Free-Surface Computational Fluid Dynamics Modeling of a Spillway and Tailrace: Case Study of The Dalles Project

Christopher B. Cook[1], Marshall C. Richmond[1],
John A. Serkowski[1], and Laurie L. Ebner[2]

[1]Pacific Northwest National Laboratory
[2]Portland District, US Army Corps of Engineers

ABSTRACT

Based upon acoustic tracking and fish tagging data, The Dalles Project has been shown to have one of the highest mortality rates for juvenile salmonids on the Lower Columbia River. In efforts to assist the hydraulic and biological communities in managing this hydroelectric project, a three-dimensional computational fluid dynamics (CFD) model was applied to the spillway, stilling basin, and tailrace zones downstream of the dam.

To simulate the highly transient and turbulent flow conditions in this region, a free-surface computational fluid dynamics (CFD) numerical model has been applied. This model is based on the volume-of-fluid (VOF) method, and is capable of simulating sudden discontinuities in the free surface, including wave breakup. The model solves the non-hydrostatic Reynolds-averaged Navier-Stokes (RANS) equations over variable-sized hexahedral cells.

To validate the ability of the numerical model to simulate flows downstream of the spillway, the model was validated against data from three different physical models at scales of 1:36, 1:40, and 1:80. Results from these physical models allow for validation of the numerical model at various scales of motion from the small scale highly dynamic variations near the baffle blocks (1:36 and 1:40 scale) to the larger scale general circulation patterns that encompass the tailrace (1:80 scale).

INTRODUCTION

The US Army Corps of Engineers constructed and operates The Dalles Project (TDA). The unusual "L" shaped footprint of this run-of-the-river hydroelectric dam was constructed to take advantage of bathymetric features in the area. Before construction of the dam, a series of waterfalls carved out deep holes in this region. The spillway of the dam sits primarily upon a flat shelf of basalt, which downstream of the dam is at elevation 68 ft. Deep holes downstream of the spillway reach depths much below sea level, with the deepest reaching elevations below –200 ft.

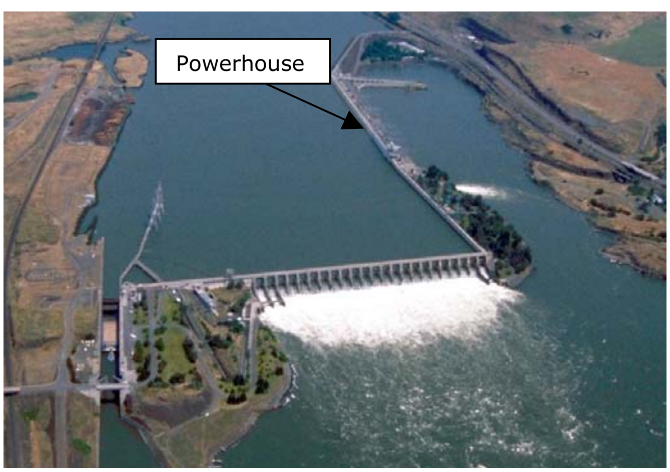


Figure 1 Aerial view of The Dalles Project. The figure shows the powerhouse and the spillway (with all bays operating). Oregon is on the right, and Washington is on the left side of the Columbia River.

Numerical modeling of this region is further complicated by the turbulent nature of the stilling basin downstream of the spillway. It is suspected that this region may be lethal to juvenile fish that are migrating downstream to the Pacific Ocean. It has recently been theorized that the baffle blocks and end sill may harm the fish, although this theory has not yet been verified with data from the field.

To simulate flows in the complex bathymetry of The Dalles tailrace and to better understand forces on the fish in the stilling basin, the commercially available CFD model FLOW-3D was applied to the region. The model uses the finite-volume method to solve the RANS equations over the computation domain. For each cell, values of the state variables were solved at discrete times using a staggered grid technique (Versteeg and Malalasekera, 1995). Tracking of the free-surface is performed using the Volume-of-Fluid (VOF) method described by Hirt and Nichols (1981), which produces a surface that is free of the "stair-stepping" effect normally associated with Cartesian hexahedral grids.

The full domain of the numerical model extends from in front of the powerhouse to more than 4200 ft downstream of the spillway (see Figure 2). It was constructed based on engineering drawings of the hydraulic structures and several bathymetric surveys of the tailrace. The total longitudinal length of the model (both blocks) is 9100 ft, and the lateral length of the spillway tailrace block is 4200 ft. Once the domain was constructed, it served as the foundation for the validation tests.

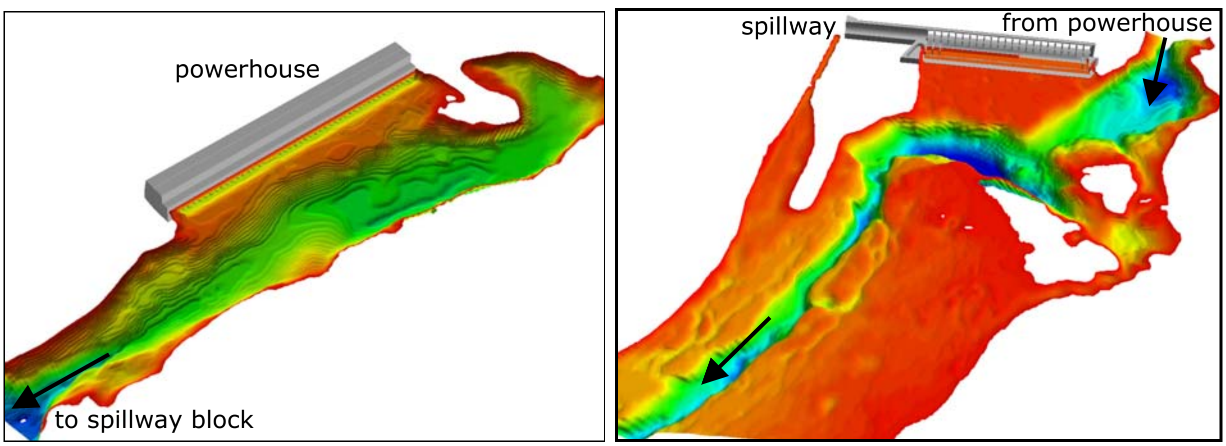


Figure 2 Overview of Powerhouse Tailrace Block (left) and the Spillway Tailrace Block (right). Contours are from elevation –160 ft (dark blue) to 76.8 ft (red). The black arrows denote the general direction of flow. Tailrace bathymetry was cropped at the water surface elevation (76.8 ft – water surface elevation of the 1:80 validation test case) for clarity.

STILLING BASIN VELOCITY VALIDATION USING 1:40 SCALE PHYSICAL MODEL DATA

A 1:40 scale sectional physical model of The Dalles spillway was constructed at the Engineer Research and Development Center (ERDC), US Army Corps of Engineers. The physical model consists of 3.5 bays that are symmetric, and represent a slice out of the prototype spillway. Since flow is only able to enter the physical model from the forebay (i.e. there is no lateral flow, unlike the prototype), and to increase computation efficiency, the CFD model domain was reduced to a single spillway bay as shown in Figure 3. CFD model boundary conditions along the left and right sides of the domain were set as symmetry planes (no velocity gradients normal to the plane) to replicate the assumed symmetry of the physical model. The forebay elevation was held steady at elevation 160 ft and the radial gate was open 3 ft, resulting in a discharge of 5850 cfs (note: this is replicating the reported physical model discharge and may not be representative of the prototype discharge for a similar gate opening). The model domain was decomposed into three nested blocks, each with its own grid resolution: upstream forebay at 1.6 ft (prototype scale), fine grid around the gate opening at 0.8 ft, and the tailrace section at 2 ft. Although each block for this particular simulation has a uniform grid resolution, this is not a requirement of the model.

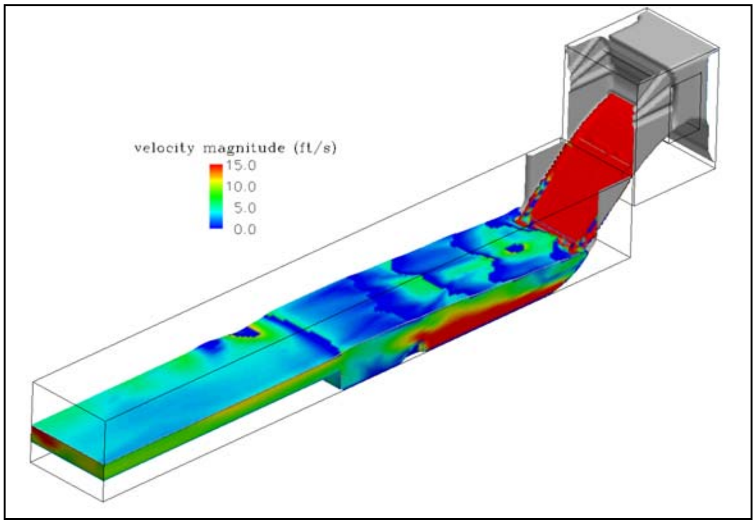


Figure 3 Orthogonal view of the CFD flume. Fluid has been colored by velocity magnitude and any velocities greater than 15 ft/s will display as red.

Figure 4 displays an elevation view of the downstream tailrace section of the model. Velocity contours have been placed along a longitudinal slice through the fluid. The longitudinal slice shown in the upper part of the figure does not intersect the baffle block, and corresponds to where data was collected in the physical model. The longitudinal slice shown in the lower part of the figure intersects a baffle block. As the flow intersects the baffle block, the bulk of the flow is guided into the upper portion of the water column, which results in a larger quiescent zone behind the block.

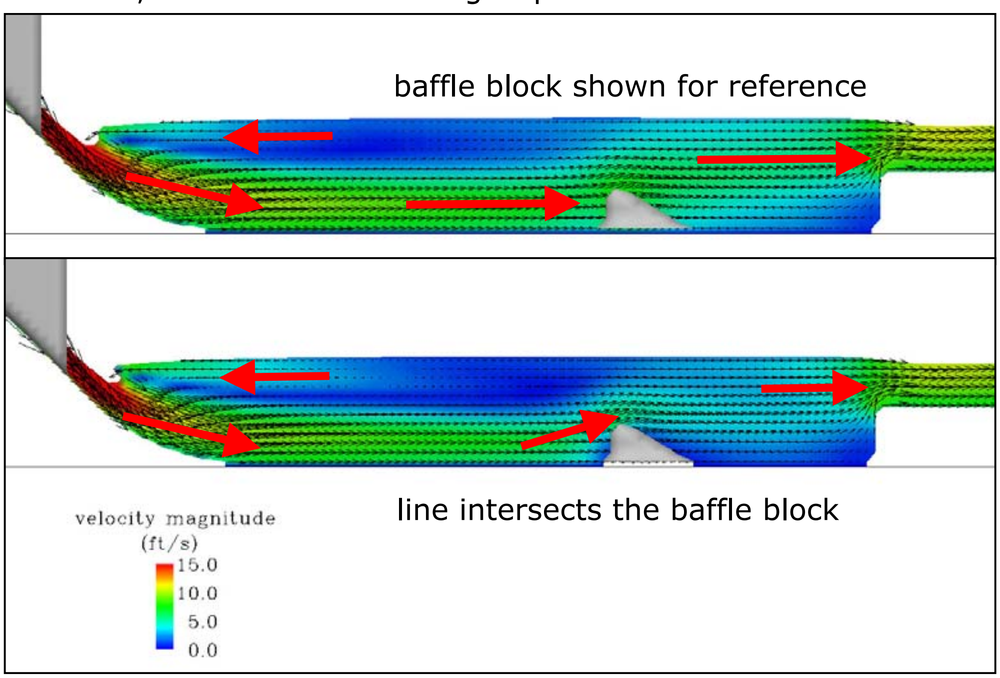


Figure 4 Velocity vectors and contours of magnitude. The upper figure is for a longitudinal line that passes between a baffle block, while the lower figure is for a line that intersects a block. The solid red arrows have been added to clarify the velocity field.

Comparisons were performed between physical model [Preslan and Wilhelms, 2001] and CFD model results. Physical model data were gathered at four locations: 7 ft in front of the baffles, 9 ft in front of the end sill, 51 ft past the end sill, and 111 ft past the end sill (Figure 5).

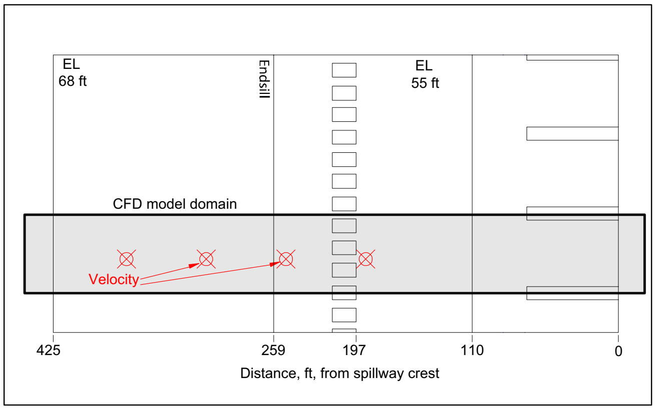


Figure 5 Plan view of the physical model flume. CFD model domain was simplified to a single bay (shown in gray), although the CFD domain extends further into the forebay and tailrace than shown in this figure.

Graphical and numerical comparisons of horizontal velocity component results are presented in Figure 6 and Table 1. In addition to time-averaged mean horizontal water velocities, standard deviations (σ) were reported at each physical model measurement location. Using a threshold of ±σ about the mean, 27% (6/22) of the CFD velocities fall outside of the observed physical model results. If the threshold is raised to ±2σ of the mean, 4.5% (1/22) fall outside of the observed physical model results.

At 190 ft, CFD and physical model profiles roughly agree in shape, although the CFD model profile appears to be slightly less than the physical model means. All CFD model results fall within one standard deviation at this location however, so differences may be due to the transient nature of the flow field (both the physical and CFD models). Differences may also be due to slight differences in discharge (gate opening) between the physical and CFD models.

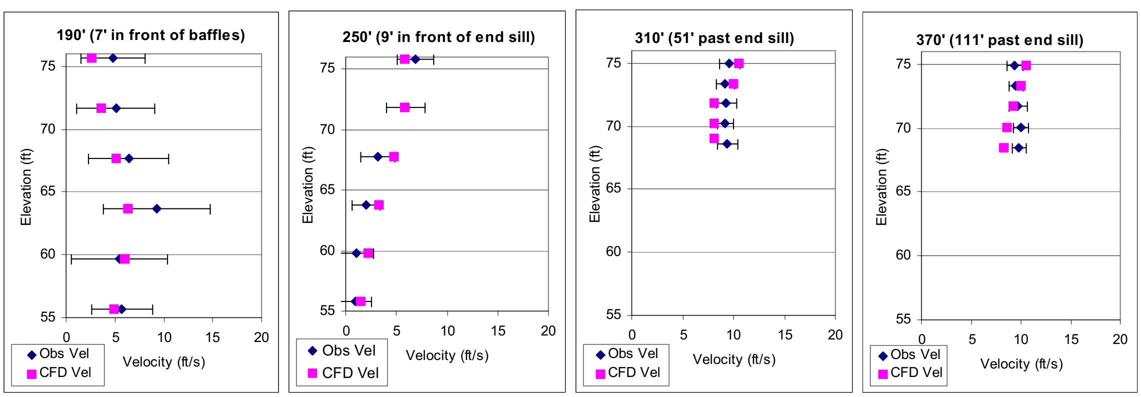


Figure 6 Velocity comparisons at four locations. Observed mean data have been plotted with blue squares and with bars representing one standard deviation from the mean.

At a distance of 310 and 370 ft from the spillway crest the velocity distribution observed in the physical model displays a more uniform trend over the water column than in the CFD model. The largest difference between the two data sets occurs at 370 ft. The bottom velocity measurement at this location is at elevation 68.5 ft, or 0.5 ft off the bottom. At 1:40 scale, 0.5 ft is approximately 1/6th of an inch. Differences between the two models this close to the bottom may be due to a number of factors including: boundary layer influences caused by the velocity probe in the physical model, insufficient grid refinement in the CFD model, and/or turbulence and wall functions used to approximate the boundary layer in the CFD model. If the errors are due to the CFD model's approximation of the boundary layer, this impact should be diminished when the CFD model is applied at prototype scale.

<u>STILLING BASIN PRESSURE VALIDATION USING 1:36 SCALE PHYSICAL MODEL DATA</u>

The Bonneville Hydraulic Laboratory, US Army Corps of Engineers, constructed a 1:36 scale sectional (three bay) model of a spillway for use in designing The Dalles project [BHL, 1952]. Measurements in the model consisted of collecting pressure heads around a single baffle block and along the end sill.

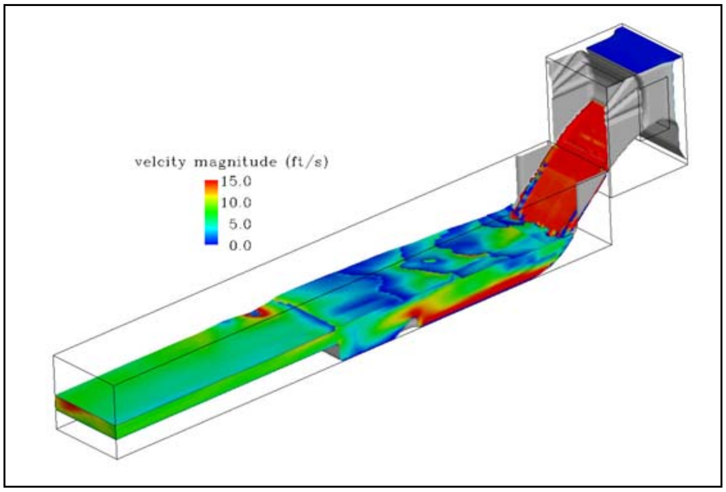


Figure 7 Perspective view of the CFD model. Water surface has been contoured by total water velocity magnitude.

Data was collected for a matrix of operational conditions. Prototype scale discharge through the physical model ranged from a minimum of 100 kcfs to a maximum flood of 2,290 kcfs (total flow for 22 bays). Since typical spillway flows for anadromous fish migration are generally in the range of 100 kcfs, only the 100 kcfs test case was examined in the numerical model.

Conditions in the physical model for the test case were a uniform discharge of 5000 cfs per bay, an upstream forebay elevation of 160 ft, and a tailwater elevation of 76.8 ft. Because the discharge from each bay was uniform, pressure taps were only placed in a single baffle block and one section of the end sill behind the baffle block (i.e. three-dimensional variations in the spillway were not measured). Because of this assumption in the physical model, the numerical model domain was only developed for a single bay.

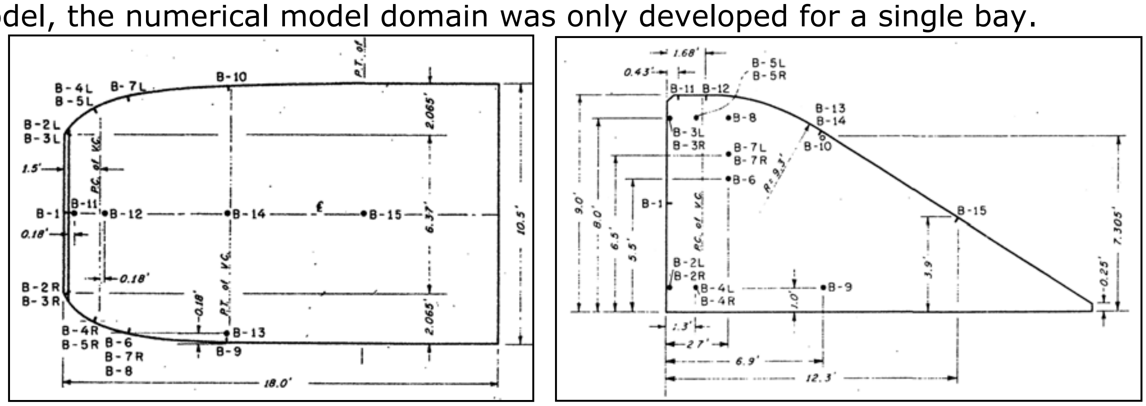
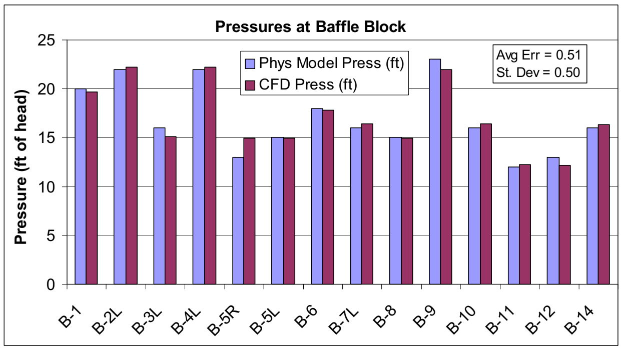


Figure 8 Plan (left) and side (right) view of measurement locations around the baffle block in the 1:36 scale physical model. Note: the "B" prefix signifies that these measurements are located on the baffle block.

The CFD model was constructed with grid sizes similar to the 1:40 scale model discussed above. The numerical model results were sampled at locations corresponding to piezometer locations in the physical model (see Figure 8). Average pressure differences between the CFD and physical model data were 0.51 ft over the baffle block (see Figure 9) and 0.40 ft over the end sill. It should be noted that pressures (and hence errors) are in prototype scale feet; a difference of 0.5 feet at

1:36 scale is 0.014 ft (0.006 psi). An error difference of this amount is probably within the accuracy of the measurement devices, although BHL (1952) did not document the expected error range of the pressure measurements.



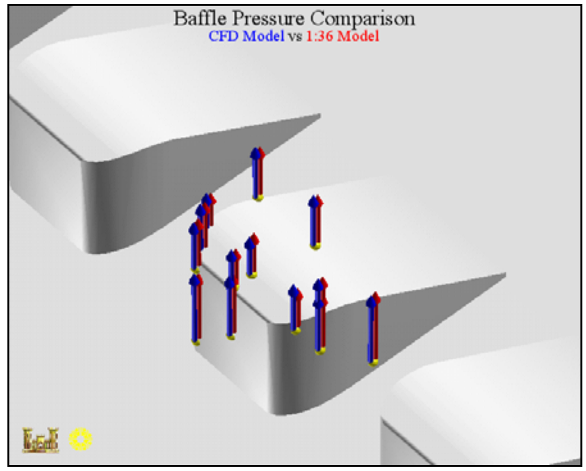


Figure 9 Comparison of observed and numerically modeled data in the 1:36 scale models around the baffle block. The chart on the left shows differences between the two data sets. Locations are shown in Figure 8. The graphic on the right shows these pressure data as arrows with yellow spheres representing the data measurement location.

<u>TAILRACE VALIDATION USING 1:80 SCALE PHYSICAL MODEL DATA</u>

The model was validated against data collected in a 1:80 scale physical model of The Dalles Project. The physical model is located at the US Army Corps of Engineers Engineer Research and Development Center (ERDC).

Spillway Bay	Discharge (kcfs)
1	3.0
2	3.0
3	4.5
4	4.5
5	4.5
6	4.5
7	4.5
8	4.5
9	4.5
10	4.5
11	3.0
12	3.0
13	3.0
14-23	0.0
Total Spill	51.0

Total River	162.9 kcfs
Tailwater Elev	76.8 ft

Powerhouse Unit	Discharge (kcfs)
1	13.0
2	0.0
3	12.7
4	0.0
5	12.4
6	0.0
7	12.3
8	0.0
9	12.3
10	0.0
11	12.3
12	0.0
13	12.3
14	0.0
15	12.3
16	0.0
17	12.3
18-22	0.0
Total Powerhouse	111.9

Table 1 Operational condition for the 1:80 scale model test. Davis (2001) labeled these project operations as Flow 1.

Operational conditions for the physical model were derived from TDA Project operations between May 21-25, 2001. The general flow conditions for this historical period were approximated by a synthetic condition that approximated powerhouse, spillway, and tailrace conditions, and are presented in Table 1. The CFD model neglected several small discharges into the tailrace to simplify input boundary conditions. The total discharge neglected by the CFD model was 7.1 kcfs or 4.2% of the total.

The model domain discussed above (see Figure 2) was interpolated within the limits of the CFD model domain based upon a user specified grid size. All elements in the FLOW-3D grid are rectangular bricks (i.e. hexahedrals). To minimize numerical errors, vertical to horizontal aspect ratios were kept under 7. The grid size (both horizontal and vertical) was varied to minimize computation effort. Areas of high gradients and high hydrodynamic concern (i.e. the spillway) had grid sizes on the order of 1 to 2 ft. Areas away from the spillway (the area of concern for this study) and of low gradients generally had larger grids, with a maximum horizontal spacing of 30 ft and a maximum vertical spacing of 18 ft. The powerhouse tailrace block contained approximately 2.5 million cells and the spillway tailrace block contained 1.8 million cells.

The powerhouse tailrace block is approximately 4500 ft long (east-west), and terminates at a constriction in the bathymetry approximately 1600 ft downstream of Unit 1, as shown in Figure 2. The block is approximately 1850 ft wide (north-south) and extends vertically from the water surface to elevation –160 ft. The bathymetry in front of the powerhouse is quite complex, with the flow exiting the draft tubes (between elevation 5.75 and 31 ft) being immediately directed upwards over a sill (height of approximately 40 ft). The crest height of the sill is non-uniform, and has various peaks and valleys producing complex three-dimensional flow patterns in the tailrace downstream.

The tailrace block downstream of the spillway is approximately 4600 ft long (east-west), and terminates approximately 2100 ft downstream of the Highway 197 Bridge. The block is approximately 4200 ft wide (north-south) and extends vertically from above the free surface to elevation -195.

The upstream boundary of the spillway tailrace model is 175 ft upstream of the downstream boundary of the powerhouse tailrace model (i.e. the grids of the two models overlapped by 175 ft). The upstream boundary condition for the spillway model was of a fixed velocity type and was calculated by interpolating the powerhouse model solution to the spillway grid along the upstream spillway boundary (i.e. called a "grid overlay" boundary condition in FLOW-3D). Since only the upstream boundary of the spillway model was fixed, downstream velocities could vary based upon bathymetry and other flow conditions. This allowed for a independent check between the spillway and powerhouse models in the overlapped portion to ensure that velocities in this region were not impacted by non-included portions of the domain. The velocity fields generated by both models in the grid overlap region are nearly identical.

Discharge from each spillway bay was simplified in this validation case due to the number of computational cells and the length of simulation time necessary to reach a dynamic equilibrium. Instead of modeling the complete spillway bay, as was done for the 1:36 and 1:40 model, the flow from each bay entered the domain from a mass source placed at the bottom of the spillway face and care was taken to ensure that the jet exited with an approximately correct momentum flux. At the downstream and right (closest to Washington) sides of the domain, a pressure boundary was imposed that fixed the tailwater elevation to the value observed in the physical model.

The figure below displays contours of velocity magnitude along two horizontal planes: one plane was placed at elevation 50 ft, which is deeper than the shelf that extends away from the powerhouse (approximately elevation 68 ft), and one plane at elevation 72 ft, which is just below the free-surface. Figure 11 shows the general direction and magnitude of the velocity field downstream of the non-spilling bays. In this area the CFD model replicates an observed phenom-enon at The Dalles Project. A strong lateral flow (i.e. flow parallel to the spillway face) is produced by entrainment as the spillway jets flow through the stilling basin. This lateral flow may be of biological significance since flow that passes underneath the spillway jets becomes supersaturated with dissolved gas. One possible way to improve fish survival in The Dalles tailrace may be to structurally alter the spillway to minimize the quantity of flow that is laterally entrained.

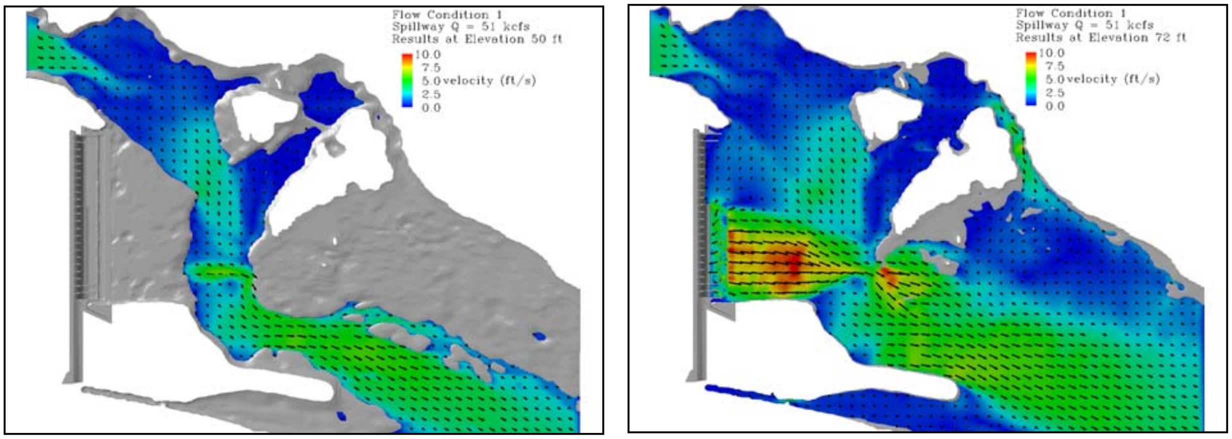


Figure 10 Numerical model solutions for the powerhouse tailrace block at elevations 50 ft (left) and 72 ft (right). The gray model bathymetry has been cropped at the water surface elevation for clarity.

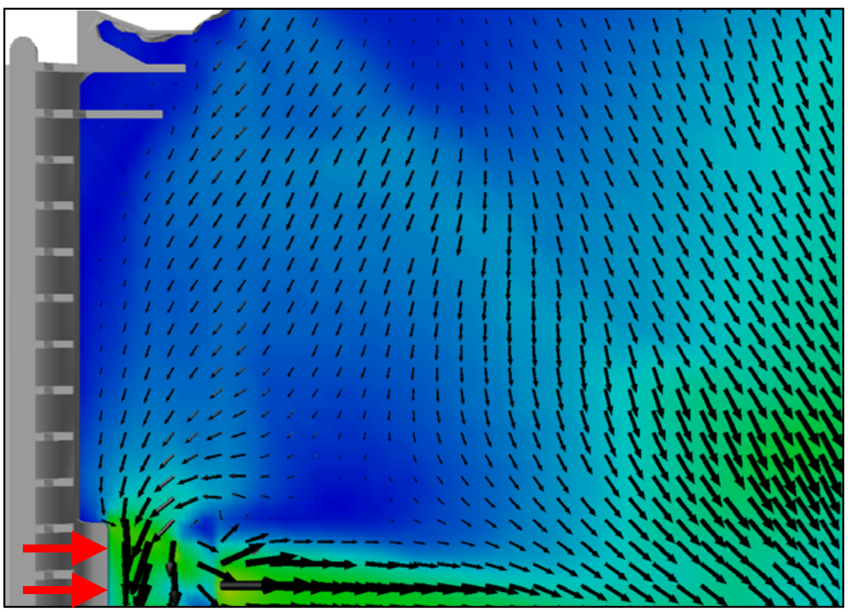


Figure 11. Close up of the velocity flow field downstream of the non-spilling bays. The mass blocks used to approximate the spillway jets can be seen in the lower left corner (red arrows denote direction of flow from the last two spilling bays).

Velocities were observed in the physical model using a Nixon meter along three transects downstream of the spillway (Davis, 2001). The first transect extended longitudinally downstream from the first non-spilling bay (Bay 14). The other two transects were longitudinal, extending from Bay 1 to the intersection with the longitudinal transect. Measurements were typically separated by 30 to 40 ft (prototype units).

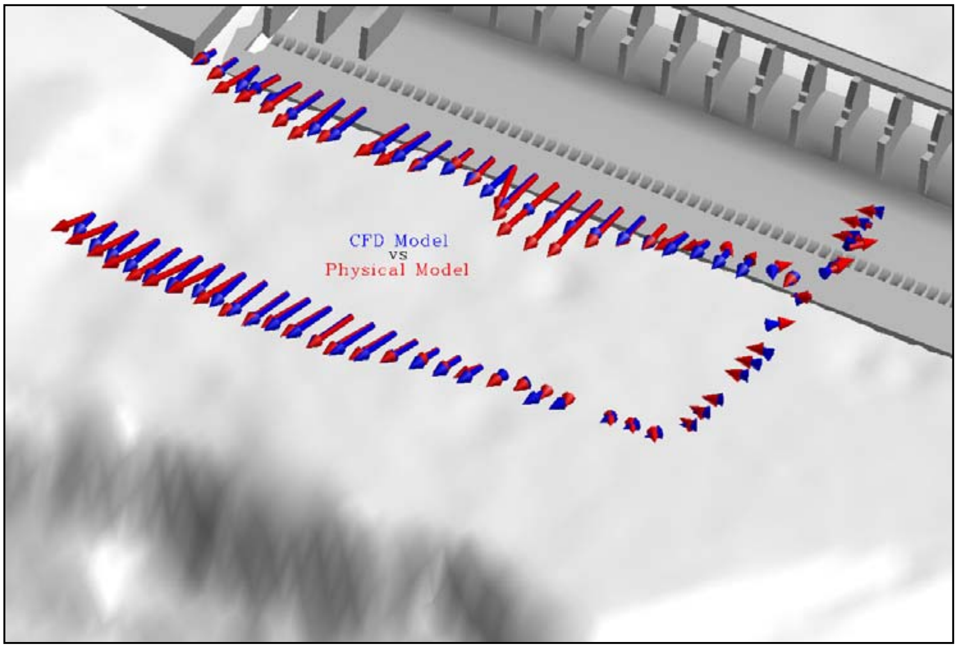


Figure 12 Oblique view comparisons of CFD and Physical Model data. The left most spillbay is Bay 1. The end of the spillway shelf is at elevation 68 ft and the sharp vertical drop downstream of the shelf is located in the lower left-hand corner of the figure.

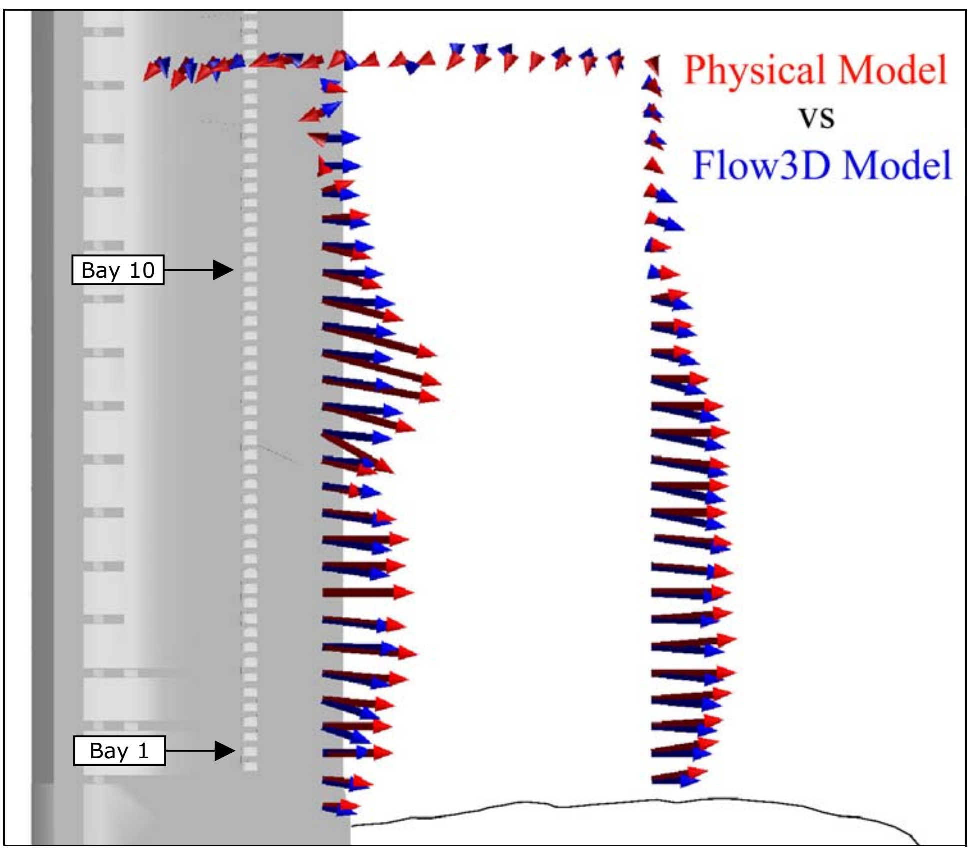


Figure 13 Plan view comparisons of CFD and Physical Model data. The black line along the bottom of the figure is the shoreline at elevation 76.8 ft.

CFD model results were compared to physical model data by extracting velocities from the vertical column of cell faces that surrounded the location of the physical model measurement (approximating the depth integrating impact of the Nixon meter's propeller). The horizontal size of the grid cells in this region was smaller than the spacing between physical model observations by a ratio of approximately two to one (two horizontal cells between every observed velocity measurement). Graphical results comparing these two data sets are shown in Figure 12 above (oblique view) and Figure 13 below (plan view).

Velocity results are in general agreement at most locations in the tailrace, however differences do exist. The largest differences in velocity magnitudes occur along the upstream lateral transect and downstream of Bays 8 and 9. At these locations velocity magnitudes reported from the physical model exceed 12 ft/s. Although there are longitudinal differences along the same transect in the CFD model, velocity magnitudes only exceed 10 ft/s at two locations, and these are both less than 10.5 ft/s.

Summary - Horizontal magnitude

Number of pts	Mean	Average difference	RMS difference	R^2
67	6.87	1.39	1.90	0.81

Summary - U velocity component

Number of pts	Mean	Average difference	RMS difference
67	5.80	1.35	2.06

Summary - V velocity component

Number of pts	Mean	Average difference	RMS difference
67	-1.45	1.28	1.63

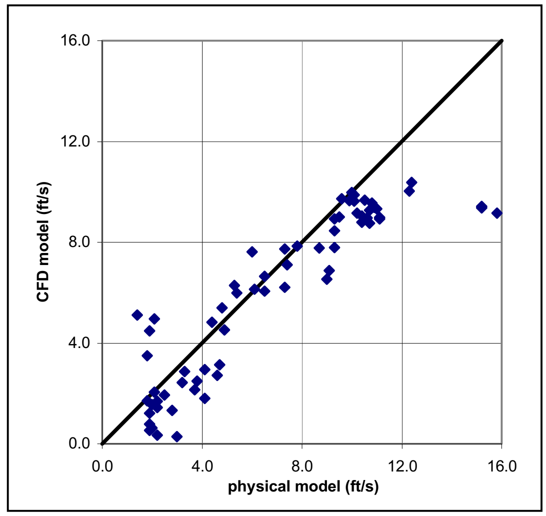


Figure 14 Comparison results between the 1:80 physical model and the numerically simulated results. All symbols should lie along the 45-degree line if the physical and numerical results were in perfect agreement. If the physical model magnitudes greater than 13 ft/s were removed (three locations) the R^2 would rise to 0.88.

Note that velocity magnitude differences downstream of Bays 8 and 9 diminish between the first and second transect. Along the most downstream transect, velocity vector differences are smallest between Bays 4 and 14. Between Bays 1 and 3 directional differences exist, although the velocity magnitudes are approximately equal. The cause of this difference may be due to discrepancies in bathymetry in front of these bays, since the bathymetry in the 1:80 scale physical model is known be 10 to 20 ft (or more) lower in this zone than in the prototype (MFC, 2001).

The table shown in the left portion of Figure 14 presents statistics regarding differences between the physical and CFD model velocity components. Included are the average (defined as the average of the absolute values of the differences) and RMS (root mean square) differences.

Average and RMS differences for the U and V velocity components are uniformly spread out across the basin. Both the U and V velocity components had comparable average differences, however their mean velocities are quite different. Reasons for these differences may be the transient nature of the flow field and how the data was sampled; data from the numerical model is from a single time-step of a transient simulation, while data from the physical model was averaged over a finite time period.

SUMMARY

A free-surface three-dimensional CFD model was applied to the tailrace of The Dalles Project. To test the validity of the results downstream of the spillway, the model was compared to results from 1:36, 1:40, and 1:80 scale physical models.

Results of the 1:40 scale CFD model were compared to observed 1:40 scale physical model data at four locations. At each location in the physical model, time-averaged mean and standard deviations of the horizontal velocity components were reported. These data were then compared to CFD model results at the same locations in the numerical model domain.

Results of this comparison show that the CFD model captured the primary features of the observed velocity profile. Out of 22 data comparison points, only six (27%) fell outside of a $\pm\sigma$ criterion when compared to physical model data (this reduces to one point (4.5%) if a $\pm 2\sigma$ criterion is used). Although differences exist, an improvement in the agreement between CFD and physical models may be possible if the exact flow rate through the physical model is known. For this test, the discharge in the physical model was approximated by integrating a single downstream velocity profile (i.e. a lateral transect was not performed). Since the gate opening and forebay stage height were fixed, this particular test case is extremely sensitive to flow rate. This is because any changes in flow rate due to an incorrect rating curve dictate that the velocity along the spillway face would change, which would impact downstream velocities throughout the entire stilling basin.

Pressure heads were observed in a 1:36 scale sectional model of the spillway and stilling basin of The Dalles Project. This flume was recreated in the CFD model with great success. A comparison of physical and numerical results shows that the pressure measurements were matched with an average error (prototype scale) of 0.5 ft around the baffle blocks and 0.4 ft around the end sill. The error between the two data sets may be within the accuracy range of the piezometers.

Creating and simulating a 1:80 scale model of the entire tailrace downstream of The Dalles Project was performed as a third validation test of the CFD model. The numerical domain used by the model consisted of a variable sized mesh of rectangular brick (i.e. hexahedral) cells. The numerical mesh was broken into two blocks: a powerhouse tailrace and a spillway tailrace. Each block contained approximately two million computational cells. Results in the powerhouse block have not been validated against physical or field data. It is hoped that in the future data in these regions will become available so that CFD results in this area can be validated.

The spillway tailrace block used the powerhouse solution as an upstream boundary condition. In addition, mass sources were added for each spillway bay that was operating during the simulation period. After the spillway tailrace solution had reached a dynamic equilibrium (i.e. although the CFD results changed over time, the time variations were relatively small and periodic), CFD results were sampled at locations that correspond to those observed in a 1:80 scale physical model. Results have been presented graphically, which show that the CFD model replicated the general trends of the 1:80 physical model. A numerical comparison revealed that the two data sets match with an average horizontal velocity difference of 1.4 ft/s.

At this time the numerical model has been validated against several physical models of varying scale. Future testing of the model against prototype data is required to gain confidence in its results, although the validation results documented in this paper show a strong ability to replicate physical model results.

REFERENCES

Davis, W.G. (2001). *Date Report: Model Study of The Dalles Dam, 1:80-Scale General Model, Feasibility Work in Support of the Spillway Improvement Study*, Engineer Research and Development Center, U.S. Army Corps of Engineers, Waterways Experiment Station, Vicksburg, Mississippi. Report dated 09/2001.

ERDC (2001). Memorandum for Commander, *Data Report, Model Study of The Dalles Dam, 1:80 scale General Model*, Comparisons of model surveys with recent hydrographic surveys. Email from Glenn Davis to Stephen Schlenker, 02/01.

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

Preslan, W. and S. Wilhelms (2001) *Dalles, Metrics Feasibility Study*, Engineer Research and Development Center, U.S. Army Corps of Engineers, Waterways Experiment Station, Vicksburg, Mississippi. Draft data report submitted to Portland District, US Army Corps of Engineers.

Versteeg, H.K. and Malalasekera, W. (1995) *An introduction to computational fluid dynamics*, Addison-Wesley, New York.

ACKNOWLEDGEMENTS

The US Army Corps of Engineers, Portland District, supported this work through PNNL project 43319.

AUTHORS

Christopher Cook, Ph.D., is a senior research engineer at the Pacific Northwest National Laboratory. Research interests include free-surface computational fluid dynamics modeling, development and application of multi-dimensional water quality models, and the use of various instruments to measure velocity fields using ultrasound (hydroacoustics).

Marshall Richmond, Ph.D., is a chief research engineer at the Pacific Northwest National Laboratory. Research interests include turbulence and computational fluid dynamics modeling, sediment transport, and the development of computational models for contaminant transport.

John Serkowski is a research scientist at the Pacific Northwest National Laboratory where he specializes in computer visualization of three-dimensional datasets.

Laurie Ebner, Ph.D., P.E., is a hydraulic engineer for the Portland District and oversees the CFD model studies for the District.