HYDRODYNAMIC FORCES ON A SPILLWAY: CAN WE CALCULATE THEM?

Bruce M. Savage, Ph.D., P.E., Dept of Civil Engineering, Idaho State University
Michael C. Johnson,Ph.D., P.E., Utah Water Research Laboratory, Utah State University
Brett Towler, Ph.D., P.E., Manager, Civil & Hydraulic Engineering, HDR|DTA

Abstract

The forces acting on a gravity dam spillway section are complicated by the hydrodynamic effects of the flow. Changes to the hydrodynamics forces occur due to the water changing speed and direction. At small discharges, nappe forces are relatively small compared to the hydrostatic forces and are often neglected in a stability analysis. At larger discharges, especially with partially drowned spillways, the nappe forces can be significant and can play a roll in stabilizing a dam. Federal regulatory agencies such as FERC may not allow the full stabilizing forces to be used unless they can be proven. This paper uses pressure data on spillway crests obtained from physical models and compares it with the results obtained via numerical modeling. The focus is primarily for partially drowned spillways due to tailwater effects. The results show that numerical modeling provides a reasonable tool to predict pressures and hence the hydrodynamic forces acting on a spillway.

Introduction

To protect against failure, many dams are being re-evaluated for stability. The re-evaluation often includes an assessment of the hydrology of the contributing watershed and the hydraulics of the spillway. Often improved methods with additional data coupled with new safety regulations have increased the predicted flood flows ranging from the 100-year flood to the Probable Maximum Flood (PMF) that a dam may be required to pass. The improved flood flow estimate may require that a spillway be modified, updated, or resized to handle the predicted increase. The increased flow will also change the hydraulic and hydrodynamic loads acting on the dam and spillway thereby requiring that the stability of the spillway/dam be re-evaluated as well.

In completing a stability analysis of a spillway, it is important to correctly identify and quantify all forces acting on the structure. Many of these forces can be computed using relatively simple methods such as calculating the hydrostatic upstream pressure or the gravity force due to the structure's weight. However, when the force needs to be calculated in a region with accelerating or decelerating flow, calculation of the fluid pressure can be complex. In the past, due to the complexity of hydrodynamic flows, stabilizing forces acting on the downstream side of the spillway that counteract the upstream forces on the face of the dam, were often neglected or significantly reduced in the analysis because they couldn't be accurately computed. Often these forces were considered non-significant because they are relatively small compared to the other forces. However, for large flows and/or when the tailwater is increased significantly thereby increasing the downstream pressure and resultant force acting on the spillway, the hydrodynamic forces can become larger and provide a stabilizing force. Exclusion of these forces does provide a conservative analysis with an enhanced safety factor via the uncounted stabilizing forces but it also comes at an economic cost that could be reduced or eliminated without compromising stability. Alternatively, using the full hydrostatic pressure on a submerged section of the dam may overestimate the hydrodynamic pressure and over predict the stabilizing forces thereby reducing the computed factor of safety. Negative pressures created by flows larger than the design flow on the ogee crest can also affect the safety of a dam. Thus, it is imperative that if the crest and downstream forces are to be used in the analysis, the hydrodynamic pressures must be accurately computed.

In the past, physical models were used to compute flow conditions for a given dam. Although physical models are reliable and accurate they can be time and cost prohibitive. Fortunately, advances in numerical techniques as well as increases in computing power have improved the numerical calculation of hydrodynamic pressure in free surface flows.

This paper compares three different ogee-crested spillways using physical model data and numerical data using Computational Fluid Dynamics (CFD); primarily pressure and flow data. Because CFD is relatively new and untested, this comparison allows engineers to verify the validity and range of CFD as applied to ogee-crested spillways.

Background

Although many spillway designs have been tested and standard operational curves have been published primarily by the USACE and the USBR, there is no single universal spillway design that works for every flow scenario. In fact, both agencies have published design nomographs for a variety of spillway designs. Information on ogee-crested spillways can be found in USACE (1990), Maynord (1985), USBR (1977), Chow (1959), and Bradley (1952). Standard discharge curves provide the ability to quickly determine an approximate flow rate for a given dam configuration. This is a great advantage in the evaluation of a spillway. However, it is reported (USBR 1977) that slight deviations from the standard design may affect the discharge capabilities. The pressures and discharge over an ogee-crested dam and spillway are dependent on the crest and spillway geometry, the upstream flow depth, and in the case of submergence—the tailwater. A slight change in design or operation can leave dam owners and regulators wondering if the change affects the function and stability of a dam. For example, as silting occurs upstream from a dam, how is the flow capacity affected?

A review of the literature shows that there is significant information detailing the change in the discharge coefficient due to dam submergence. Harleman et al. (1963) compiled a bibliography on dam design that includes references to the change in the discharge coefficients due to dam submergence. However, there appears to be very little information outlining changes in spillway pressures due to submergence on the spillway prior to dam submergence. Bradley (1945) defines four distinct types of flow over an ogee-crested dam. Type I flow is characterized by a supercritical jet of water staying attached through the length of the spillway. Type II flow is defined as true hydraulic jump forming on the spillway. Type III flow is defined as a drowned jump. A drowned jump is when the jet of water stays attached to the spillway face and does so for a considerable distance under the tailwater. This occurs when the tailwater depth is too great to allow a good hydraulic jump to form. Type IV flow occurs when the jet breaks up and the dam acts as a broad crested weir. The dam in this case, is generally under a high degree of submergence and the downstream depth becomes a significant variable in controlling the discharge. Shany (1950) noted that as the tailwater depth increases past total submergence for a Type IV flow, the pressure distribution on the downstream face of the dam approaches hydrostatic conditions. The results in this note primarily deal with Flow Types I, II and III.

For Flow Types II and III, there is no consensus on the extent that pressure force can be used in determining the stability of a dam. The U.S. Bureau of Reclamation (1977/1987) states that only the “minimum tailwater level associated with each reservoir level” be used to calculate the pressure force provided by the tailwater (Design Criteria 23; Design of Small Dams 318). Stelle et al. (1983) reported that they assumed the “full value of tailwater pressure” in calculating the stability of the concrete dam, meaning that the full hydrostatic pressure force as calculated using the depth of the tailwater was used. Corn et al. in *Advanced Dam Engineering for Design, Construction, and Rehabilitation* (Jansen, 1988) state that in flood conditions, the tailwater pressure against spillway sections should be based on the discharge height against the dam expected with the type of energy dissipater provided. It is also stated that the full tailwater pressure should be used in uplift determination. The U.S. Army Corps of Engineers (1995) is more conservative, indicating that as little as 60% of the tailwater depth should be used to calculate pressures on a dam when a significant hydraulic jump occurs; although 100% of the tailwater is used for computing uplift pressures. The U.S. Army Corps of Engineers (1995) also states that when the hydraulic jumps is reduced or eliminated that 100% of the tailwater depth can be used. This leaves the question of how much pressure force is on a submerged spillway and at what point is a drowned hydraulic jump sufficiently reduced or eliminated? FERC (2002) notes that the nappe forces can be significant at the design discharge and should be taken into account in the analysis of dam stability. Previous FERC guidance applied the 60% USACE rule but the Brand method is currently preferred. Brand (1999) determines the forces due to the nappe and tailwater by assuming that the hydraulic jump occurs downstream of the spillway; the discharge/velocity is sufficiently large and the tailwater is relatively small thereby blowing the jump downstream.

Unsubmerged Comparisons

In recent years, Computational Fluid Dynamics (CFD) has been used to compute flow over an unsubmerged ogee crest. The results have been compared and evaluated against physical model data as well as USACE and USBR design nomographs. Savage and Johnson (2001) compared the discharge characteristics and pressure distribution for flows passing over an unsubmerged ogee spillway using a physical model, a numerical model, and information published in various design guides (Maynord 1985; USACE 1990; USBR 1977/1987). Ho et al. (2003) compared standard designs for discharge and unsubmerged pressures and found good agreement with published data. Others have completed comparisons between physical model and numerical data and have shown the numerical results to be reasonable including Gesseler (2003) and Chanel and Doering (2008). Although it is subjective to the accuracy required, based on the number of completed studies, numerical modeling of unsubmerged ogee crests is an acceptable method with reasonable results.

Physical Models

In the past, to accurately estimate the hydrodynamic force distribution on a dam and/or spillway, engineers often conducted a physical hydraulic model study of the specific spillway geometry. Although site specific physical models are considered as the best analysis method, they may be costly. Published guidelines compiled from the results of multiple model studies by the U.S. Bureau of Reclamation or the U.S. Army Corps of Engineers are the easiest to use but are the least accurate.

For this study, three physical models, A, B, and C shown in Figs. 1, 2 and 3, respectively, were fabricated using plexiglas and tested at the Utah Water Research Laboratory (UWRL) in Logan, Utah. The models had the distinctive crest shape characteristic of ogee spillways and included a tangent section. Model C had a typical flipbucket whereas models A and B simply transitioned to a horizontal apron. Models A and B had the same ogee crest shape but each had different crest height (P) over design head (H_d) ratios (P/H_d). The P/H_d ratios for models A,B, and C are (need A), 6.5, and 2.7 respectively. The models were instrumented with multiple pressure taps in the center of the model that were used to obtain the pressures on the spillway. The models were approximately six feet wide. Models B and C were approximately 2.6 ft high whereas model A was approximately 0.7 ft high.

The models were tested in a flume that was approximately 40 ft long, 6 ft wide, and 4.0 ft deep. Flow rates were measured using weight tanks, volumetric tanks, or with an ultrasonic flow meter. Pressures on the spillway were measured using a piezometer board with glass tubes vented to atmosphere. The piezometer board readings provided the average pressure reading at each pressure tap location. Measurements on the piezometer board were readable to within 0.05 inches. Because pressures fluctuated temporally along the crest, most likely due to surface waves, an average pressure was recorded. Pressure fluctuations were directly proportional to the amount of turbulence near the tap. The pressure taps near the crest and down the spillway until submerged by the tailwater had fluctuations on the order of ±0.2 inches, whereas pressure taps that were submerged fluctuated on the order of ±0.8 inches.

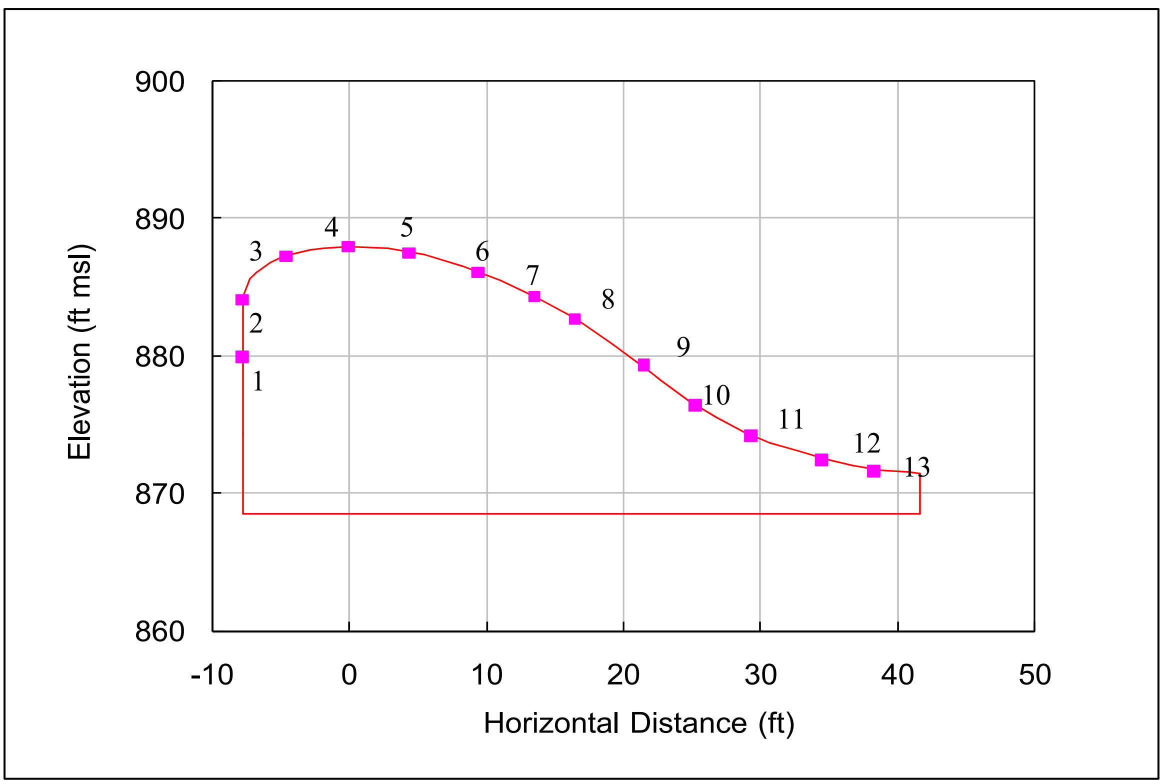


Figure 1. Profile of model A showing the shape, dimensions, and pressure tap locations.

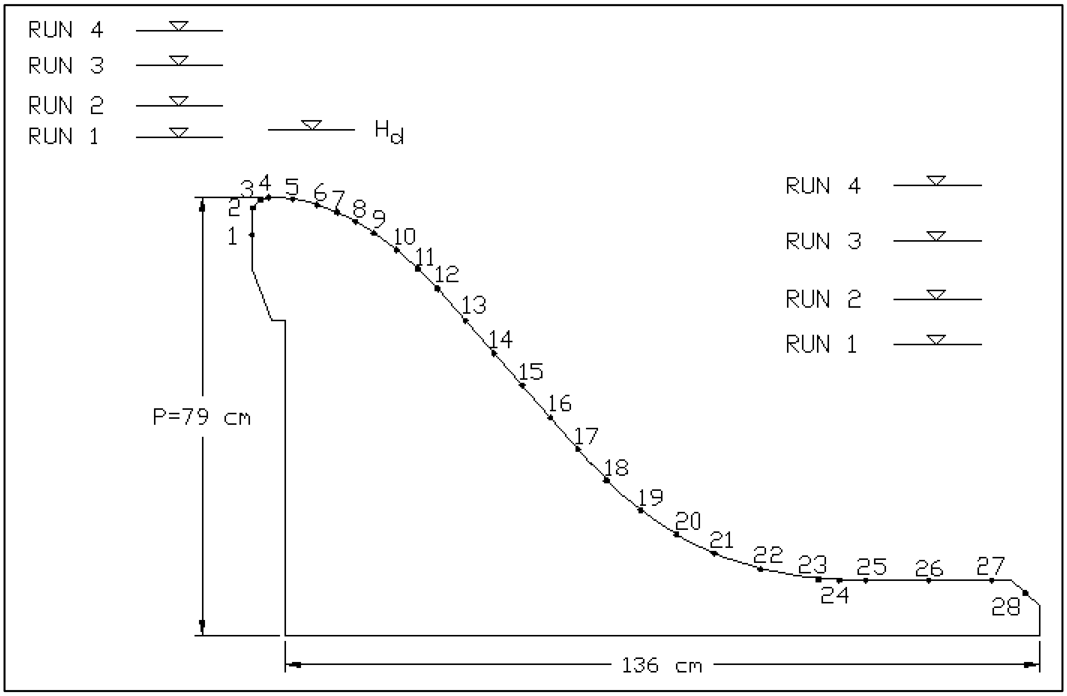


Figure 2. Profile of model B showing the shape, dimensions, pressure tap locations, and headwater and tailwater elevations.

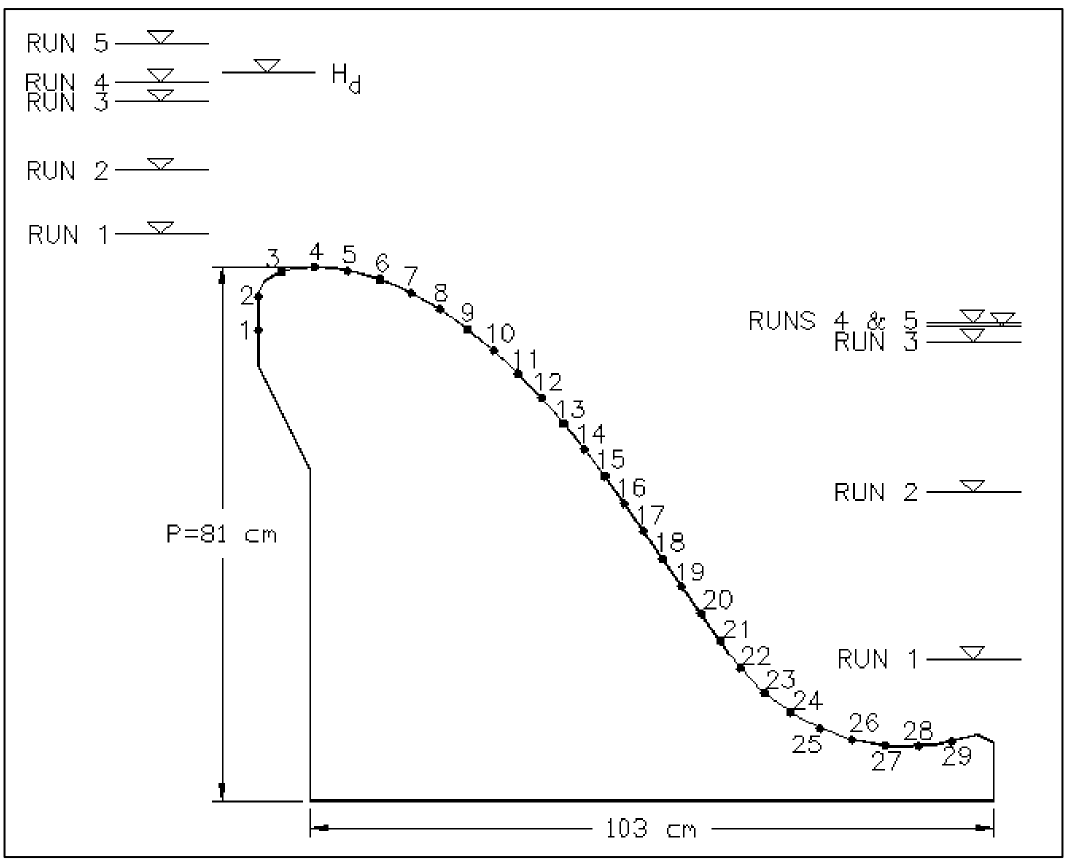


Figure 3. Profile of model C showing the shape, dimensions, pressure tap locations, and headwater and tailwater elevations.

Model Operation

The models were operated at various upstream total heads (H_e) defined as the piezometric plus velocity head with a unique tailwater elevation as shown in Figures 2 and 3. The tailwater was set using stop logs located downstream from the model with the depth measured 10 ft from the spillway crest. The headwater elevation was measured approximately 6.5 ft upstream from the spillway crest. Both the headwater and tailwater elevations were measured using piezometers. The conditions tested are shown in Table 1. Each of the conditions tested had a significant depth of tailwater on the downstream face of the spillway. Models A and B each had a condition where the tailwater was higher than the spillway crest. This is typically considered as submergence for the dam. However, the velocities and depths were sufficient for model A run 4 that no hydraulic jump was formed near the spillway, effectively creating a blown out jump scenario.

Table 1. Model Operating Conditions

	Model A		Model B		Model C	
Design Head (ft)[a]	0.035		0.035		0.965	
Run	Headwater[b] Elevation (feet)	Tailwater Elevation (feet)	Headwater Elevation (feet)	Tailwater Elevation (feet)	Headwater Elevation (feet)	Tailwater Elevation (feet)
1	1.17	0.14	2.95	1.72	2.82	0.70
2	1.52	0.30	3.14	1.98	3.14	1.53
3	1.60	0.40	3.40	2.33	3.49	2.30
4	1.81	0.73	3.64	2.65	3.59	2.35
5	-	-	-	-	3.80	2.37

[a]Design head referenced from the top of the spillway crest (EL ??)
[b]Headwater and Tailwater Elevations referenced from the base of each spillway

Numerical Model

To solve for the flow in the model, a commercially available CFD solver, Flow-3D® from by Flow Science was selected. Flow-3D® was selected because it is a general purpose CFD solver with the reputation of accurately tracking free surfaces. To solve the governing equations of fluid flow, the program solves a modification of the commonly used Reynolds-average Navier-Stokes (RANS) equations. The modifications include algorithms to track the free surface and model the flow past obstacles such as spillways. The modified RANS equations are shown as:

Continuity:
$$\frac{\partial}{\partial x}(uA_x)+\frac{\partial}{\partial y}(vA_y)+\frac{\partial}{\partial z}(wA_z)=0$$

Momentum:
$$\frac{\partial U_i}{\partial t}+\frac{1}{\mathrm{V_F}}\left(U_j A_j \frac{\partial U_i}{\partial x_j}\right)=-\frac{1}{\rho}\frac{\partial P'}{\partial x_i}+g_i+f_i$$

where u, v and w are the velocities in the x, y and z direction; V_F is the fraction of fluid in each cell based on volume; A_x, A_y and A_z are fractional areas open to flow across each cell face in the subscript directions; the variable ρ is the density; P' is defined as the pressure; and g_i is the gravitational force in the subscript direction. The variable f_i represents the Reynolds stresses, which were added by Reynolds-averaging. To solve for f_i, a turbulence model was required and the Renormalized Group (RNG) turbulence model (Yakhot and Orszag, 1986) was used.

One of difficulties in solving flow numerically over a weir is the presence of a free surface that tends to be transient in nature (changing with time), in which case the location must be solved as part of the solution. This is especially difficult when the water surface is rapidly changing with a high degree of curvature, such as when the flow changes from subcritical flow to supercritical flow and back again. This is true with the ogee crested spillways as the nappe can go supercritical as it passes over the spillway and then back to subcritical through a submerged jet.

To find, define, and apply appropriate boundary conditions on a free surface, Flow-3D® uses a true Volume-of-Fluid (VOF) method (Hirt and Nichols, 1981). The VOF method works by defining the volume of fluid within each discretized cell. If a cell is empty it receives a value of 0. If a cell is full, it receives a value of 1. If a cell contains the free surface, it receives a value between 0 and 1 that correlates to the ratio of fluid volume to cell volume. The angle of the water surface in the cell is determined by the location of fluid in surrounding cells. In essence, the location of the water surface within a cell is defined as a first-order approximation; a straight line in 2-D space and a plane surface in 3-D space. Therefore, as the flow field is calculated at each time step, the location of the free surface is updated. This allows the free surface to move temporally and spatially.

To define the weir structure within the grid, a grid porosity technique called the Fractional Area/Volume Obstacle Representation (FAVOR) algorithm is used in the program. The FAVOR algorithm is outlined by Hirt and Sicilian (1985) and Hirt (1992). The FAVOR method is similar to the VOF method in that it also uses a first-order approximation to define the flow obstacle but it doesn't change with time. Cells that are constructed entirely of an obstacle are given a value of 0. Cells outside an obstacle are assigned the value of 1. Cells that contain both obstacle and flow volume are assigned a value per the ratio of the volume of the obstacle to the volume of the cell. The FAVOR method also defines obstacle surface as a straight line in 2-D space and as plane in 3-D space. The obstacle surface also defines the area on each cell face that fluid flux can pass through. It is important to note that although a curved obstacle surface or free surface can be approximated with a straight line, it is nonetheless still an approximation. In order to fit a curved surface, a second-order or higher numerical method is required. Given the VOF and FAVOR methods, the assignment of each cell in this configuration becomes one of five conditions; completely solid, part solid and fluid, completely fluid, part fluid, and completely empty.

Results

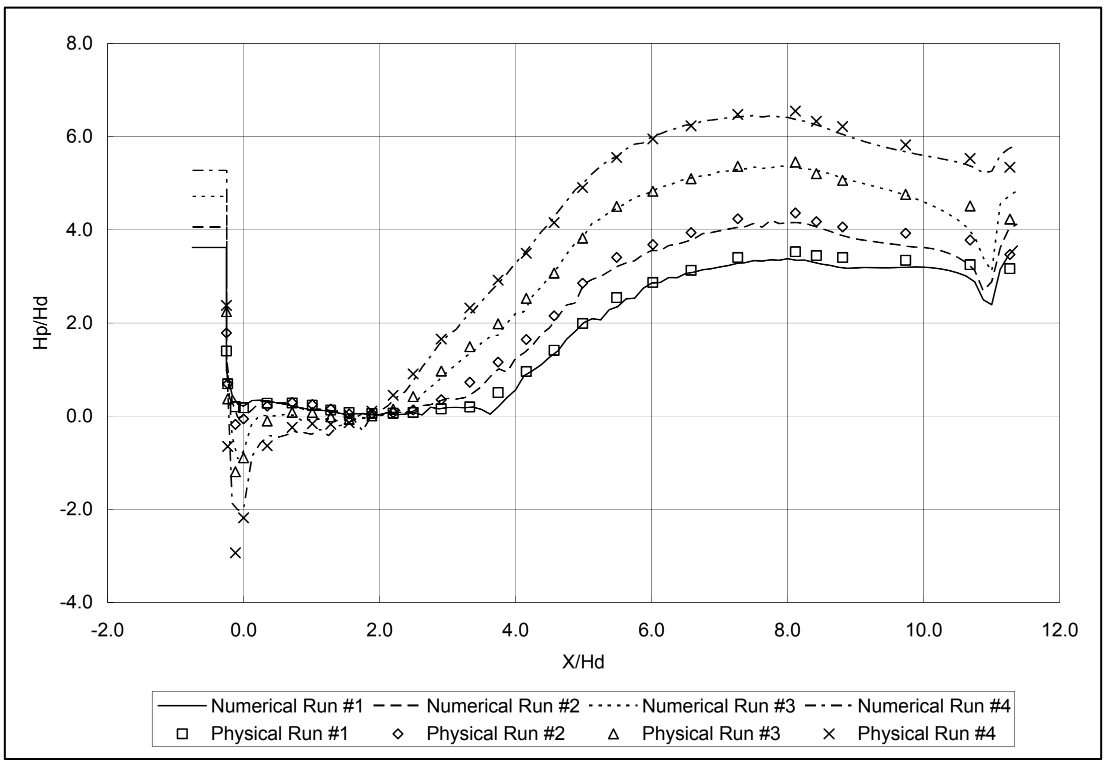
The flow rates over the spillway crest and the pressures acting on the crest are used to compare the differences between the physical models and numerical models for flows on submerged spillways. Table 2 shows the physical model measured flows and the calculated flows from the numerical model.

Table 2. Comparison of Observed Flow Rate versus Computed Flow Rate

	Model A			Model B			Model C		
Run	Physical (ft^3/s)	Numeric (ft^3/s)	% Diff	Physical (ft^3/s)	Numeric (ft^3/s)	% Diff	Physical (ft^3/s)	Numeric (ft^3/s)	% Diff
1	1.278	1.272	-0.5	1.218	1.211	0.6	0.805	0.791	1.8
2	3.143	3.244	3.2	2.938	2.949	-0.4	1.646	1.601	2.8
3	3.772	3.878	2.8	3.588	3.574	0.4	3.076	3.024	1.7
4	5.415	5.529 [c]	2.1	4.933[c]	4.955	-0.4	4.428	4.386	1.0
5	-	-	-	-	-	-	4.686	4.630	1.2
[c] Submerged flow due to high tailwater									

The pressure heads (Hp) on the spillways evaluated have been nondimensionalized by dividing them by the design head (Hd) for each spillway. The pressure position on the spillway is shown nondimensionally as X/Hd, with X being the horizontal distance from the crest axis. Fig. 4 provides a comparison of average spillway pressures for four different conditions on Model B; 0.90Hd, 1.39Hd, 2.05Hd, and 2.66Hd. Figure 5 provides a comparison of average spillway pressures for five different conditions on Model C; 0.17Hd, 0.50Hd, 0.86Hd, 0.97Hd, and 1.18Hd.

Fig. 4. Comparison of Numerical Model and Physical Model relative pressures for Model B.



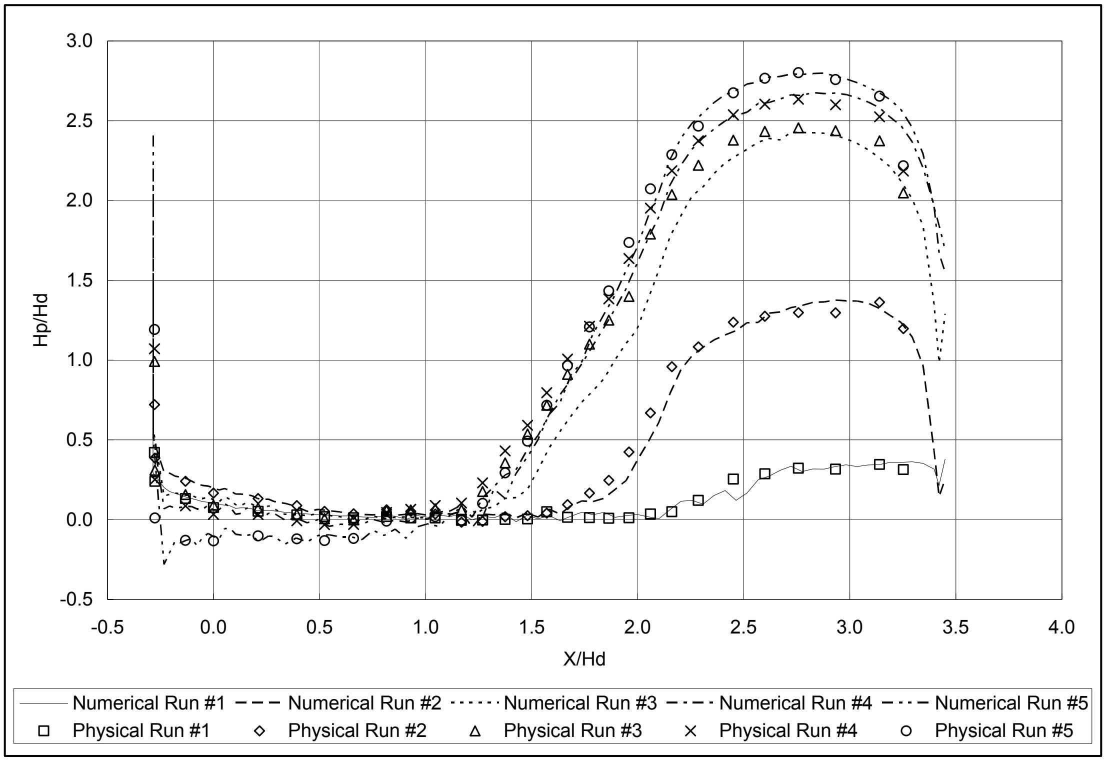


Fig. 5. Comparison of Numerical Model and Physical Model relative pressures for Model C.

Using the physical models as the basis, Fig. 6 shows the difference, in centimeters of water, between the numerical model and the physical model data at a given X/Hd position for Model B. Fig. 6 shows the difference, in centimeters of water, between the numerical model and the physical model data at a given X/Hd position for Model C. For the comparison, absolute error is used instead of a relative error because many of the crest pressures are nearly atmospheric (Hp=0). At pressures near zero, even a small difference can result in a large relative error. For example, if a pressure difference of 1 cm of water was divided by a reference pressure of 0.5 cm, a relative error of 200% would result even
though the absolute difference is relatively small when considering the size of each model tested.

Although there are no prototype pressure data available for comparison to the numerical solution, the writers scaled the data from Model C Run 5 to prototype dimensions and performed a two-dimensional numerical simulation of the prototype for such a comparison. The physical model data were scaled up 28 times and the numerical model was developed and several runs were completed to ensure grid convergence. The results of this comparison are shown on Fig. 8. Pressures from the numerical model compared quite favorably with the scaled physical model data with the exception of Taps 23–28. The greatest absolute deviation between the numerical simulation and the scaled physical model data for these taps was 1.9 m and it occurred at Tap 26. The relative error of this deviation was 7.8%.

It is interesting to note the significant differences between the numerical Model C Run 5 results and numerical simulation of the prototype structure in the vicinity of Taps 23–28. When numerically modeling the physical scale model, the results are considerably closer to those obtained by modeling the prototype and scaling the physical model to the prototype. This raises the question of is it better to model the model than to model the prototype. Of course, the answer lies in the question of why there is a difference. Additional research is being completed to look into why a difference.

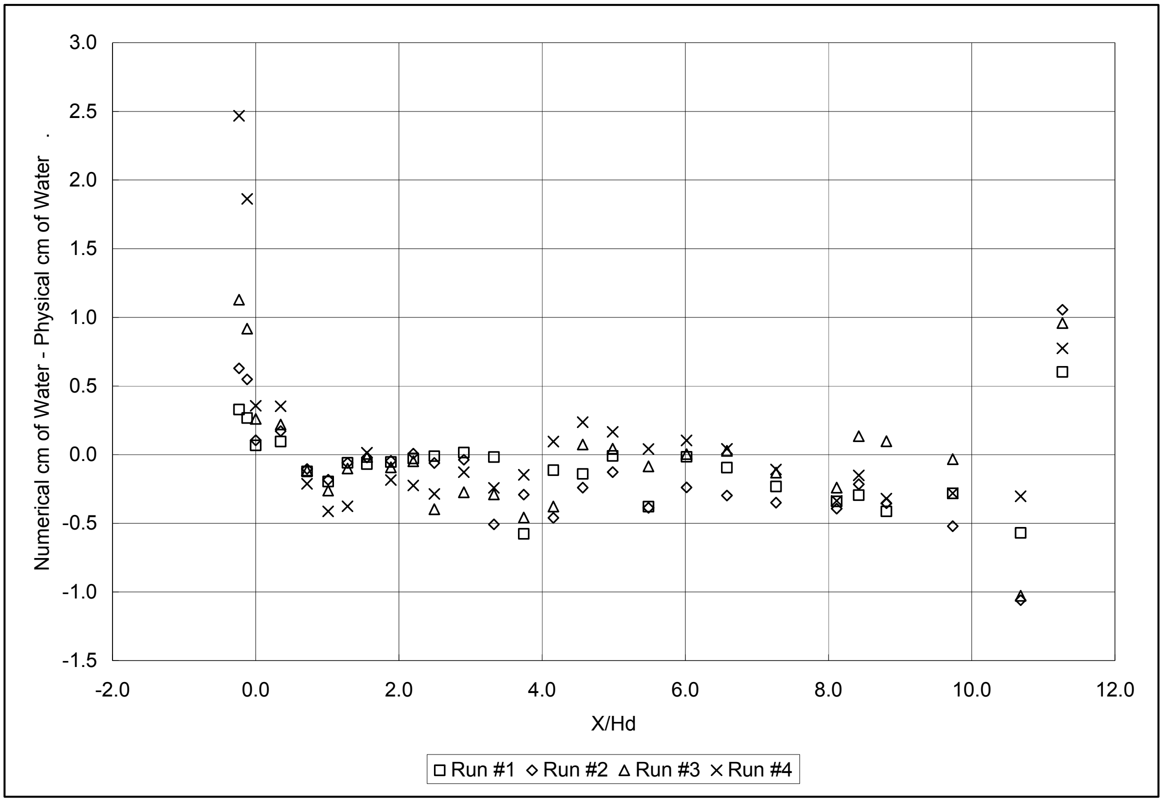


Fig. 6. Absolute pressure head differences between the Numerical Model and Physical Model for Model B.

Discussion

The flow rate results from Table 2 show that the numerical model provided a reasonable solution, even when the dam crest becomes submerged, as in the case of Model B, Run 4. The flow rates predicted by the numerical model are all within 3% of the numerical model's results for each model investigated. The data presented in Figs. 6 and 7 demonstrate that numerical modeling is capable of reasonably predicting pressures on spillways with significant tailwater. The concern of modeling supercritical flow transitioning to subcritical flow has been and continues to be a difficult problem to solve, however numerical advances are rapidly reducing the inherent difficulties of this problem.

Examination of Figs. 6 and 7 show that at worst, 2.5 cm of water of difference between the numerical and physical models. This difference was most pronounced on the vertical face of the dams and is likely due to the increased acceleration the flow experiences as it approaches the crest of the dam. In the submerged zone of flow, the differences on Model C were usually less than 2 cm of water; the majority of differences were less than 1 cm. For Model B the results were similar however the differences in the submerged zone were usually less than 0.5 cm of water. In both model cases investigated, the numerical model was robust enough to provide data that is quite accurate.

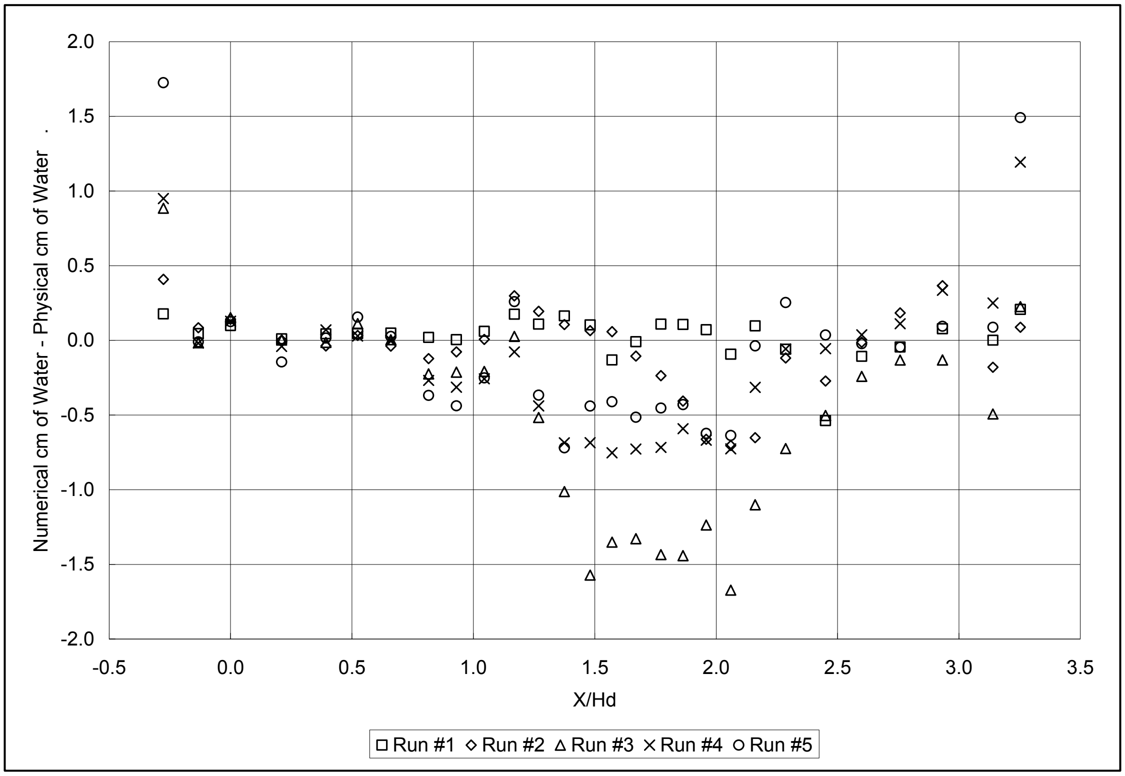


Fig. 7. Absolute pressure head differences between the Numerical Model and Physical Model for Model C.

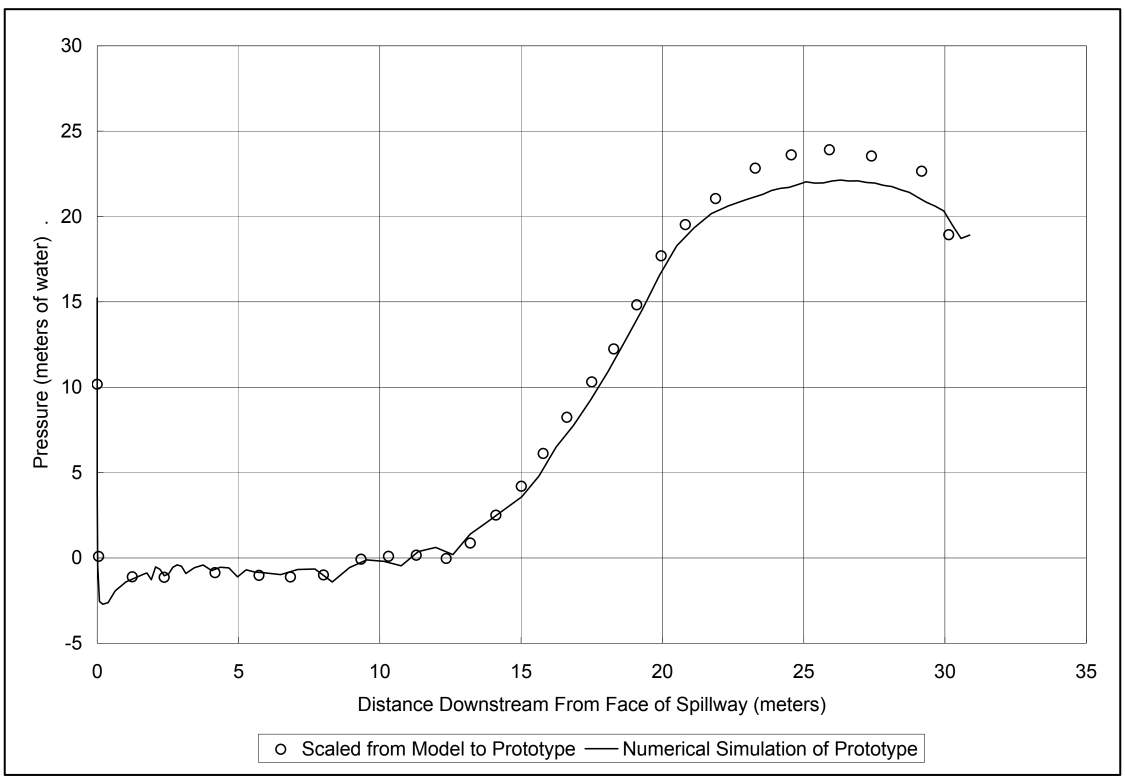


Fig. 8. Comparison of physical model data scaled to prototype dimensions and numerical simulation of prototype.

Although numerical tools still have limitations (including turbulence representation, aeration and bulking, grid resolution, run times, and numerical instabilities to name a few), there are many areas where current numerical methods may offer increased accuracy over design monographs and be sufficiently accurate for the required application. Numerical models can provide more detail about velocity and pressure distributions than can a physical model and may be more economical in some cases. In the past, engineers have had to rely on model studies and design monographs to obtain data necessary for analysis. For uncontrolled spillways with a relatively simple geometry that operate without flow separation around piers, the writers believe that a numerical model may be sufficient to rapidly obtain information necessary to complete a dam stability analysis.

References

Bradley, J. N. 1945. Studies of flow characteristics, discharge and pressures relative to submerged dams." *Hydraulic Laboratory Rep. No. 182*, Denver.

Chanel, P.G. and Doering, J.C., 2008. Assessment of Spillway Modeling Using Computational Fluid Dynamics. *Canadian Journal of Civil Engineering*, 35: 1481-1485

Gessler, D. 2005. CFD modeling of spillway performance, EWRI 2005: Impacts of global climate change. *In Proceedings of the World Water and Environmental Resources Congress*, Anchorage, Alaska, 15-19 May 2005. Edited by R. Walton. American Society of Civil Engineers, Reston, Va.

Harleman, D. R. F., Wagner, W. E., and Barnes, S. M. 1963. Bibliography on the hydraulic design of spillways. *J. Hydr. Div*., 89(4),117–139.

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 Nicholes, B. D. 1981. Volume of Fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys*., 39, 201–225.

Jansen, R.B. editor 1988. Advanced Dam Engineering for Design, Construction, and Rehabilitation. *Published by Van Nostrand Reinhold*. ISBN-13: 9780442243975.

Maynord, S. T. 1985. General spillway investigation. *Technical Rep. No. HL-85-1*, Dept. of the Army, Waterways Experiment Station, Corps of Engineers, Vicksburg, Miss.

Rahman, M., and Chaudry, H. M. 1997. Computation of flow in open channel flow. *J. Hydraul. Res*., 35(2), 243–256.

Savage, B., and Johnson, M. 2001. Flow over ogee spillway: Physical and numerical model case study. *J. Hydraul. Eng*., 127(8), 640–649.

Shany, M. B. 1950. Pressure distribution on the downstream face of a submerged weir. *Thesis, State Univ. of Iowa.*

Stelle, W. W., Rubin, D. I., and Buhac, H. J. 1983. Stability of concrete dam: Case history. *J. Energy Eng*., 109(3), 165–180.

U.S. Army Corp of Engineers (USACE). 1990. Hydraulic design of spillways. *EM 1110-2-1603*, Dept. of the Army, Washington, D.C.

U.S. Army Corps of Engineers (USACE). 1995. Gravity dam design. *EM 1110-2-2200*, Dept. of the Army, Washington, D.C.

U.S. Bureau of Reclamation (USBR). 1977/1987. Design of small dams, U.S. Government Printing Office, Washington, D.C.

Versteeg, H. K., and Malalasekera, W. 1996. An introduction to computational fluid dynamics, *Longman Scientific and Technical*, New York.

Yakhot, V., and Orszag, S. A. 1986. Renormalization group analysis of turbulence. I: Basic theory. *J.Sci.Comput*.,1(1),1–51.

Yakhot, V., and Smith, L. M. 1992. The renormalization group, the expansion and derivation of turbulence models. *J. Sci. Comput*.,7(1), 35–61.