Leadership in Sustainable Infrastructure
Leadership en Infrastructures Durables

Vancouver, Canada
May 31 – June 3, 2017/ *Mai 31 – Juin 3, 2017*

BOUNDARY DAM TDG (TOTAL DISSOLVED GAS) ANALYSIS USING A CFD MODEL

Jalayeri, Nikou[1, 5] , Groeneveld, Joe [2], Bearlin, Andrew [3], and Gulliver, John [4]
[1] Engineer, Hatch, Vancouver, British Columbia, Canada
[2] Senior Engineer, Hatch, Calgary, Alberta, Canada
[3] Project Manager, Seattle City Light, Seattle, Washington, USA
[4] Consulting Engineer, University of Minnesota, Minneapolis, Minnesota, USA
[5] nikou.jalayeri@hatch.com

Abstract: The Boundary Dam is located on the Pend Oreille River in northeastern Washington. The project consists of a 340 ft. high concrete arch dam, seven low level sluiceway outlets, two high level overflow spillways, and a 660 MW powerhouse. The spillway and sluiceway discharge at the Boundary Hydroelectric Development have been shown to produce high total dissolved gas (TDG) concentrations in the tailwater of the spillway and the river reach downstream. Studies were commissioned to determine modifications to the project's spillway structures to help mitigate this gas production. Resolution of many of the hydraulic design issues for the study relied heavily on the results of numerical hydraulic models. These modifications were constructed and tested in the field. The CFD model that was developed in support of these studies was used to simulate flows through a number of the project's seven sluice gates and two overflow spillways. This model was also used to simulate the entry and movement of these flows through the project's downstream plunge pool and powerhouse area. The model was set up to track the pressure- and time-histories of representative air bubbles within the plunge pool and tailrace. These data were then used as input to a TDG predictive tool to help predict total dissolved gas production in the tailrace. The overall predictive performance was successfully calibrated and validated to actual prototype (field) TDG data.

1 Introduction

Boundary Dam is the third of five dams on the Pend Oreille River. The Project operates in a load-following mode, generating power during peak-load hours and curtailing generation during off-peak hours.

Figure 1 shows the key project features, consisting of an arch dam, reservoir, and underground power plant. Boundary Dam is a variable-radius concrete arch dam with a structural height of 340 feet, a crest length of 508 feet, and a total length of 740 feet. The arch dam varies in thickness from eight feet at the crest, which is at elevation 2,000 feet (NGVD 29) to 32 feet at the base at elevation 1,644 feet. The normal tailrace water surface elevation below the dam is 1,733 feet with approximately 261 feet of gross head for power purposes for a normal upstream pool level.

Boundary Reservoir extends 17.5 miles upstream to Box Canyon Dam. Total usable storage is approximately 41,000 acre-feet from the normal maximum water surface elevation of 1,994 feet to 1,954 feet as the 40-foot operating range authorized by the current license. Figure 2 shows a downstream view of the dam looking upstream. Spillway and sluiceway discharge at the Boundary project has been shown to produce high Total Dissolved Gas (TDG) concentrations in the river reach downstream. TDG is the amount of air held in saturation in the water and is quantified in the standards in terms of percent of saturation pressure relative to ambient barometric pressure. These high gas levels can be detrimental to the local fish species.

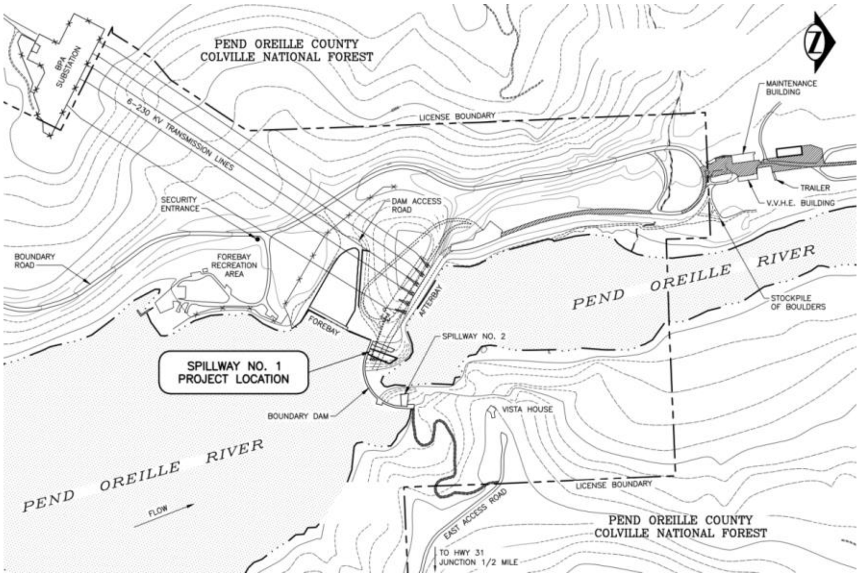

Figure 1 Boundary Dam Key Project Features

2 Planning of TDG Upgrades

The Seattle City Light (SCL) TDG team has convened a series of design workshops to brainstorm and develop alternatives, review Project progress, discuss technical aspects of the work, evaluate alternatives, and provide direction for ongoing development of operational and structural measures to reduce TDG downstream of the Project. The initial development of alternatives occurred during a two-day workshop in 2007, attended by SCL staff and an expert panel. During this initial workshop a large number of potential mitigation measures were proposed. SCL and the expert panel selected six alternatives that they considered to be feasible and likely to be effective, and presented these in relicensing study plan reports (SCL 2007 and SCL 2009a); these are the alternatives on which the SCL TDG team has focused their subsequent design development efforts. Since 2009, the TDG mitigation design team holds an annual design workshop, and provides input to or attends periodic agency meetings. The annual team meetings are a forum to review the years' progress and significant design advances, to brainstorm ideas and discuss challenges, and to establish the goals and prioritize the studies and design development for the following year.

The TDG abatement alternatives identified during the design workshops held in 2009 were investigated from the perspective of hydraulic performance and engineering feasibility. The three short-listed alternatives selected during this process (SCL 2009) were:

- Throttle Sluice Gates
- Roughen Sluice Flow
- Install Flow Splitters/Aerators on Spillways
    - Spillway deflectors, roughness elements, and/or dentated spillway

Figure 2 Downstream View of Boundary Dam Looking Upstream

Based on both physical and numerical hydraulic analysis, as well as engineering considerations, proposed modifications to Spillway No. 1 (SW1) and Spillway No. 2 (SW2) were selected for early implementation. SW2 modifications were scheduled first because SW2 had more favorable foundation conditions that would require a simpler analysis than that of SW1. Additionally, SW2 produced more TDG and was shorter, giving better access for construction of modifications.

The spillway modifications consisted of adding Roughness Elements (REs), similar to baffle blocks, to the spillway to induce turbulence, to accelerate break-up of the falling jet and to reduce jet penetration into the tailwater. The spillway REs were selected as the preferred first alternative to be implemented. Figure 3 shows the installed baffle blocks on the end of the SW2 chute.

Figure 3 View of REs Installed on SW2 Chute

3 Hydraulic Analysis and Design

Resolution of many of the hydraulic design issues for this concept relied heavily on the results of both physical and numerical hydraulic models. Both were used in complementary roles to maximize their particular strengths. The focus of this paper is on the numerical modeling portion of the work.

3.1 Computational Fluid Dynamics (CFD) Model

Full three-dimensional CFD models of SW1, SW2, a sluiceway, and the downstream plunge pool and river channel were developed in 2009. These models have been utilized as part of the suite of design tools used to develop an understanding of the governing hydraulic and hydrodynamic processes driving gas exchange in the tailrace of the existing project under spill conditions. In addition, these models been used to develop the designs of structural TDG mitigation alternatives, and in combination with the TDG predictive model described in subsequent sections of this paper, to predict the TDG performance of proposed TDG mitigation alternatives. The CFD models were used in conjunction with the physical model utilizing the relative strengths of the two analysis methods. The physical model was used to its greatest utility in iteratively developing physical geometry changes of the mitigation alternatives, while the CFD models were used to provide quantified information in terms of hydrodynamic loads for design purposes and flow fields for use in TDG prediction.

3.2 TDG Prediction model

TDG predictions were made for the project using a two step process: the CFD model was first applied to assess the plunge pool hydraulics and flow patterns, and then the hydraulic output of the CFD model was imported into the Plunge Pool Gas Transfer (PPGT) model on an Excel spreadsheet.

As a first step, the CFD model was run to simulate the hydraulics within the plunge pool area of the project for a particular configuration and flow scenario. The model used, FLOW-3D, is capable of simulating and tracking the movement of individual particles of air within a computational grid. This tracking feature provided a means to better understand how air, drawn in at the water surface, might circulate within the downstream channel and plunge pool. To do so, numerical models are initially run to achieve a quasi-steady flow regime downstream of the site where the mean velocities did not change appreciably with time. This is known as a “spin up” period. After this, the models are restarted and a number of discrete particles are released within the plunging jet. The particles are released in lines across the main flow stream at locations around the periphery of the jet or nappe to replicate the actual points at which air will become entrained.

The particles themselves are 0.02 ft in diameter, and are provided with a buoyancy that will propel them upwards at a velocity of 0.2 m/s (0.7 ft/s) in still water. This replicates the average still-water rise rate of entrapped bubbles. The model can then be used to track the movement of these particles. Where there are high downward vertical velocities, the particles tend to be drawn to depth. The buoyancy of the particles will eventually bring each back to the surface where they will be released into the atmosphere.

At the completion of each simulation, the particle data is extracted from the output database. The extracted data includes the position of the particle, the elapsed time, the particle velocity (x, y, and z directional velocities as well as total velocity magnitude) and the pressure at the particle location. Of these, the depth versus time and total velocity magnitude data are the most important for the subsequent calculation of TDG content. A large velocity generates greater gas transfer, and those particles that are drawn to depth, and remain at depth for periods of time, will contribute the most to TDG increases.

When the CFD simulations were completed, they were then imported into the PPGT model, where the transfer of air between the bubbles and the water is computed based on the overall depth/pressure time series that was tracked for the mass particles. When the overall mass transfer of the particles approaches zero, the steady state concentration of the tailwater is achieved in the computations. The PPTG model makes the following assumptions:

- There is sufficient air entrainment so that the rate of air entrainment is not a limiting factor. This means that there is sufficient air entrained to achieve a steady state concentration.
- The tailwater pool is well mixed.
- TDG concentration in the tailwater pool has reached steady state, in that any air added from the bubbles into the water at depth is balanced by an equal amount taken out of the water by the shallow bubbles, or that an overall mass balance has been achieved.
- The transfer of dissolved gas across the water surface is negligible. This assumption will cause the PPTG model to predict tailwater TDG that is above measurements.

Using this strategy, the transfer of air across a moving bubble-water interface is computed as:

[1] $$\frac{dM}{dt} = K_L A (C - C_E)$$

where;

dM = the amount of air transferred into the dissolved phase,

dt = the time interval,

K_L = transfer coefficient for nitrogen and oxygen,

A = the surface area of the bubble

C = the concentration of air within the water

C_E = the equilibrium water concentration, which is directly related to concentration in the bubble.

The problem is thus formulated so that each bubble is followed through the pool. The bubbles should be placed at the locations where the air would be entrained, and followed through the pool until they leave at the surface. Dividing Eq. (1) by the equilibrium concentration at atmospheric pressure, Cs, gives:

[2] $$\frac{1}{C_S}\frac{dM}{dt} = K_L A \left(\frac{C}{C_S} - \frac{C_E}{C_S}\right)$$

or

[3] $$V\frac{d\left({}^{C}/_{C_S}\right)}{d_t} = K_L A \left(\frac{C}{C_S} - \frac{C_E}{C_S}\right)$$

Where C/Cs and C_E/Cs are fractions of atmospheric saturation, and V is the volume in which C is determined ($M = C \cdot V$). The surface area of the bubble will decrease with the increase in pressure at depth:

[4] $$A \cong A_0 \left(1 + \frac{depth(ft)}{34}\right)^{2/3}$$

where A_0 is the surface area of a given bubble at zero depth. An additional assumption in Eq. (4) is that the surface area of each bubble is independent of the amount of mass that has previously been transferred. This assumption may affect the surface area of each bubble, but would not violate the overall mass balance of all bubbles because the system is at steady state with regard to TDG concentration.

Equation (2) allows one to compute the concentration in the tailwater at steady state, when the sum of *dM/dt* for all bubbles is equal to zero. Although each bubble may have a non-zero value of *dM/dt,* the sum of *dM/dt* for all bubbles at all times must be equal to zero for the tailwater to be at steady-state TDG concentration. For example, with SW1 operating at 12,000 cfs, and SW2 operating at 10,000 cfs, the average residence time of the bubbles in the tailrace model domain is approximately 40 seconds. In the computations, bubble location and velocities were output every 0.5 seconds. Thus, with 50 bubbles, average residence time of 40 seconds, and mass transfer computations every 0.5 seconds, a total of 50 x 40 s / 0.5 s = 4,000 separate computations of mass transfer as the bubbles are followed through the flow. Some of these mass transfer calculations will be positive (adding TDG to the water) and some will be negative (stripping TDG from the water), depending upon the depth of the bubbles and the TDG concentration assumed for the tailwater. The value will also depend upon the velocity of the bubble and, to a lesser extent, upon water temperature. The tailwater TDG level is changed until the total sum of these 4,000 mass transfer calculations is zero, which means that steady state and an overall tailwater equilibration of mass transfer has been achieved.

For a bubble at depth, assuming limited change in the molar ratio of oxygen and nitrogen in the bubble,

$$[5] \quad \frac{C_E}{C_S} \cong 1 + \frac{depth(ft)}{34}$$

Azbel (1981) developed an equation for the mass transfer coefficient from a bubble swarm, which Thompson and Gulliver (1997) altered to result in an equation that could describe the mass transfer from bubbles in a shear flow:

$$[6] \quad K_L = (2\pi D)^{1/2} \frac{U^{\eta}}{L^{1-\eta} \nu^{\eta-1/2}}$$

where;

D = diffusion coefficient of air in water (~2x10^{-8} ft^2/s),

U = a representative turbulent velocity,

L = a representative length scale,

v = kinematic viscosity of the water (~10^{-5} ft^2/s)

η = a coefficient that generally varies between 0.5 and 1.

Azbel (1981) used η = 0.75. Thompson and Gulliver used η = 0.55. The difference in computed TDG with the two exponents is not great, so 0.75 is used in these computations.

The length scale, L, is assumed to be the same for both the sluice and spillway discharge, and is typically set at 60-75% of the smallest discharge dimension. L has a small effect on predicted TDG levels.

The turbulent velocity, U, is the velocity of the buoyant particle, calculated from FLOW-3D.

4 Validation of the TDG Tool

TDG measurements have been made at various locations and times of the year. These tests have been grouped into operational scenarios involving only sluice flow, and operational scenarios involving only discharge through on or more of the high-level spillways. The data has been further subdivided in to tests that showed some stripping of the TDG, tests in which downstream TDG levels remained neutral, and tests in which the downstream TDG was seen to rise significantly.

Upon reviewing the data, a number of test cases were selected to test the performance of the TDG predictive tool, i.e., the CFD and PPTG models. These cases involved operation of each spillway individually, as well as cases involving the combined operation of both.

One such test case involved the operation of SW1 in tandem with the project powerhouse facility. The first scenario represents an operating condition which occurred on June 9, 2007, in which 10,220 cfs was passed through SW1 and 45,110 cfs though the powerhouse. The incoming TDG level for this test was 121.5% and the outgoing TDG was measured as 127.5%. A second more recent test case was also simulated.

The model was setup to replicate both of these operational scenarios, and the results of these validation runs are shown in **Error! Reference source not found.** below. As shown, although the simulation tool appears to over-predict the TDG levels at the monitoring site in both cases, (by up to approximately 2.5 percent), the overall match was considered to be quite reasonable. It should be noted that the field data also show considerable variation in the measured TDG level at the USGS gauge, even for comparable flow conditions.

Table 1 Summary of Spillway 1 Validation Runs

Spillway 1 Q (cfs)	**Powerhouse Q (cfs)**	**Total Q (cfs)**	**Upstream TDG**	**Tailrace TDG (Measured)**	**Tailrace TDG (TDG Tool)**	**Difference in TDG**
10,000 (June 9, 2007)	45,110	55,110	121.5%	127.2%	129.7%	-2.5%
13,000 (June 20, 2016)	24,000	37,000	106.0%	124-125%	125.6%	-0.6%

4.1 Application of the TDG Tool

The validation tests have shown that the developed tool, based on the particle tracking modeling technique, does a reasonable job of predicting TDG levels downstream of the project. The next step in the assessment was to use the TDG tool to predict the TDG reduction that might be possible at the project's downstream pool with and without the proposed modifications in place. This allows a relative comparison to be made on the effectiveness of each proposed modification in reducing TDG levels.

For these tests, it was assumed that a flow of 48,000 cfs would be released through the powerhouse. The reservoir level for the tests was assumed to be at the project's Full Supply Level of elevation 1990 ft, and the incoming TDG level was assumed to be 106%. The following sections of this paper summarize test results for the proposed SW1 modifications, which involve the installation of a single row of roughness elements to the downstream chute.

4.1.1 Existing Case - No Modifications

The model was first run to simulate flow conditions for the existing or base case scenario with flows of 10,000, 13,000, and 20,000 cfs through SW1. The simulated hydraulic conditions for this test were analyzed. Bubble particles were then added to this model, the run was re-started, and the particles were tracked until they were able to reach the surface, and exhaust back into the atmosphere. Figure 4 shows the overall nature of the jet as it falls into the plunge pool for flows of 10,000 and 13,000 cfs. As shown, the spread of the jet is quite limited. The particle data was then extracted from the model, and input to the spreadsheet model.

The results for these tests are summarized in Table 2 below. Based on these results, it is expected that with SW1 operating, TDG levels downstream of the project will rise significantly. This is to be expected, since the maximum and average particle plunge depth increased with larger flow releases through SW1. For the smaller flow of 10,000cfs, the spillway jet is more easily broken up during its fall into the plunge pool, whereas for higher flows, while the edges of the jet may break up, a competent jet core persists through the jet's entry into the plunge pool. The larger, more competent core associated with these higher flows is able to penetrate to the pool bottom, bringing entrained air to a much greater depth.

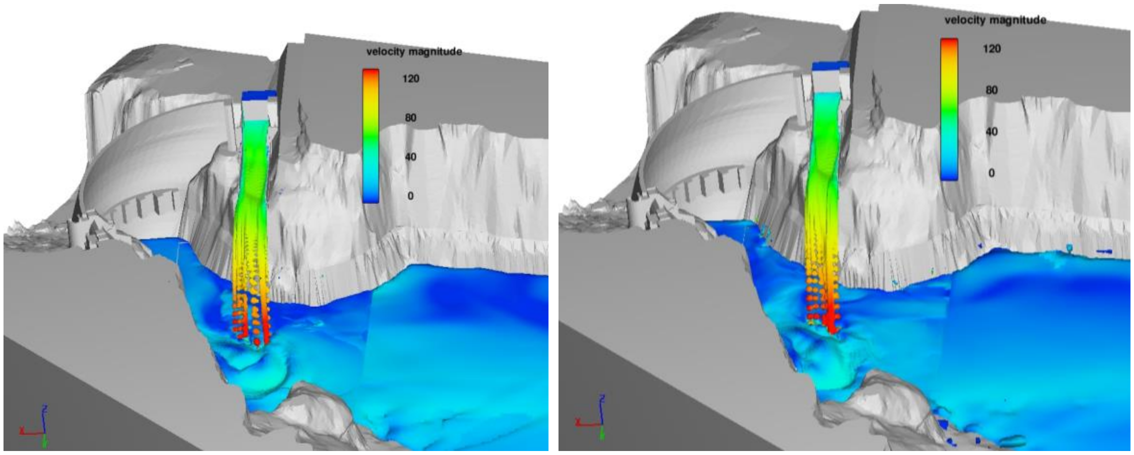

Figure 4 3-D View of the Unmodified Spillway 1 Jet : 10,000 cfs Flow (left), 13,000 cfs Flow (right)

4.1.2 SW1 Modification

Following the base case runs, various CFD simulations were conducted to assess the hydraulic conditions that would result from the introduction of REs on the downstream end of the SW1 chute. The introduction of these REs helps to break up the jet at the end of the chute more quickly and efficiently, accelerating boundary layer growth and resulting in the formation of small "packets" of water entering the plunge pool rather than coherent streams/jets.

Given concerns for potential cavitation damage on the spillway chute floor and on the REs themselves, additional runs were undertaken to test the effect on flow conditions at the REs if a ramp were to be installed immediately upstream of the roughness elements. The ramp would rise from the floor of the chute, to meet the upstream face of the first row of roughness elements, as shown in Figure 5 below.

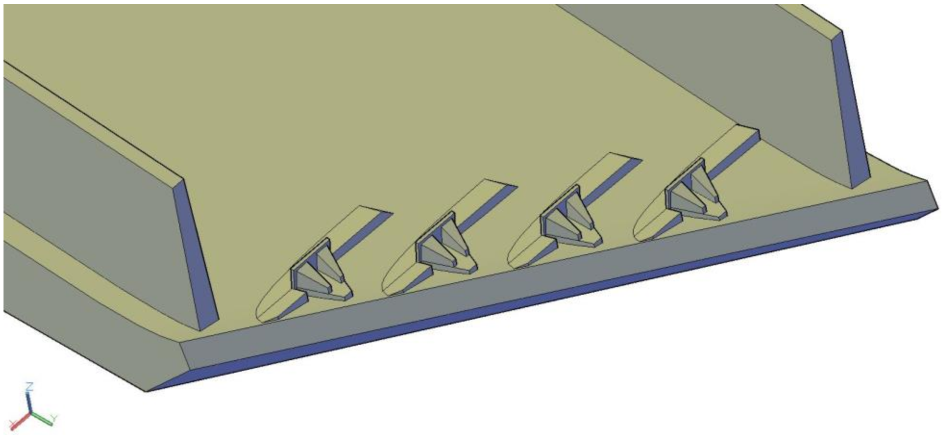

Figure 5 3-D View of Spillway No.1 Roughness Elements

The next step in the assessment of these schemes was to estimate the impact that the addition of these modifications would have on TDG levels downstream of the project under a range of flows. To do so, three predictive runs were made. For each run, it was assumed that 48,000 cfs would again be passed through the powerhouse. The reservoir level for each test was assumed to be at the project's Full Supply Level of elevation 1990 ft, and the incoming TDG level was again assumed to be 106 % of saturation. CFD runs were made with identical flow releases through SW1 of 10,000 cfs, 13,000 cfs, and 20,000 cfs, and bubble histories were extracted from the CFD results and input to the TDG predictive spreadsheet model.

The results of the CFD runs are presented in Figure 6 below. Comparing Figures 4 and 6 indicates that the plunging jet is more broken up and spread further with RE. **Error! Reference source not found.** also provides a final summary table for the RE runs, and compares key parameters associated with each flow condition, including the maximum plunge depth, the average plunge depth, the predicted TDG percentage at the USGS gauge downstream of the project, and the TDG contribution above incoming levels by the spill release.

As shown in Table 2 the proposed RE configuration appears to deliver the greatest TDG reduction when SW1 is operating at a flow of approximately 10,000 cfs. With the roughness elements in place, the reduction in TDG value for this flow is approximately 6.6 percent when compared to baseline conditions. For 13,000 cfs, the expected TDG reduction below the baseline condition is smaller, and estimated to be approximately 4.9 percent. For the 20,000 cfs, the expected reduction from the baseline condition is estimated to be approximately 2.4 percent. For the 20,000 cfs flow case, the ability of the roughness elements to break up the jet appears to be reduced, since the jet begins to override the roughness elements. This results in the formation of a more competent jet core that is able to penetrate the plunge pool to a greater depth.

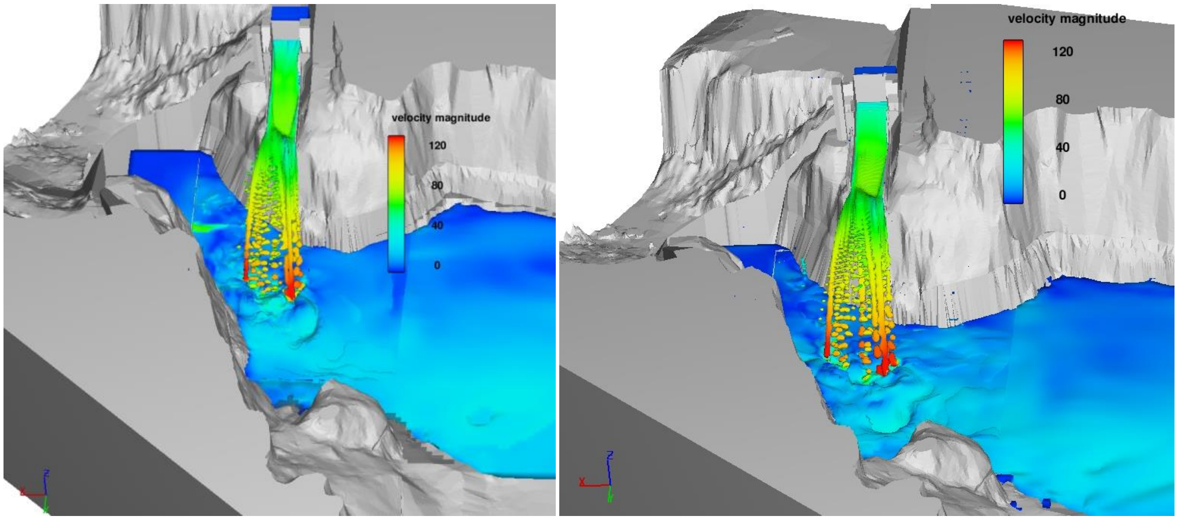

Figure 6 3-D View of the Modified Spillway 1 Jet : 10,000 cfs Flow (left), 13,000 cfs Flow (right)

Table 2

Summary of TDG Predictive Results: SPW1

Parameter	**Values for Various Discharges**					
	Baseline (cfs)			**With Roughness Elements and Ramp (cfs)**		
	10,000	**13,000**	**20,000**	**10,000**	**13,000**	**20,000**
Maximum Depth of Plunge (ft)	66	68	108	36	49	85
Average Depth of Plunge (ft)	40	42	92	18	22	43
TDG Computed at USGS Site (%)	117.6	117.9	127.4	111.0	113.0	125.0
TDG Contribution from Spill (%)	11.6	11.9	21.4	5	7	19
Change in Computed TDG Relative to Baseline	n.a.	n.a.	n.a.	-6.6	-4.9	-2.4

5 Summary

In summary, a tool has been developed to help support the design and evaluation of TDG mitigation projects for the Boundary project. This tool, which includes both a particle tracking CFD model and a spreadsheet (PPTG) analysis model has been validated through various field tests. Going forward, it will continue to be used to provide a means to assess the overall magnitude of gas reduction possible for various operational and structural options.

References

Azbel, D., 1981. *Two phase flows in chemical engineering.* Cambridge University Press, Cambridge, England, 27-50, 155-210

Flow Science Inc., 2005. FLOW-3D User's Manual. Flow Science Inc., Santa Fe, NM 87505, USA

SCL. 2016. Boundary Hydroelectric Project (FERC No. 2144) Total Dissolved Gas 2016 Annual Report. Nov 2016.

Thompson, E.J. and Gulliver, J.S.,1997. "Oxygen transfer similitude for a vented hydroturbine," J. Environmental Engineering, 123(6) 529-538.