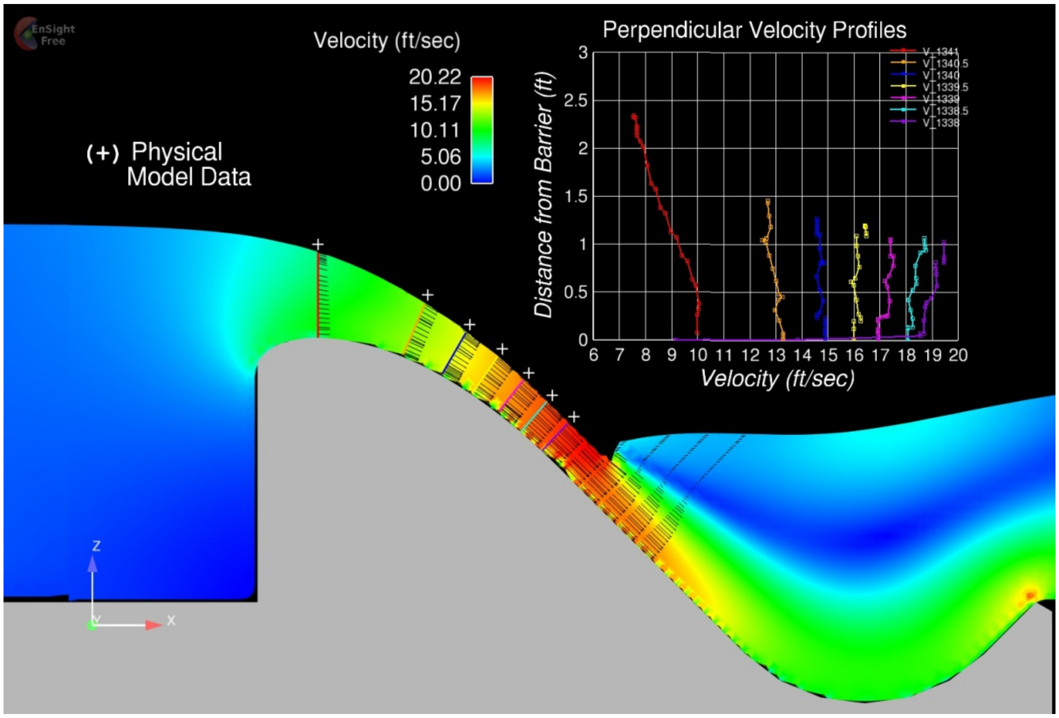


Figure 8 - Numerical model velocity contour and profiles with physical model overlaid at 20,000 ft^3/sec

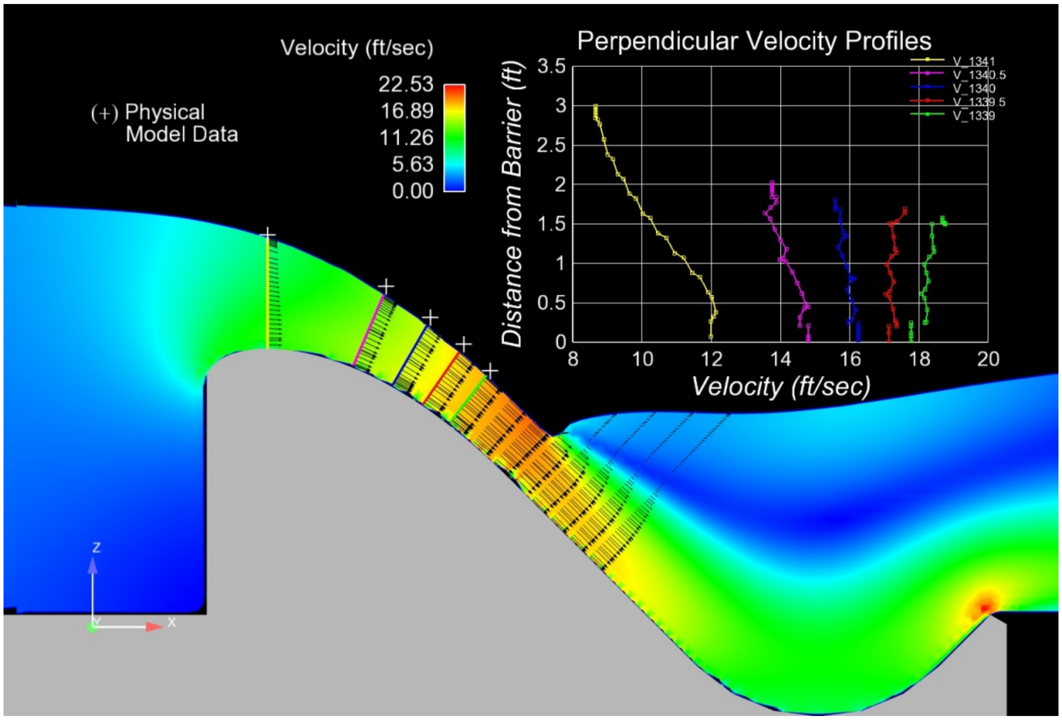


Figure 9 - Numerical model velocity contour and profiles with physical model overlaid at 30,000 ft^3/sec

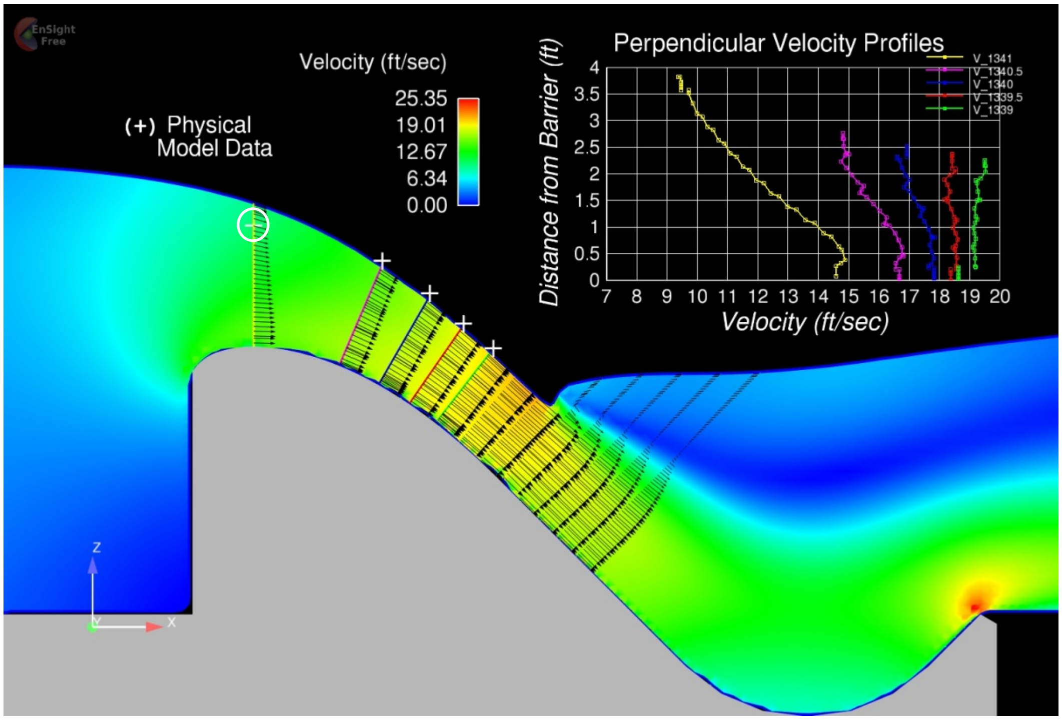


Figure 10 - Numerical model velocity contour and profiles with physical model overlaid at 45,000 ft^3/sec

Upstream water surface elevations compare very well between the models having a max, min and average deviation of 0.8, -0.05 and 0.02 ft with a standard deviation of 0.05 ft, prototype scale.

Downstream water surface elevations compare very well between the models having a max, min, and average deviation of 0.02, -0.01, and 0.00 ft with a standard deviation of 0.009 ft, prototype scale.

Water surface elevations at the crest of the barrier compare very well between the models having a max, min, and average deviation of 0.21, 0.03, and 0.10 ft with a standard deviation of 0.06 ft, prototype scale.

The perpendicular depths along the barrier determined from the numerical model were consistently less than the depths from the physical model. To ensure that the numerical model was calculating these depths accurately over the barrier crest, Flow Science was contacted and asked to review the model and provide their recommendations. Minor adjustments were made to the numerical model input parameters to determine if the depths would change. None of the recommendations modified the results of the numerical model.

Depth measurements used to calculate velocities in the physical model were measured conservatively high to ensure drop height and velocity criteria were met for the final barrier design. Therefore, it was not necessary to obtain the exact water surface profile during the physical modeling. Some minor adjustments were made to the location where the depth of flow over the barrier was taken. These adjustments enabled the modelers to obtain the necessary data and still meet the desired goal and could be the cause for some of the deviations in the depth measurements along the barrier.

No literature was found that directly compares physical model water surface profiles and velocities over an ogee crest with numerical model results. However, other research has shown that numerical modeling can provide accurate water surface profiles through flumes of various types (Heiner, 2009; Temeepattanapongsa, 2012). Previous studies only compared water surface profiles in subcritical flow regimes, not in supercritical regimes as is the case with this study.

Depth averaged velocities calculated in the numerical model across the same perpendicular sections were consistently higher than those in the physical model. This is a result of the numerical depths being less than the physical depths. The velocity profiles included in Figures 6-10 show that flow becomes more uniform over the depth as it approaches the tailwater hydraulic jump.

Turbulent Energy

Figures 12-16 provide contour plots of turbulent energy provided in units of ft^2/sec^2. Typically when upstream fish migrants encounter an obstacle that is difficult to pass over they will stage directly below the obstacle and use their burst swimming speeds to overcome a high velocity or large drop. Increasing the turbulent energy in areas that fish typically use to stage before traversing an obstacle (barrier) is likely to reduce the frequency that fish will be able to pass. With this current barrier, 100 percent exclusion was the goal so decreasing staging areas is vital to ensuring success of the project. When the discharge over the barrier increases stream surface drop decreases and max turbulent energy increases (Table 8 and Figure 11). In the future it may be possible to quantitatively relate turbulent energy to the potential for a structure to act as a fish deterrent. Stream surface drop was calculated by subtracting the downstream water surface elevation from the crest water surface elevation.

Table 8 - Max turbulent energy and stream surface drop

Prototype Discharge	Stream Surface Drop		Max Turbulent Energy
	Physical	Numerical	
(ft^3/sec)	(ft)	(ft)	(ft^2/sec^2)
200	6.1	6.0	NA
600	5.7	5.7	NA
1,000	5.5	5.5	4.27
5,000	4.5	4.4	8.57
10,000	3.9	3.8	11.16
20,000	3.5	3.3	14.22
30,000	2.9	2.9	15.54
45,000	1.7	2.3	16.87

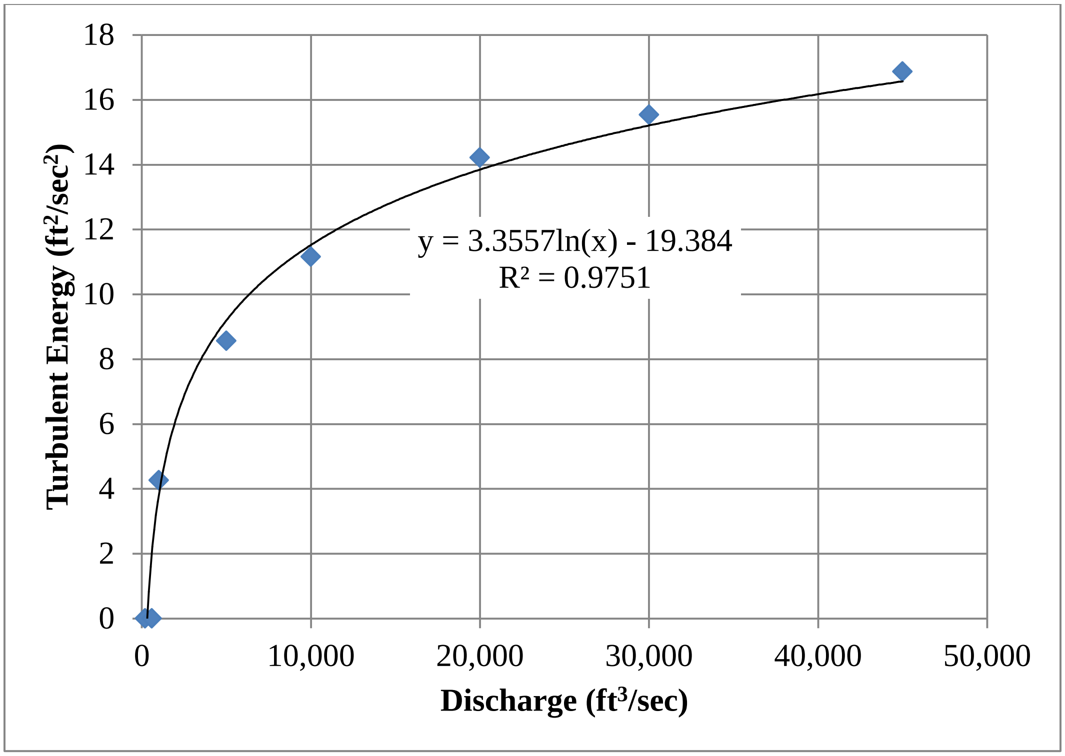


Figure 11 - Max turbulent energy and barrier discharge.

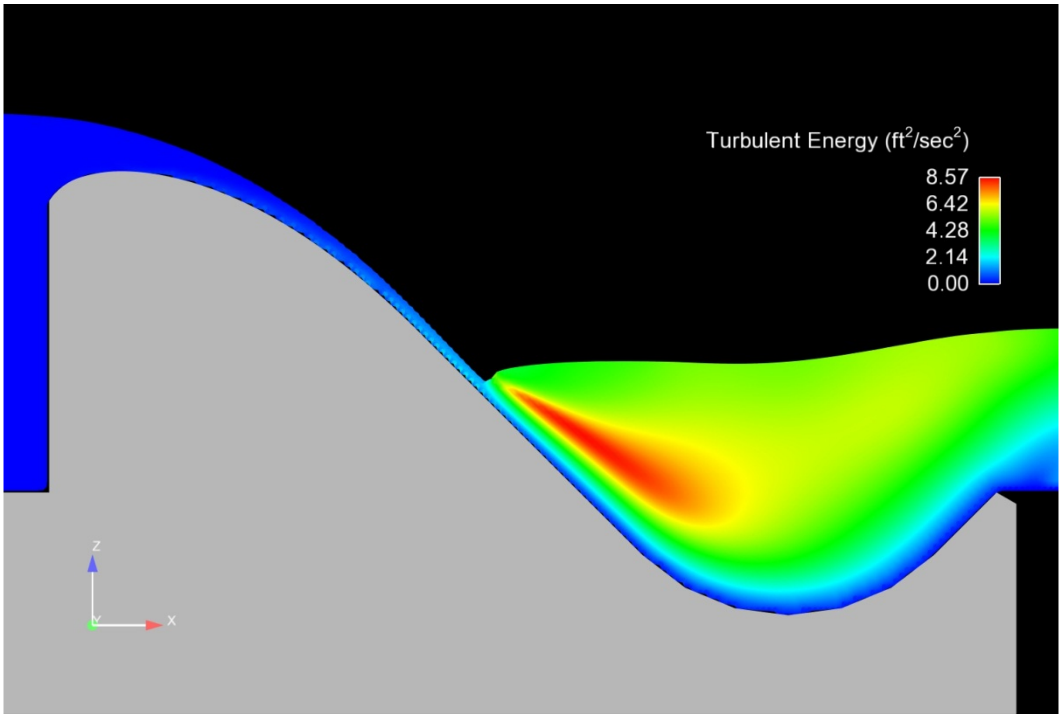


Figure 12 - Turbulent kinetic energy for 5,000 ft^3/sec.

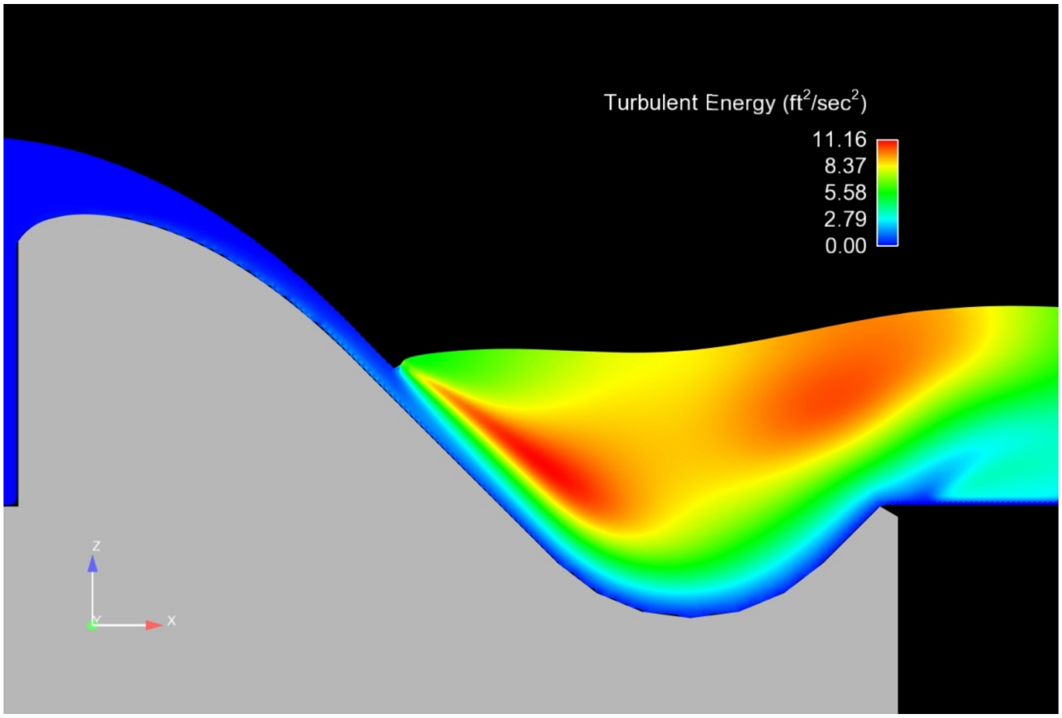


Figure 13 - Turbulent kinetic energy for 10,000 ft^3/sec.

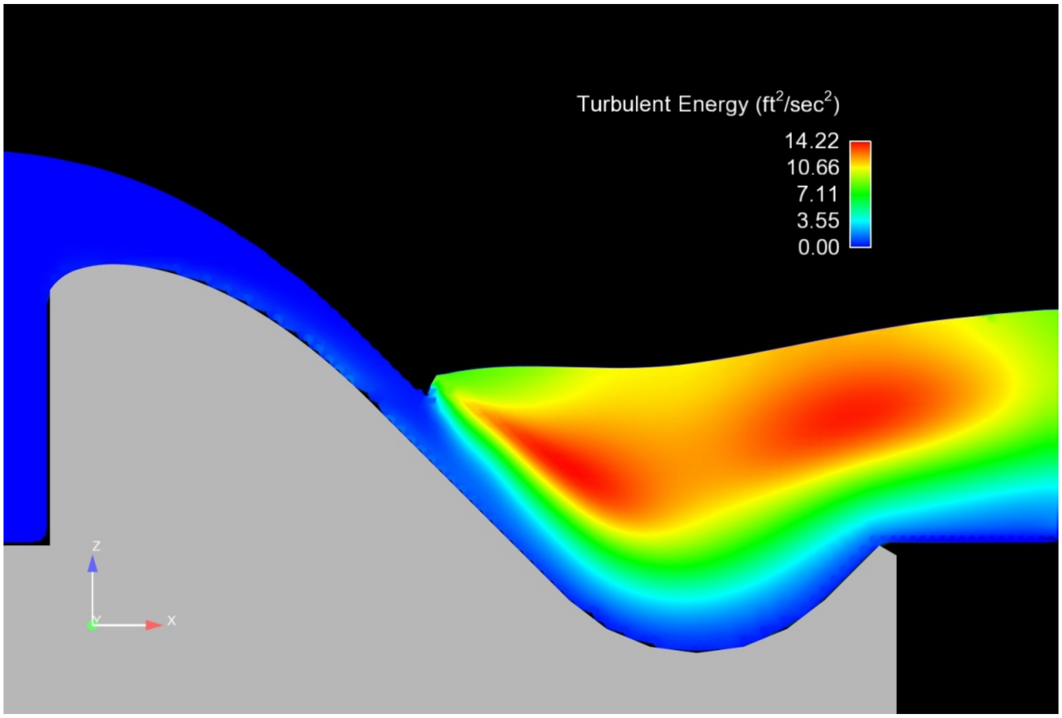


Figure 14 - Turbulent kinetic energy for 20,000 ft^3/sec.

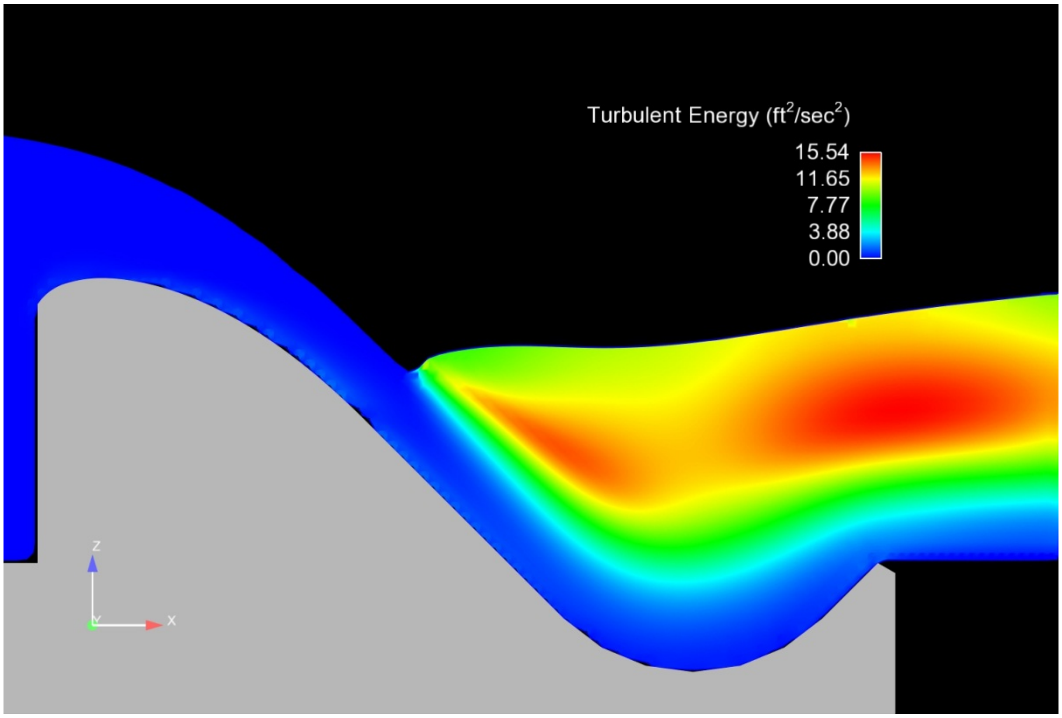


Figure 15 - Turbulent kinetic energy for 30,000 ft^3/sec.

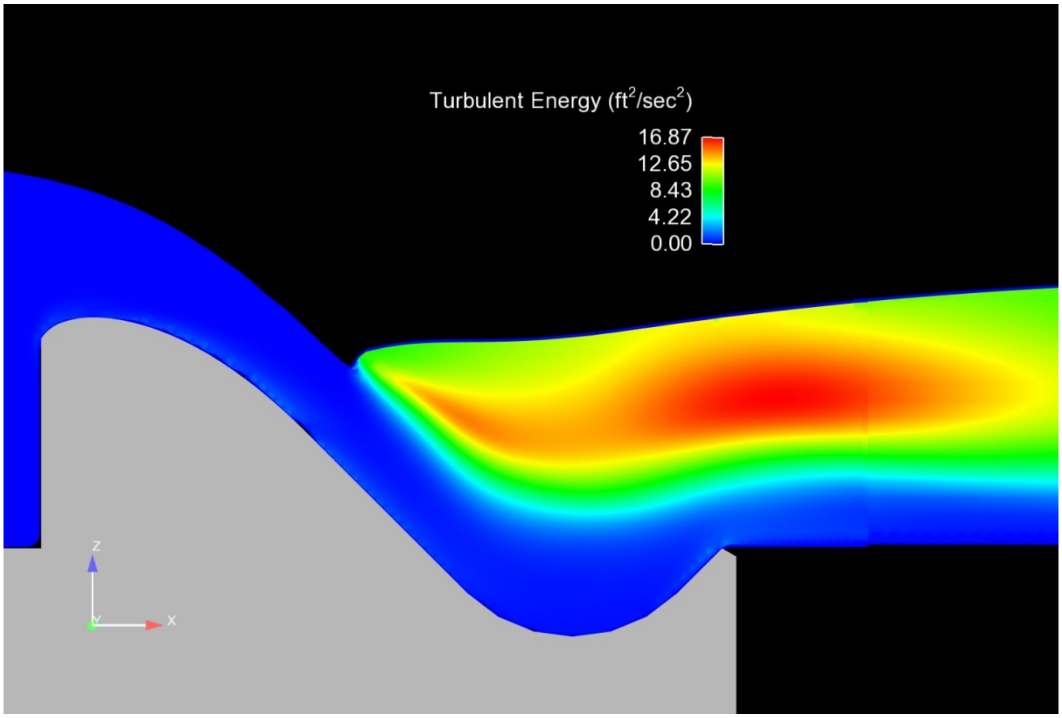


Figure 16 - Turbulent kinetic energy for 45,000 ft^3/sec.

Cost Benefit Analysis

Regardless of the deviations between the numerical and physical model, the results of the final design determined from the original study would not have changed if composite modeling (numerical and physical) were conducted. A simple cost analysis was performed for this particular study comparing the overall cost of the project as performed (three iterations of the physical model with sediment scour included) and an estimated cost for the project if composite modeling were used (at least three iterations of the physical model and one iteration of the physical model with sediment scour included).

Original Study (Hanna and Lentz, 2012):
Physical Modeling, Analysis & Report = $127,000

Study Using Composite Modeling (Physical and Numerical Modeling):
CFD Modeling Portion = $25,000
Physical Modeling (Erosion + Verify Final Design) = $50,000
Analysis & Report Writing = $15,000

Total = $90,000

Estimated savings using composite modeling = $37,000 or about 30%

It should be noted that all projects are unique in scope and size and will not see the same savings from doing composite modeling. The numerical model in this study was conducted in 2D which allowed for short simulation times enabling many iterations of the barrier with a low cost. In some cases using composite modeling could increase the overall cost of a project, however completing composite modeling will likely produce an end product with more detailed information regarding flow fields, currents, velocity profiles, turbulence and many other useful parameters.

Conclusions

Reclamation's Hydraulic Investigations and Laboratory Services group in Denver, CO developed an effective fish barrier for the Virgin River at Halfway Wash. The barrier is intended to protect native listed fish species by preventing upstream passage of invasive species entering the Virgin River from Lake Mead (Hanna and Lentz, 2012). A 1:5 scale physical model was constructed and used to verify that design criteria were met. Later a numerical model was used to compare to the physical model results and quantify the turbulent energy created by the roller bucket. Good agreement between the models existed and a simple cost benefit analysis showed that using composite modeling on this project could have saved about 30 percent of the modeling costs while offering more detailed information of useful parameters such as turbulent kinetic engergy.

In addition using composite modeling could have improved the end product of the study by allowing for more detailed information than could have been obtained by conducting either a numerical or physical model alone. The numerical model could have provided initial design information to more quickly and efficiently determine a final design. The physical model could have then been implemented to verify the numerical results over the full range of flow conditions and incorporate sediment scour which were critical to the final design of the barrier.

References

Flow Science Inc. (2012). “Flow-3D® Help User’s Manual Version 10.0.3.5” USA.

Gerritsen, H., and Sutherland, J. (2011) “Users Guild to Physical Modelling and Experimentation, Experience of the HYDRALAB Network – Chapter 6 – Composite Modeling” Edited by T.G. Mercer CRC Press, Balkema, Leiden, The Netherlands.

Hanna, L. J., and Lentz, D. (2012). “Halfway Wash fish Barrier Physical Hydraulic Model Study.” *Hydraulic Laboratory Report HL-2012-01.* U.S. Bureau of Reclamation, Denver, CO.

Heiner, B. J. (2009). “Parshall Flume Staff Gauge Location and Entrance Wingwall Discharge Calibration Corrections” *All Graduate Theses and Dissertaions.* Paper 480 http://digiatalcommons.usu.edu/etd/480

Hirt, C. W. 1992. “Volume-fraction techniques: Powerful tools for flow modeling.” *Flow Science Rep. No. FSI-92-00-02*, Flow Science, Inc., Santa Fe, N.M.

Hirt, C. W., and Sicilian, J. M. (1985) “A porosity technique for the definition of obstacles in rectangular cell meshes.” *Proc. Fourth International Conf. Ship Hydro.*, National Academy of Science, Washington, DC.

Hirt, C. W., and Nichols, B. D. (1981). “Volume of fluid (VOF) method for the dynamics of free boundaries.” *J. Comput. Phys.*, 39, 201–225.

Rahymeyer, W., Barfuss S., Savage, B. (2011) “Composite Modeling of Hydraulic Structures” *The Association of State dam Safety Officials, Dam Safety 2011 Conference Proceedings.*

Temeepattanapongsa, S. (2012) “Unified Equations for Cutthroat Flumes Derived from a Three-Dimensional Hydraulic Model” *All Graduate Theses and Dissertaions.* Paper 1308 http://digiatalcommons.usu.edu/etd/1308