A Hybrid Numerical and Physical Hydraulic Model Study of the Canton Dam Spillway System

Martin Wosnik[1,2], Songheng Li[1], Christopher Miller[1], Russell Wyckhoff[3], Hasan Kocahan[4], Dan Gessler[1], Stuart Cain[1]
[1]Alden Research Laboratory, Holden, MA 01520, USA
[2]Department of Mechanical Engineering, University of New Hampshire, Durham, NH 03824, USA
[3]U.S. Army Corps of Engineers, Tulsa District, Tulsa, OK 74128, USA
[4]Hydroplus Inc., Falls Church, VA 22041, USA

Abstract
The existing flood control dam at Canton, OK, is being upgraded with an auxiliary spillway to enable it to safely pass the new Probable Maximum Flood (PMF). The auxiliary spillway weir will be equipped with Fusegates, which will tip individually at predetermined water elevations to release flood water as needed. The service and auxiliary spillways together must be able to discharge a PMF of 17,000m^3/s without overtopping the dam. To facilitate this, a hybrid numerical and physical hydraulic model study of the spillway system was conducted at Alden Research Laboratory.

First, a numerical model study was carried out for various approach geometry designs to investigate approach flow patterns, resulting water surface elevations throughout the reservoir and spillways, as well as flow rate splits between the two spillways. Based on the CFD results, a favorable design was selected, constructed and tested in a large-scale 1:54 scale topographic physical model. The advantage of this hybrid, integrated numerical and physical modeling approach is that each model can be used where it has its strengths: Numerous modifications of the approach channel geometry were made in a cost-effective way in the numerical model. The large-scale physical model was then used to validate the numerical results, for final modifications that brought the maximum reservoir elevation at PMF to within acceptable levels, to obtain the spillway rating curves and for Fusegate-specific tests.

Introduction
Hydrological information was comparatively limited when dams were designed and built in the first half of the 20th century. Throughout the 1960s, spillways were sized using the Spillway Design Flood (SDF), which is calculated by transposing an actual storm that occurred nearby and centering it over the reservoir under consideration. Since the 1970s, the SDF criteria has been replaced by the probable maximum flood (PMF), which is the flood that may be expected from the most severe combination of critical meteorological and hydrological conditions that are reasonably possible in the region. As more long-term hydrological data is gathered and processed, the inflow design flood (IDF) used for many existing dams and spillways is being reviewed, usually resulting in an increase and creating new dam safety challenges. Under an increased IDF, insufficient discharge capacity or undesirable performance of spillways can potentially result in dam failure.

Physical scale models have been used to simulate and study the behavior of hydraulic structures and projects for over 100 years. Physical modeling is a mature, proven tool and its results are used with high confidence. With the availability of powerful computational fluid dynamics (CFD) codes and advances in computer technology, numerical modeling/CFD can now be effectively included in the design process of hydraulic structures. One of the primary functions of either type of modeling is to test design alternatives. Which of the two tools is faster and cheaper depends on the specific project, the extent of modifications between design alternatives and the answers sought. Some modeling requirements cannot be met by CFD at this stage, or would be prohibitively costly, such as simulating the tipping of Fusegates at predetermined water levels, navigation or air entrainment due to free surface vortices. In many situations numerical models require validation by physical models, for example when evaluating complex hydraulic conditions or a non-standard design, or when there is high risk such as in dam safety.

Often a hybrid, integrated approach that combines physical and CFD modeling is the most effective. CFD modeling is typically employed early in the modeling process to evaluate design alternatives. The physical model is then used to validate the CFD results, establish experimental spillway rating curves and conduct aspects of a hydraulic model study that would not be cost-effective in a numerical model. While CFD modeling after careful validation can be a cost-effective and reliable tool, it cannot be considered a complete replacement of physical modeling. Hydrodynamic characteristics of flow over spillways have been systematically investigated since the late 1950s at the US Army Corps of Engineers (USACE) Waterways Experiment Station (WES) using physical models (USACE WES 1977). To this day, hydraulic engineers still benefit from a series of design charts developed from physical models of spillway designs. More recently, researchers have begun to use CFD for simulations of spillway flows. (for example Savage and Johnson 2001, Ho et al 2006, Chanel and Doering 2008, Song and Zhou 1999, Cagri et al. 2008, Lee et al. 2008).

Canton Dam Spillway System

Canton Dam is located in western Oklahoma on the North Canadian River, approximately 75 miles northwest of Oklahoma City. It was constructed under the Flood Control Act and completed in 1948. Canton Dam was built with one service spillway equipped with 16 Tainter gates and three outlet works with a total discharge capacity of 9,970m^3/s, which is inadequate to safely pass the revised reservoir inflow PMF of 17,950m^3/s (Figure 1). The US Army Corps of Engineers (USACE) decided to add a 146.3 m wide auxiliary spillway, equipped with a broad-crested weir on which 9 Fusegates will be placed. Fusegates, designed by Hydroplus, tip individually when a chamber underneath each block is flooded at a predetermined water elevation and release flood water as needed. The operating principle is illustrated in Figure 2. In order to improve the design of the auxiliary spillway, an integrated model study approach by combining CFD and a physical model was proposed by Alden.

First, a three-dimensional CFD model was developed for the base design, and a PMF simulation was conducted to predict flow patterns, water surface elevations throughout the reservoir, and flow rates through the existing and auxiliary spillways. Then, a series of design modifications were made and simulated, with the goal of lowering the reservoir water surface during a PMF event. Based on the CFD results, a favorable design was selected, constructed and tested in a 1:54 scale physical model. The physical model test results were then used to validate the CFD model.

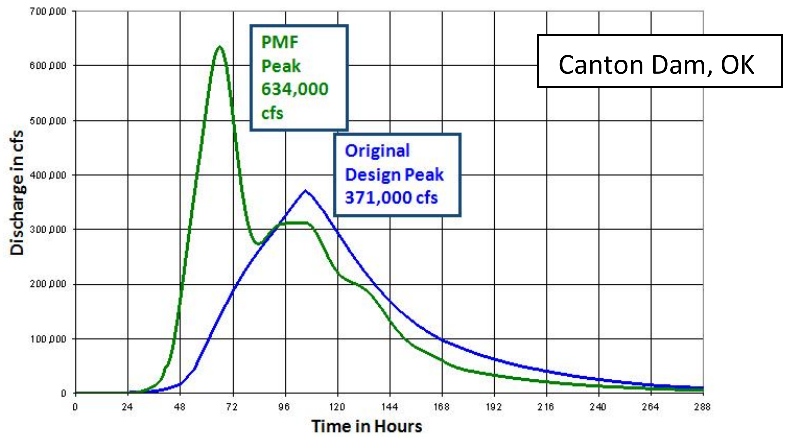

Figure 1. Canton Dam inflow hydrograph with original inflow design flood (10,500m^3/s), and the new reservoir inflow PMF (17,950m^3/s).

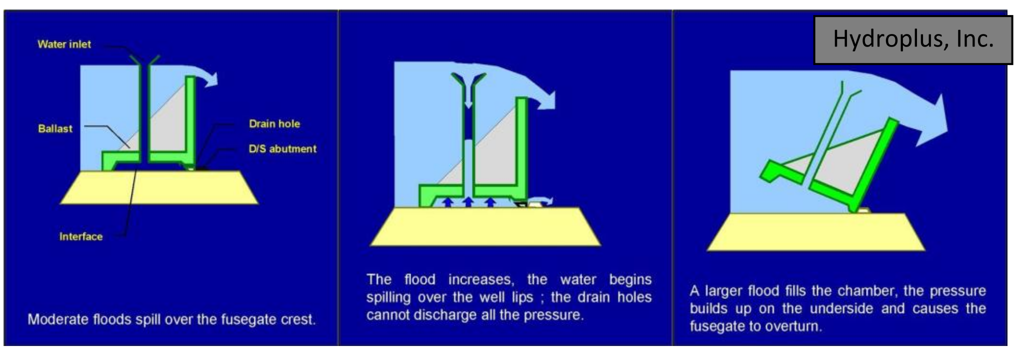

Figure 2. Operating principle of Fusegates (courtesy of Hydroplus).

Numerical Model

The numerical simulations were performed by solving the unsteady Reynolds-averaged Navier-Stokes (U-RANS) equations. The standard k-ε turbulence model (Launder and Spalding 1974) with wall functions was used for turbulence closure. The interface between the air and water was tracked using the volume of fluid (VOF) model, which is addressed with a separate water volume fraction equation. Figure 3 shows the configuration of the CFD model. The CFD model domain was extended as far upstream as bathymetry data, provided by USACE, was available. The downstream ends of the spillways were simplified as rectangular or trapezoidal open channels. The Fusegates were not included in the PMF simulation since they will all have tipped over and washed away by the time the reservoir outflow reaches its maximum. High mesh resolution was used in areas of high velocity gradients at the spillways, where the flow accelerates and transitions from subcritical to supercritical. Proper mesh topology and density were used, especially in areas of importance to flow patterns, such as a partially submerged berm and flow guiding walls on the left side of the existing service spillway, and in areas of importance to flow splitting, such as the newly created island between the two spillways.

The PMF reservoir outflow was prescribed as the CFD model's inflow boundary condition, and the discharge flow rates for the two spillways were established by the simulation. Eight cases representing a combination of six different geometries were simulated. The design alternatives focused on three areas: Streamlining of the right wall of the approach channel (Mod1), shaping the island created between the two spillways (Mod2) and excavation of a partially submerged berm and placement of flow-guiding walls on the left side of the existing service spillway (Mod3). The different approach channel design alternatives are shown in Figure 4.

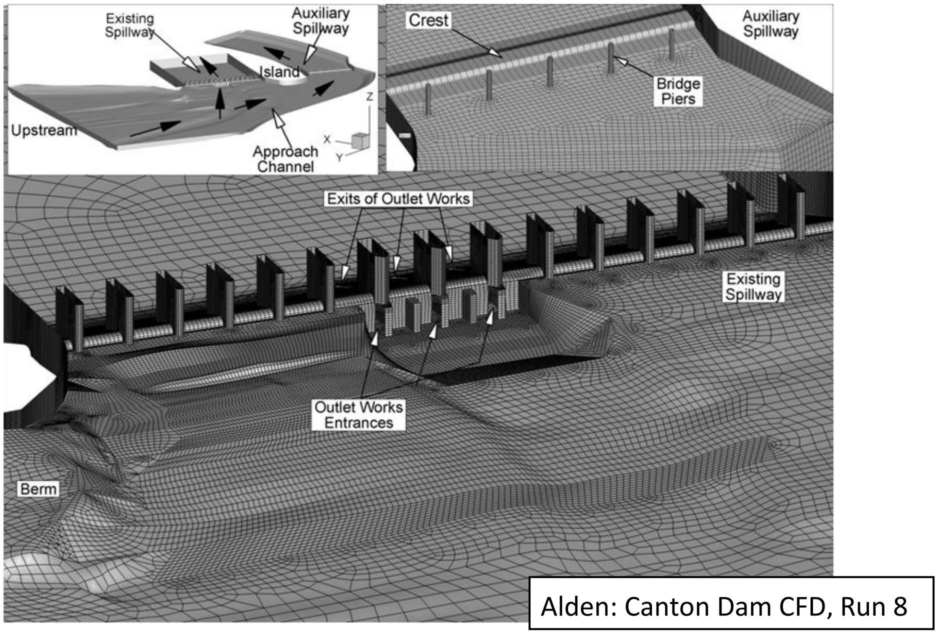

Figure 3: Configuration of the Canton Dam CFD Model. Upper left corner: overview of entire model area; upper right: mesh detail on the solid boundary of the auxiliary spillway, including broad-crested weir; main plot area: mesh detail in the existing service spillway area (Run 8 geometry is shown).

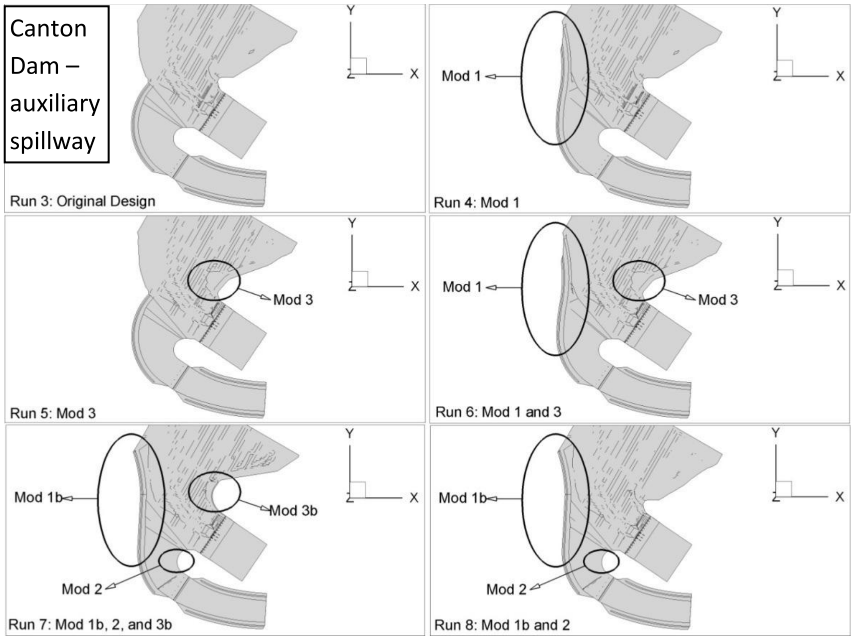

Figure 4: The different approach channel design alternatives studied with CFD.

Major flow patterns were revealed. In the original design, flow separated off the sharp corner on the right bank of the auxiliary spillway and a large recirculation zone formed. An area of flow separation with a strong eddy existed downstream of the partially submerged berm, upstream of the left-most bays of the service spillway. The water level was higher at the right bays than at the left bays of the service spillway. This is consistent with observations in the original physical model study (USWES, 1942), however, the increased PMF and the auxiliary spillway create higher approach velocities and exacerbate this problem. Run 7 significantly improved the flow patterns in the auxiliary spillway approach channel compared to the original design and produced the lowest water surface elevation at the inflow boundary. Hence it was the most favorable design. However, it was decided to forgo the excavation of the partially submerged berm (Mod3 or 3b) for now due to cost, and keep the left side of the service spillway as it exists in the field. Therefore, the final CFD simulation (Run 8) was performed with modifications Mod 1b and Mod 2. This was the configuration to be constructed and tested in the physical model (Physical Model: Build 1).

Physical Model

The physical model was constructed as an undistorted, Froude-scaled 1:54 scale topographic model. The model boundaries were selected to represent the reservoir approximately 800 m upstream and 1100 m downstream of the service spillway. The model footprint was 34 m by 21 m and the maximum model flow rate was approximately 0.83m^3/s. The general model layout is shown in Figure 5.

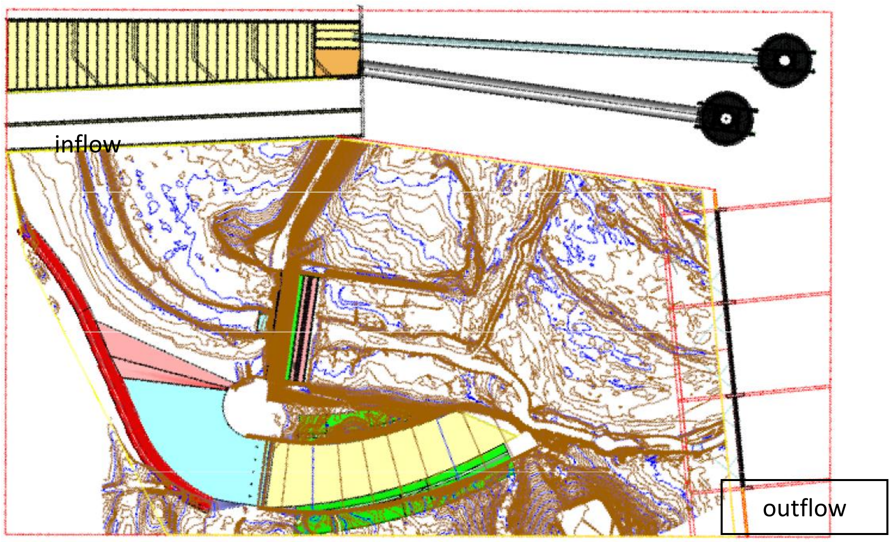

Figure 5: Model layout in 34 x 21 m high-flow topographic facility at Alden.

A physically likely approach flow distribution based on reservoir topography had been chosen as the numerical model inflow velocity profile, and the physical model inflow distribution was carefully adjusted to match it. The different inflow boundaries and the plane for numerical/physical model comparison are shown in Figure 6. Water surface elevations measured at reservoir tap locations (Taps 1-4) in the physical model are compared to the numerical model elevations in Table 1 and match closely. More variation is seen between computed and measured average velocities, partly due to the increased uncertainty when scaling up the model velocity measurement. Water surface elevations for all upstream tap locations are shown in Figure 7 and show good agreement. Flow patterns observed in the physical model also matched the flow patterns observed in the CFD model, cf. Figure 8. The physical model results thus validated the predictions of the CFD Run, and thereby they also validated the CFD model and methodology in general.

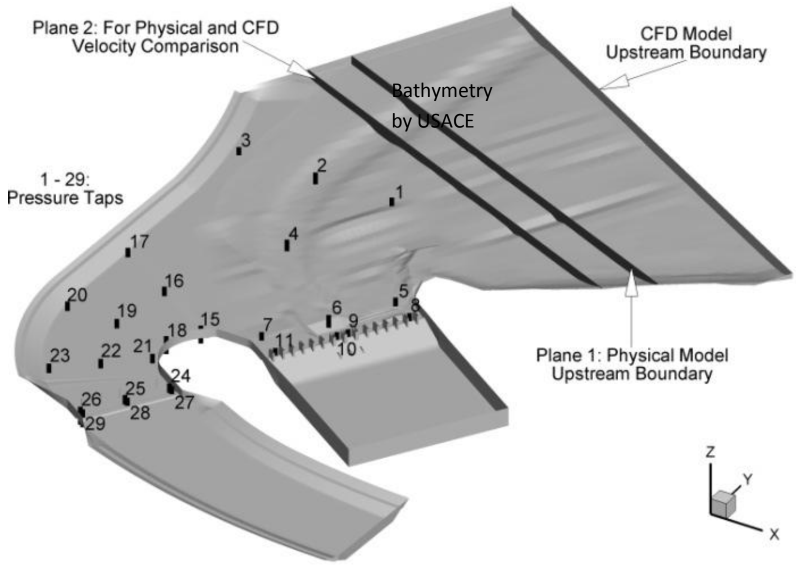

Figure 6: Upstream extent of numerical and physical models, model plane for inflow velocity comparison and locations of upstream piezometric taps in the physical model.

Tap	Location (m)		Water Surface Elevation (m)			Velocity (m/s)		
	Easting	Northing	CFD	Measured	Difference	CFD	Measured	Difference
1	545645.3	120496.9	500.73	500.76	0.03	4.15	4.33	3.9%
2	545527.4	120498.7	500.82	500.79	0.03	3.44	3.81	9.7%
3	545409.4	120500.8	500.88	500.94	0.06	3.05	3.02	1.7%
4	545577.7	120338.7	500.30	500.24	0.06	4.66	4.72	1.4%

Table 1: Comparison of water surface elevation and velocities in reservoir.

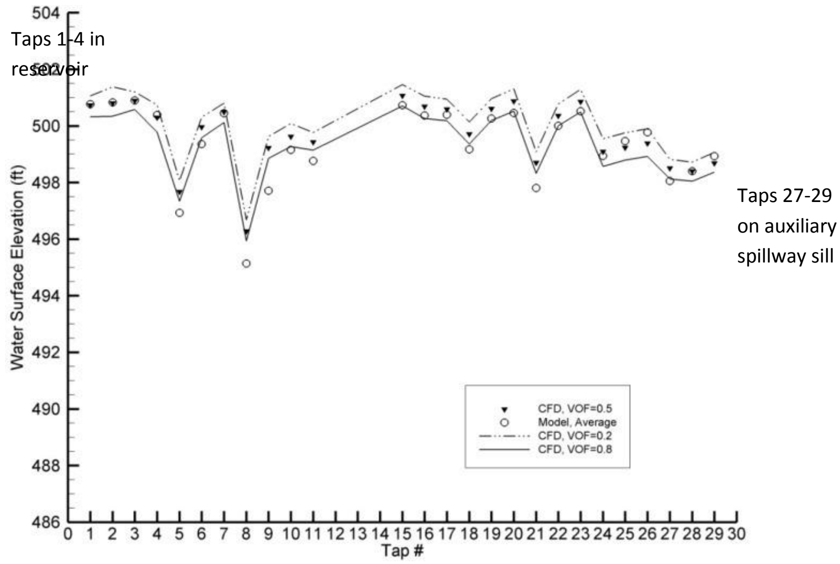

Figure 7: Comparison of water surface elevations between CFD and physical models.

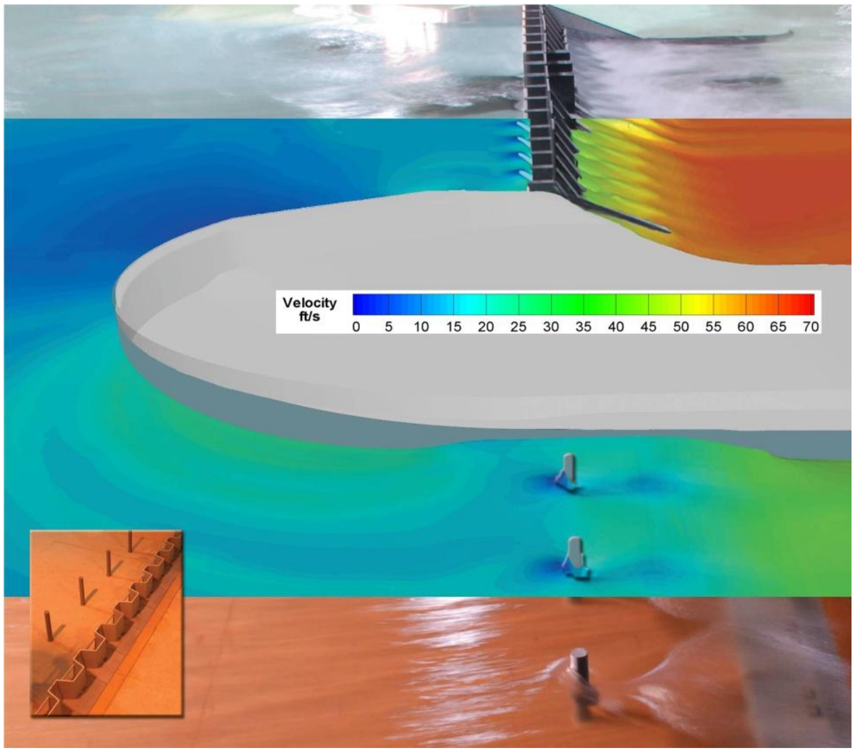

Figure 8: Overlay of numerical model flow patterns of Run 8 with photograph of physical model Build 1 at PMF. Inset: 9 Fusegates on broad-crested weir (dry), downstream of bridge piers.

For accurate water surface elevations and rating curves, measurements were performed in the physical model. Rating curves were obtained for each spillway separately, and for combined spillway operation.

The latter showed that the still pool elevation at PMF was still unacceptably high. Relying on the comparative data from the CFD model and data from the physical model, it was determined that an acceptable reservoir water surface elevation could be reached through several modifications in the physical model (Physical Model: Build 2). These included removal of the partially submerged berm, Mod 3, lowering the auxiliary spillway sill by 1.2m while increasing Fusegate crest height by 0.6m. The Fusegate tipping sequence was also modified to lower the maximum reservoir PMF outflow to 17,100m^3/s. The maximum pool elevation for the PMF was thus reduced to an acceptable, safe elevation. The rating curves for combined spillway operation for both model builds are shown in Figure 9.

The physical model was then used to evaluate Fusegate tipping order and evacuation through the return channel, to select the intake conduit entrance location, and to determine the intake well operating elevations. Performing these tasks in a CFD model would have been difficult and prohibitively costly. In addition, since the emergency spillway and Fusegate system are intended to control flooding and limit potential loss of life, CFD would have to be validated in a physical model anyway.

Summary

A hybrid numerical and physical hydraulic model study of the Canton Dam spillway system was conducted at Alden Research Laboratory to facilitate the addition of an auxiliary spillway, which is required to safely pass the new Probable Maximum Flood (PMF). Each model was used where it has its strengths: Numerous modifications of the approach channel geometry were made cost-effectively in the numerical model. The large-scale physical model was used to validate the numerical results, for final modifications that brought the maximum reservoir elevation at PMF to within acceptable levels, to obtain the spillway rating curves and for Fusegate-specific tests.

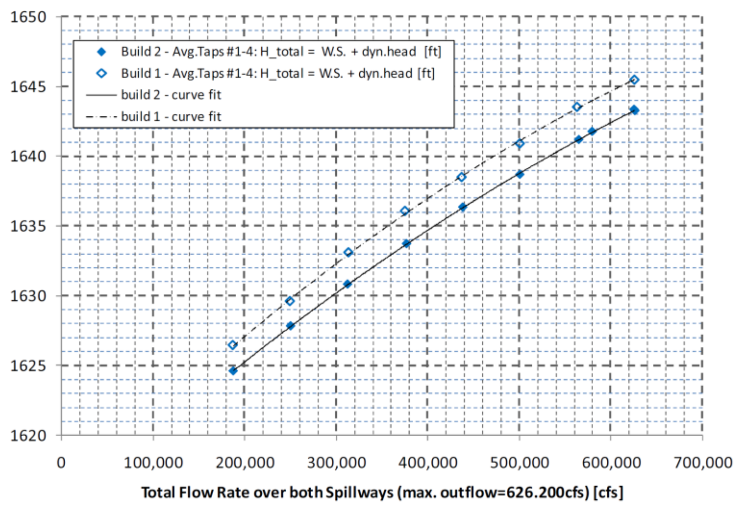

Figure 9: Rating curves for combined spillway operation (existing service spillway and proposed auxiliary spillways) at Canton Dam [1ft=0.305m, 1000cfs=28.3m^3/s].

References

Chanel P; Doering J (2008) "Assessment of spillway modeling using computational fluid dynamics." *Can. J. Civ. Eng*., 35, 1481-1485.

Ettema R; Arndt REA; Roberts P; Wahl T (2000) Hydraulic Modeling: Concepts and Practice. ASCE Manuals and Reports on Engineering Practice No. 97.

Ho D; Cooper B; Riddtte K; Donohoo S (2006) "Appl. of numerical modeling to spillways in Australia." In *Dams and Reservoirs*. eds. Berga et al.,Taylor and Francis.

Launder B; Spalding D (1974) "The numerical computation of turbulent flows." Computer Methods in Applied Mechanics and Engineering, 3, 269-289.

Lee H; Lin C; Weber LJ (2008) "Application of a non-hydrostatic model to flow in a free surface fish passage facility." *J. Hydraul. Eng.,* 134(7), 993-999.

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

Song C; Zhou F (1999) "Simulation of free surface flow over a spillway." *J. Hydraul. Eng.,* 125(9), 959-967.

US Army Corps Engineers, Waterways Experiment Station (1977). "Corps of Engineers Hydraulic Design Criteria", Volume 2.

U.S. Waterways Experiment Station (USWES) (1942). "Model Studies of the Spillway and Integral Sluices for the Canton Dam, North Canadian River, Oklahoma" U.S. Army, Technical Memorandum No. 190-1, Vicksburg, Miss.