Three-Dimensional Computational Fluid Dynamics Model Study Report

Lowell Hydroelectric Project

(FERC No. 2790)

November 1, 2021

Prepared by:

HDR

This page is intentionally left blank.

Contents

Tables

Figures

List of Acronyms

3D	three dimensional
ASCII	American Standard Code for Information Interchange.
AWS	auxiliary water supply
CFD	Computational Fluid Dynamics
cfs	cubic feet per second
DEM	digital elevation model
El.	elevation
FERC	Federal Energy Regulatory Commission (or Commission)
ft	feet
GIS	Geographic Information System
LAS	LASer file format
LiDAR	Light Detection and Ranging
NAD	North American datum
NAVD 88	North American vertical datum of 1988
NGVD 29	National Geodetic vertical datum 1929
Normandeau	Normandeau Associates, Inc.
Project	Lowell Hydroelectric Project (or Lowell Project)
RNG	Renormalized Group
STL	Standard Tessellation Language
Study	CFD Modeling Study
TIN	triangulated irregular network
URANS	Unsteady Reynolds Averaged-Navier Stokes
USGS	U.S. Geological Survey
UTM	Universal Transverse Mercator
WSE	water surface elevation

1 Introduction

1.1 Purpose

The sections below detail the methods used to develop the Computational Fluid Dynamics (CFD) model and provide the requested data for the Lowell Hydro Fish Passage CFD study. This technical memo describes the numerical methods, model geometry, mesh development, boundary conditions, and results.

1.2 Background

The Federal Energy Regulatory Commission's (FERC) June 15, 2018 Scoping Document 1 identified the environmental resource issues at the Lowell Hydroelectric Project (Project) (FERC Project No. 2790) for analysis. The U.S. Fish and Wildlife Service, National Marine Fisheries Service, Massachusetts Division of Fisheries and Wildlife, and New Hampshire Fish and Game Department submitted formal requests for a CFD Modeling Study (Study) of the Project's fish passage facilities.

The goal of the Study is to determine the flow field conditions that exist in and around the Project's fish passage facilities, including around the fishway entrances, within fishway structures, and in the E.L. Field Powerhouse forebay. Information derived from this Study may be used in conjunction with telemetry and other studies performed for the Project relicensing (American Eel Passage Downstream Study, Juvenile Alosine Downstream Study, and Adult Alosine Passage Study Upstream and Downstream) to analyze fish behavior in response to hydraulics. This is anticipated to aid in the interpretation of preferable conditions for the guidance of migrating fish to and through the fish passage facilities.

The study area includes a portion of the Pawtucket Dam E.L. Field Powerhouse forebay, tailrace, and fish lift, the bypass reach in the vicinity of the Pawtucket Dam fish ladder entrance, and within the fish ladder.

2 CFD Model Development

2.1 Model Description

FLOW-3D is a commercially available software developed and supported by Flow Science, Inc. that is capable of solving the three-dimensional (3D) Unsteady Reynolds Averaged-Navier Stokes (URANS) equations. The software utilizes a Volume of Fluid method to calculate the free surface within the domain (Hirt & Nichols, 1981). The package contains the meshing module (pre-processor), solver, and post-processor. FlowSight was used to produce the results presented below and is provided with the FLOW-3D software package.

2.1.1 Modeling Approach

The governing equations used in Flow-3D are provided in the software user's guide (Flow Science, Inc, 2019). The software solves fully URANS equations on structured grids. Model fitted meshes were developed for the Pawtucket Dam fish ladder, E.L. Field Powerhouse forebay and E.L. Field Powerhouse tailrace. Known water surface elevations (WSE) and/or known flowrates were applied to the upstream boundaries of the CFD models. Applying the known WSE reduces the complexity of the URANS equations.

PRESSURE SOLVER OPTIONS

Two numerical schemes are available for the pressure solver with multiple options. Explicit and implicit solvers are available. Within the implicit solver, multiple options are available. Limited compressibility models can be toggled to relax the constraints of the pressure solver for cases where solution stability is an issue. The implicit pressure solver was applied to the model for the results presented below.

TURBULENCE MODELS

Various one (Prandtl Mixing Length and Turbulent Energy Model) and two equation (k-ε, k-ω, and RNG) turbulence models are available in FLOW-3D. A large eddy simulation model is also available for selection depending on the type of flow expected and desired flow feature resolution. The Renormalized Group (RNG) model was selected and is an applicable closure for the CFD Study based on anticipated flow patterns (Orszag & Yakhot, 1986).

MODEL LIMITATIONS

The CFD model is limited in the data it can accurately produce. Some hydrodynamic features are not accurately modeled with the selected solver and turbulence closure. For example, recirculation patterns and vortices are approximate in size and strength. Results within this memorandum are subject to the inputs at the time of the study.

2.2 Model Geometry

HDR developed a topographic Digital Elevation Model (DEM) using Geographical Information Systems (GIS). All geo-processing of the following data sources was achieved using the ArcMap (Esri) application's 3D analyst geo-processing tools.

A single DEM was created from the following data sources:

- Ortho imagery Light Detection and Ranging (LiDAR) based elevation data collected by Normandeau Associates, Inc. (Normandeau) provided in coordinate system Universal Transverse Mercator (UTM) Zone 19N, North American datum (NAD) 83 2011, units meters, North American vertical datum of 1988 (NAVD 88), units meters.
- Bathymetry depth points collected by Normandeau from three areas; 1) just upstream of the main dam fish ladder, 2) the power canal, and 3) the powerhouse tail race. Data provided in coordinate system UTM Zone 19N, NAD 83 2011, units meters, vertical datum NAVD 88, units feet.

- LiDAR acquired from United States Geological Survey (USGS) LASer file (LAS) format 2011. Data provided in coordinate system UTM Zone 19N, NAD 83, units meters, vertical datum NAVD 88, units meters.

The domain for each of these data sources is presented on Figure 2-1.

Most of the coverage area required for CFD modeling was comprised of data received by Normandeau as a randomly spaced point cloud xyz file. The xyz file was derived from remotely sensed survey in NAVD 88 vertical datum. These data were transformed as a discrete point feature class using "American Standard Code for Information Interchange (ASCII) 3D to Feature Class" geo-processing tool. Both the horizontal coordinate system (UTM 83 meters) and vertical units (NAVD 88 meters) were maintained. Figure 2-1 shows the Project reach and model terrain data sources.

The vertical units for the bathymetric survey data points were converted from feet (ft) to meters in order to match the vertical units of the larger areas of the overall CFD model area. A final step of converting the terrain to National Geodetic vertical datum (NGVD 29) was performed by adding 0.242 meters (0.793 ft) to the vertical units.

In several small overbank areas upstream of the fish ladder and along the east bank of the power canal and tailrace, additional terrain data were needed to provide additional detail; USGS LiDAR randomly spaced point cloud data was utilized. These USGS LiDAR derived files were converted from LAS to points using geo-processing tools "LAS Dataset to triangulated irregular network (TIN)" then "TIN Node".

In the areas of the fish ladder and fish elevator, polygon feature class were created to depict a flat area higher in elevation (El.) than the bottom of the Sterolithography file in Standard Tessallation Language (STL) file format in order to exclude water from ‘seeping’ between the final terrain DEM and the STL.

All components as described above were combined into a TIN using the "Create TIN" geo-processing tool. This resulting TIN was then converted to a final DEM raster using "GIS software within ArcMap (ESRI). The final step was to convert the .tiff raster to ASCII file "Raster to ASCII" for final import into the CFD model.

The fish ladder and E.L. Field Powerhouse structures were created in AutoCAD and exported to an STL in NGVD 29 vertical coordinates. The STLs and ground surface were rotated to align to an arbitrary vertical position. The rotation allows the structure location to align with the orthogonal mesh elements required for the solver. The rotated model reduces the number of mesh elements necessary to define the features within the model domain.

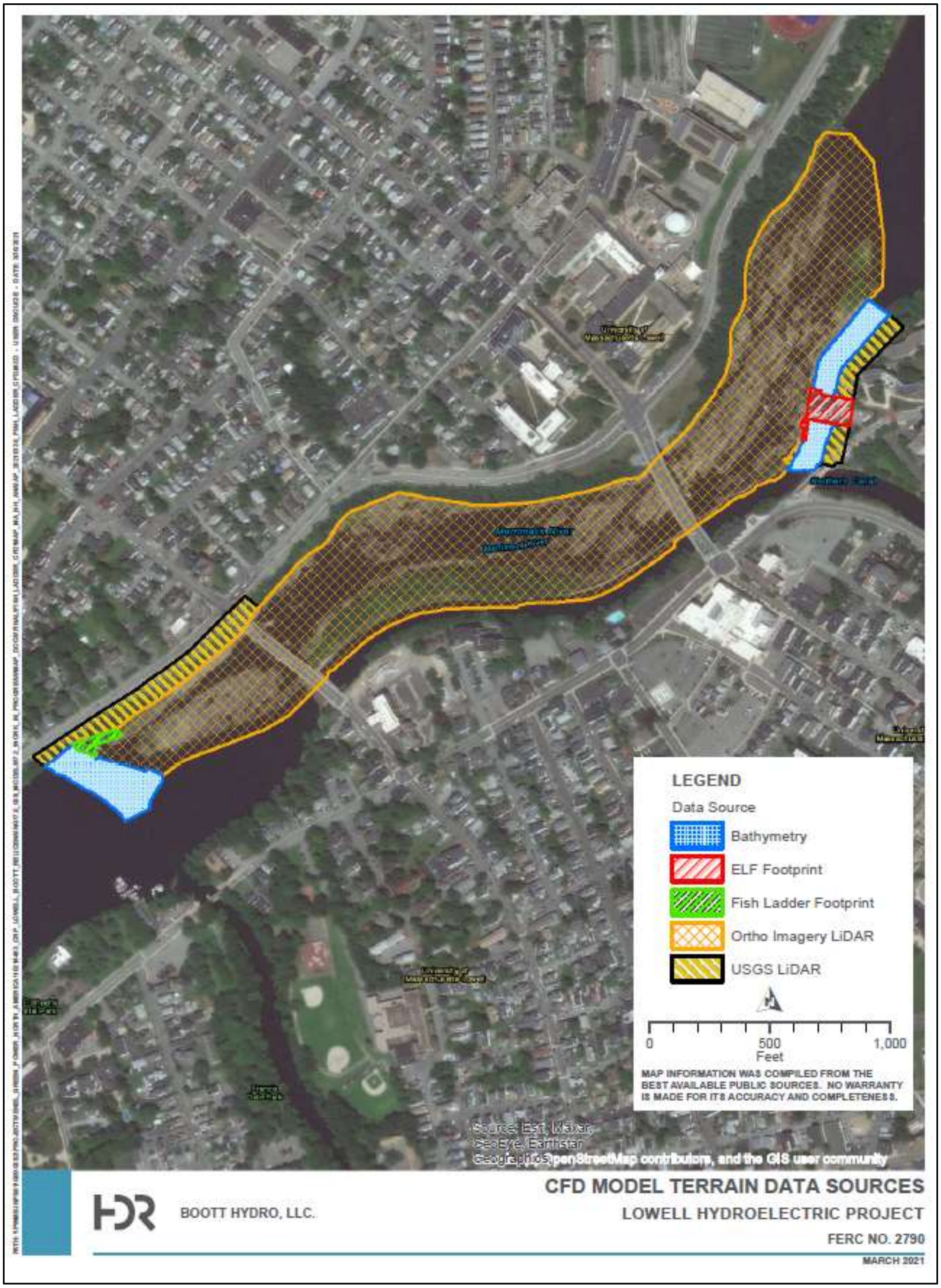

Figure 2-1: Data Sources Used in CFD Modeling Study Terrain Model Development

3 Mesh Development

The pre-processor for FLOW-3D works with orthogonal elements. The model topography and features were rotated to capture significant features with fewer elements. A balance between mesh density and computational time was desired. Several iterations of mesh density were performed, providing the basis for a mesh sensitivity analysis. A multi-block approach was utilized to build the model domain. The use of a multi-block domain allows multiple mesh sizes to be used throughout the domain. Each mesh block conformed to best practice guidelines provided by Flow Science (Flow Science, Inc, 2019).

3.1 Fish Ladder

3.1.1 Mesh Sensitivity Study

Multiple meshes of the fish ladder were analyzed to determine if the solution was grid independent. The first grid tested used 0.8-ft spacing in all three axial directions (coarse). After completion of the first simulation, the grid was refined to use a 0.4-ft spacing in all three axial directions (standard). The 0.4-ft mesh was further refined in a third test to limit the mesh spacing to 0.2-ft (refined). Figure 3-1 shows the meshed blocks with 0.4-ft spacing.

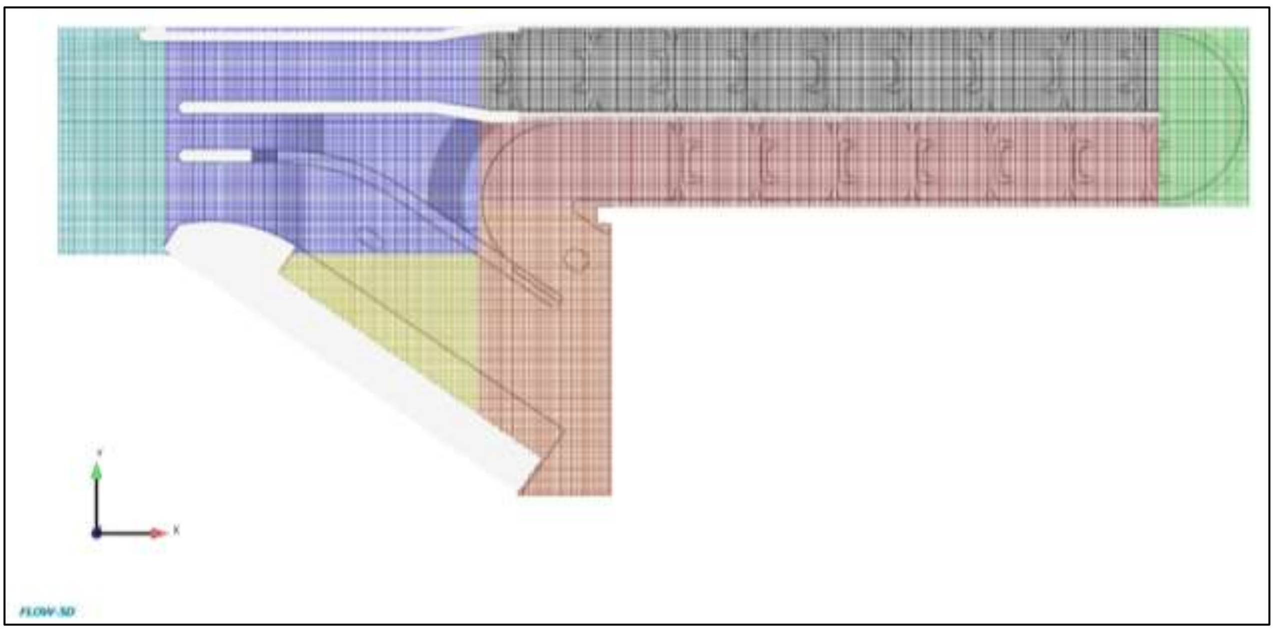

Figure 3-1: Fish Ladder Mesh Blocks (0.4-ft grid spacing)

During the mesh development stage of the CFD Study, multiple runs using identical boundary conditions were completed to determine the sensitivity to mesh parameters. Virtual probes measuring depth and velocity were placed throughout the fish ladder to measure the effects of the mesh refinement on model hydraulics.

The standard 0.4-ft grid spacing was used as a base for the final model. Further refinement to the vertical spacing resulted in a 0.2-ft vertical mesh size in the fish ladder. Additional refinements to the fish ladder mesh were assigned to the inflow area to improve model stability. WSE differences were minimal, and depth and velocity differences were minor between the refined and standard grid spacing.

3.1.2 Adding Terrain and Fish Ladder Features

Following the mesh sensitivity study, the Merrimack River bathymetry and Lowell Hydro Dam features were added upstream and downstream of the fish ladder. The combined bathymetry, dam, and fish ladder geometry is shown on Figure 3-2. It should be noted that the best available LiDAR data was used to capture the downstream bathymetry. Water levels in the Merrimack River at the time of LiDAR collection were such that only the five most up-stream weirs were captured in the terrain data. These weir geometries, and the face of the dam, were then refined within Flow-3D using existing as-built drawings as a guide. The bypass weirs were added to the model to provide a visual representation of potential flow features and not intended for detailed analysis.

A meeting between HDR and Boott Hydropower, LLC was held on March 2nd, 2021. Several updates to the fish ladder geometry were requested during the meeting and the following features were added:

- Baffles (2-ft and 4-ft) were installed along the inside and outside runs (facing downstream), respectively. These baffles sit approximately 6 inches above the floor of the fish ladder (shown on Figure 3-3).
- The inside section of the fish ladder exit has been sealed (shown on Figure 3-3).

The model extends from the upstream entrance of the fish ladder approximately 1000 ft downstream to the Mammoth Road Bridge.

As the fish ladder was the area of interest, the combined model mesh was refined to 0.4-ft grid spacing in the x- and y-axial directions and 0.2-ft grid spacing in the z-axial direction in the fish ladder vicinity. To facilitate reasonable computation run times, the mesh refinement was decreased from 0.4-ft, to 0.8-ft, and finally 1.6-ft grid spacing in the x- and y-axial directions moving away from the ladder. For the bypass reach blocks (non-fish ladder mesh blocks), the z-axial mesh spacing was a constant 0.4-ft. The three refinement levels are shown on Figure 3-4 as red (0.4-ft), green (0.8-ft) and blue (1.6-ft grid spacing) areas.

3.1.3 Bypass Weir Modifications

After discussions of preliminary results for the bypass area, a request to modify the bypass weirs to reflect the recommended configuration was made. The modifications to the bypass weirs were done to match the surveyed elevations of the recommended configuration. The surveyed elevation was originally provided on April 1st, 2021 (US Fish and Wildlife, 2014). The modified weir configuration was confirmed on a call with the Zone of Passage working group on June 21, 2021.

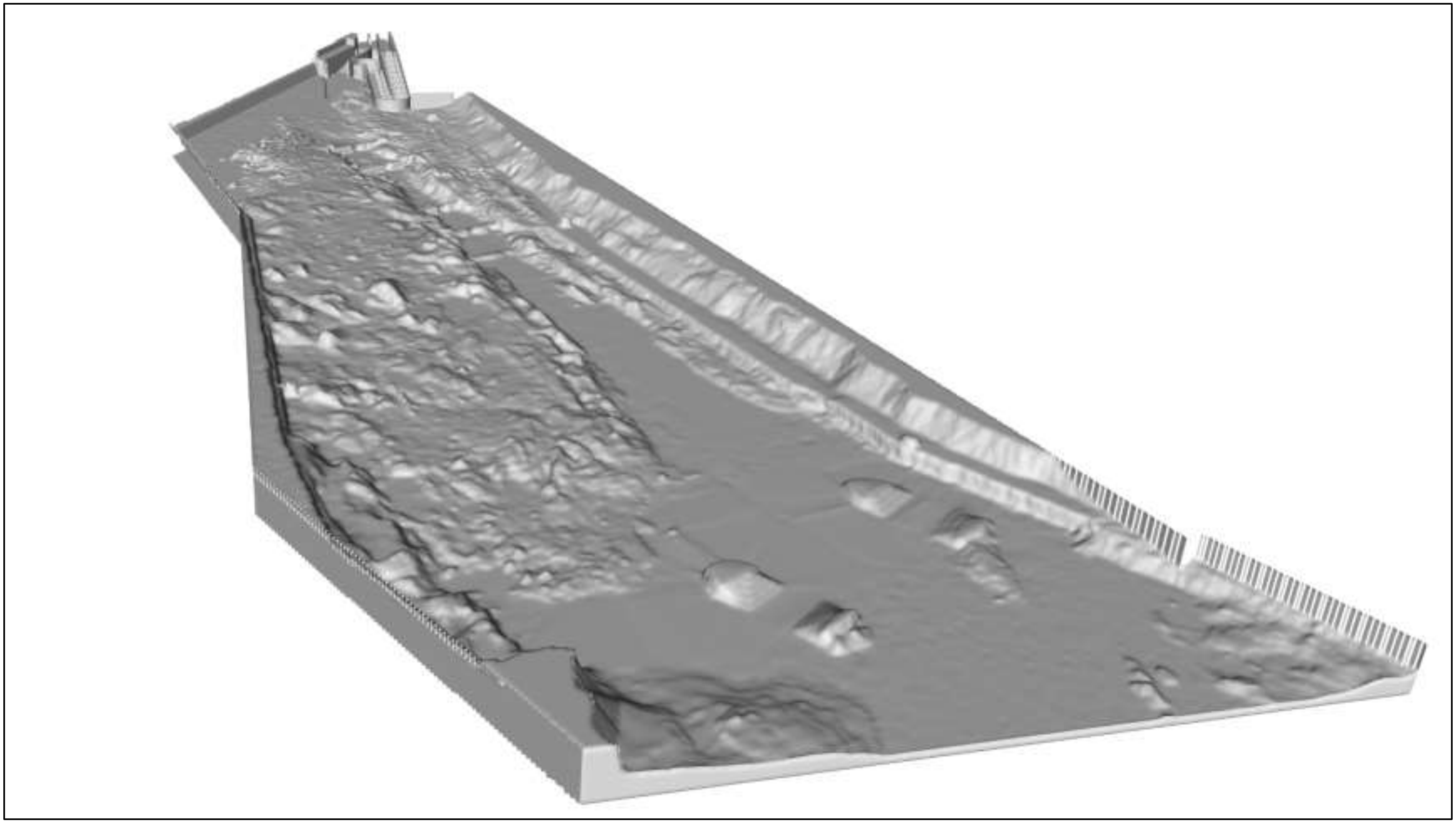

Figure 3-2: Combined Fish Ladder, Dam, and Bathymetry Geometry

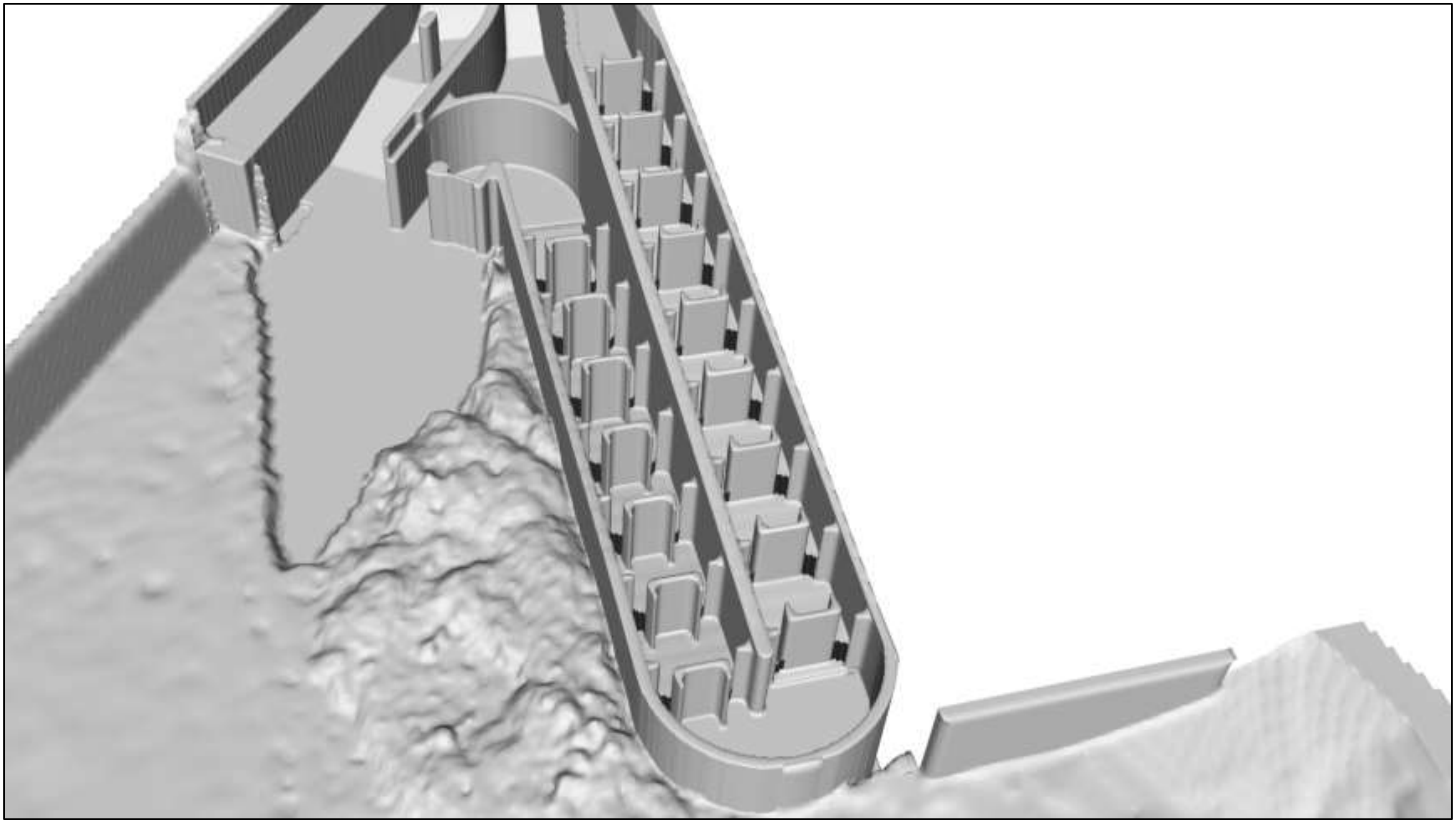

Figure 3-3: Fish Ladder Baffles (black) and Sealed Exit Section

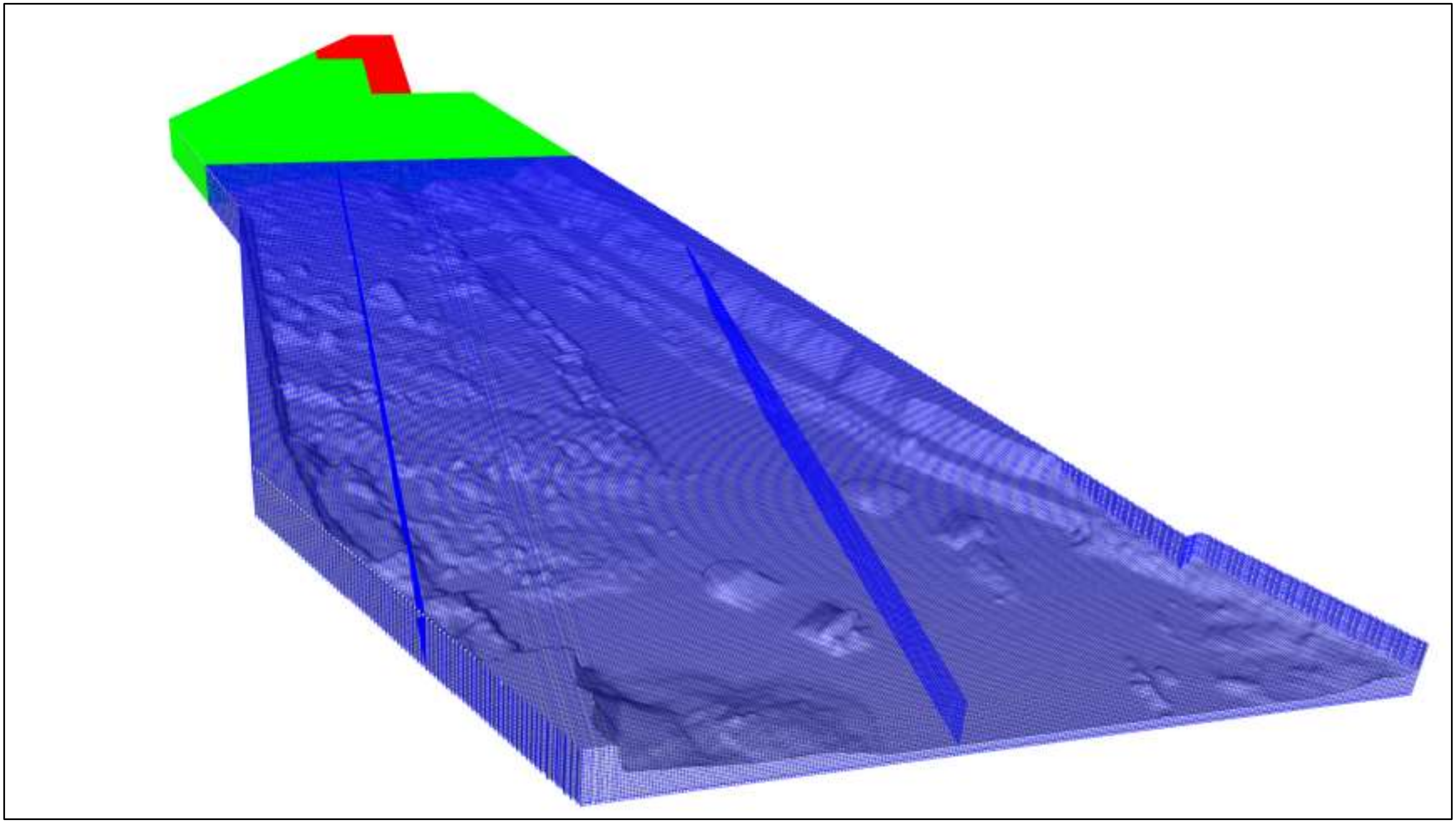

Figure 3-4: Combined Fish Ladder, Dam, and Bathymetry Mesh Refinement Zones

3.1.4 Model Scenarios

Table 3-1 lists the model analysis scenarios for the fish ladder model. The three simulations feature different inflow conditions to vary the hydraulics through the reach. It was assumed the fish ladder and diffuser flow was consistent between all three simulations. The flow (in cubic ft per second [cfs]) over the crest gate was assigned to a mass source using a unit flow rate calculation to evenly distribute flow over the crest.

Table 3-1: Model Analysis Scenarios for the Fish Ladder Model

Case	Fish Ladder (cfs)	Diffuser (cfs)	Crest Weir (cfs)	Sluice (cfs)	Notes:
1	30	60	300	0	Uses 40 ft section of crest weir adjacent to fish ladder
2	30	60	0	360	N/A
3	30	60	19,310	0	5% duration (26,000 cfs) minus E.L. Field Powerhouse operating at full capacity (6,600 cfs)
3	47	60	0	360	5% duration (26,000 cfs) minus E.L. Field Powerhouse operating at full capacity (6,600 cfs)

3.2 E.L. Field Powerhouse Forebay Model

A forebay model was developed to study the hydraulics of flow to the powerhouse. The full model geometry is shown on Figure 3-5.

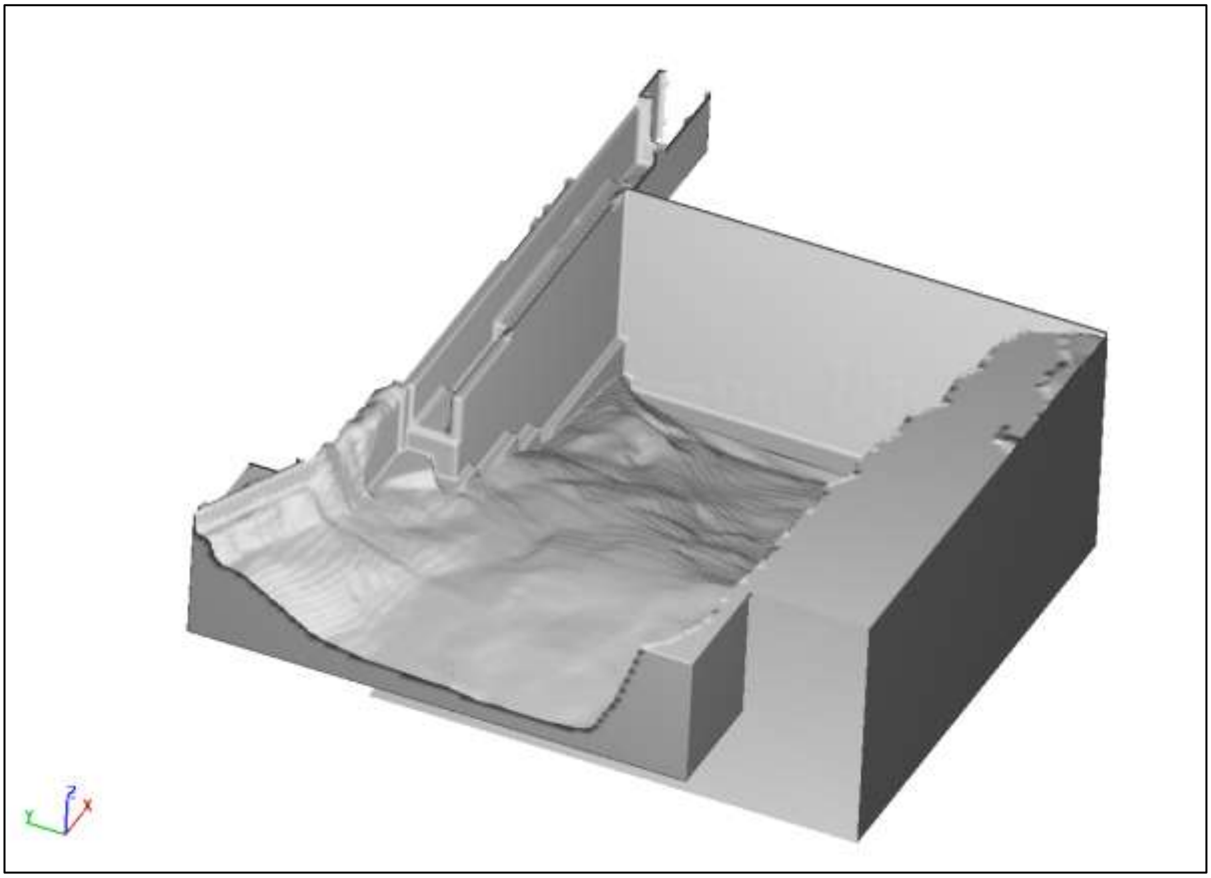

Figure 3-5: E.L. Field Powerhouse Forebay Model Geometry

3.2.1 Mesh Sensitivity Study

A mesh sensitivity study on the forebay model was completed with multiple mesh sizes. The first simulation used a 1.0-ft mesh in all three axial directions. Once the simulation was complete, the mesh was refined to use a 0.5-ft mesh.

The large recirculation patterns and low velocities resulted in similar results for the simulated meshes. The final mesh size was 0.5-ft in all axial directions.

3.2.2 Model Scenarios

Table 3-2 lists the boundary conditions for the forebay model. The forebay model used a constant WSE flow from the powerhouse channel as the upstream boundary condition. Mass sinks were used to allow flow to be removed from the powerhouse tunnels and the auxiliary water supply.

Table 3-2: CFD model scenarios for forebay model

Case	Unit 1 Discharge (cfs)	Unit 2 Discharge (cfs)	AWS Discharge (cfs)	Forebay WSE (ft, NGVD29 [NGVD88])	Notes:
1	3,300	3,300	130	91.0 [90.2]	5% exceedance flow tailwater El. (26,000 cfs)
2	1,310	1,310	130	91.0 [90.2]	75% exceedance flow tailwater El. (2,750 cfs)
3	600	600	130	91.0 [90.2]	Minimum unit operations

3.3 Tailrace Model

A tailrace model was developed to study the hydraulics of the flow leaving both the powerhouse and the auxiliary water supply (AWS). The full model geometry is shown on Figure 3-6. A detailed view of the powerhouse exits and flow directions with approximate powerhouse discharge and AWS boundaries is shown on Figure 3-7.

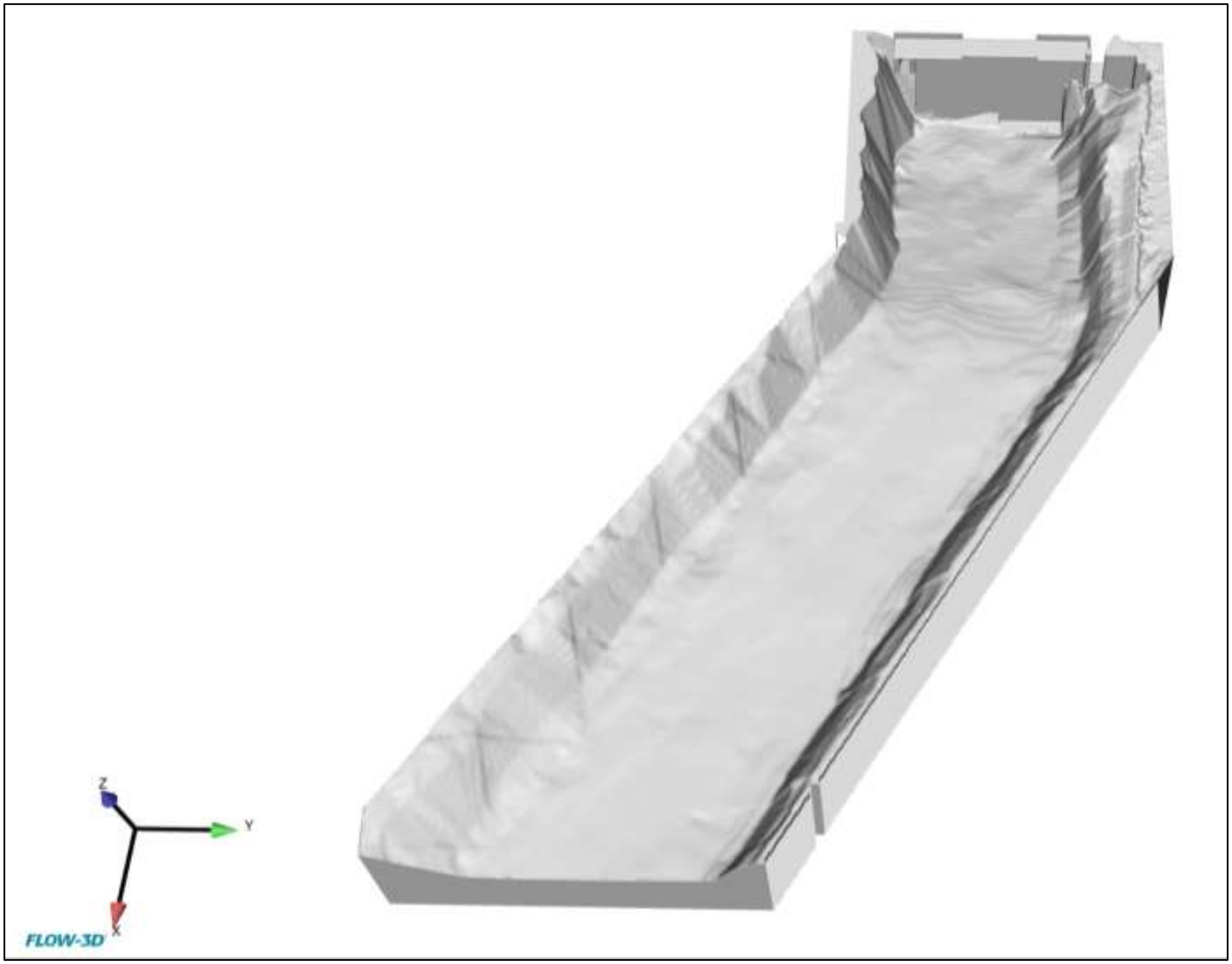

Figure 3-6:Tailrace Model Geometry

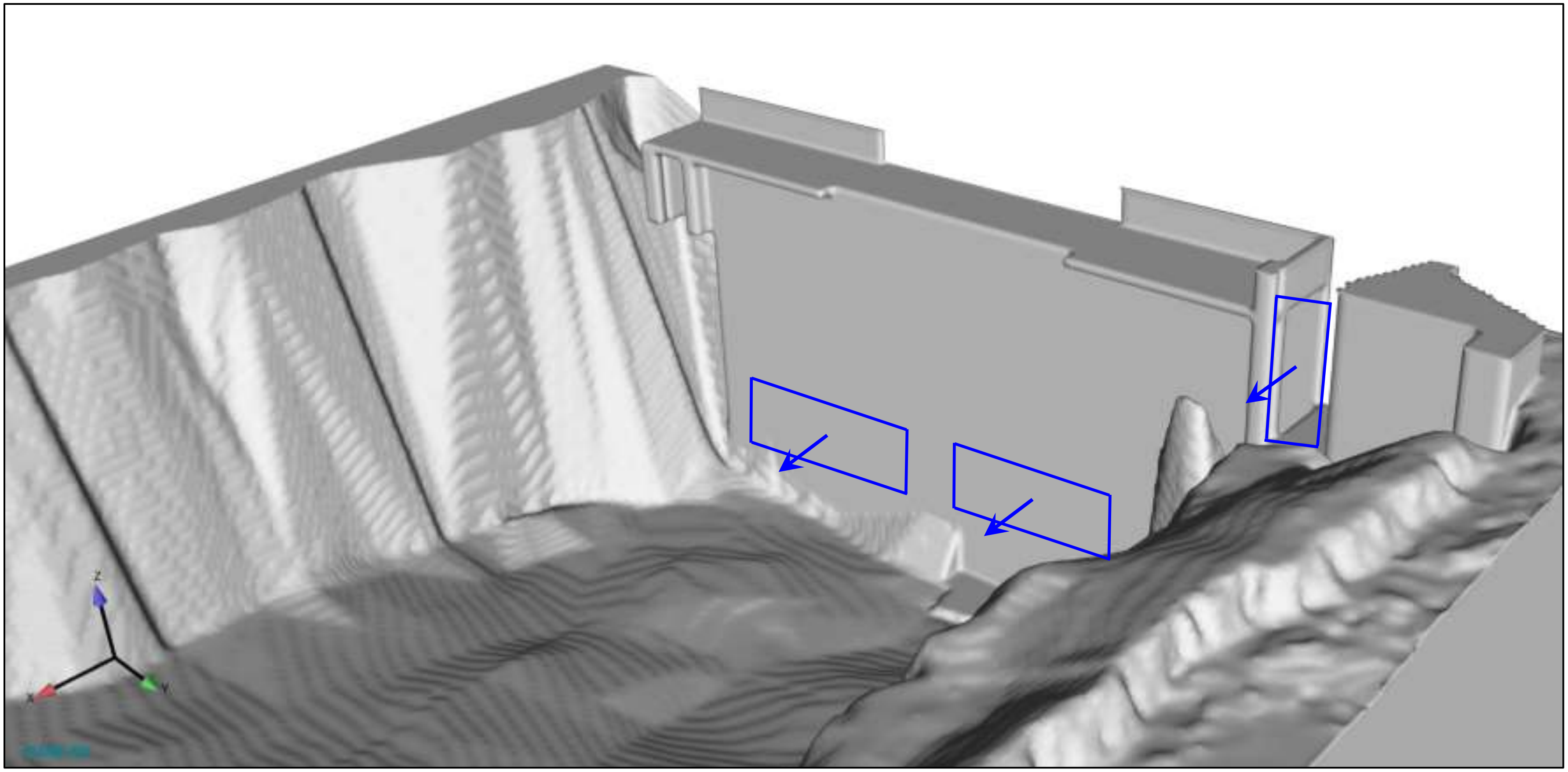

Figure 3-7:Tailrace Model Geometry, ELF Exit and Powerhouse Inflow Boundaries

3.3.1 Mesh Sensitivity Study

Multiple meshes were analyzed to determine if the solution was grid independent. The first grid tested used 1.6-ft spacing in all three axial directions. After completion of the first simulation, the grid was refined to use a 0.8-ft spacing in all three axial directions. Sensitivity to a second refinement was then analyzed. To ensure reasonable computation run times, the model mesh was then refined in the first 100 ft downstream of the powerhouse to 0.4-ft spacing in all three axial directions. The remainder of the domain was left at 0.8-ft spacing for the most refined. Figure 3-8 shows the mesh for the 0.8-ft spacing in the vicinity of the powerhouse.

A 0.8-ft mesh spacing in all directions for the entire tailrace model was chosen for the final analysis.

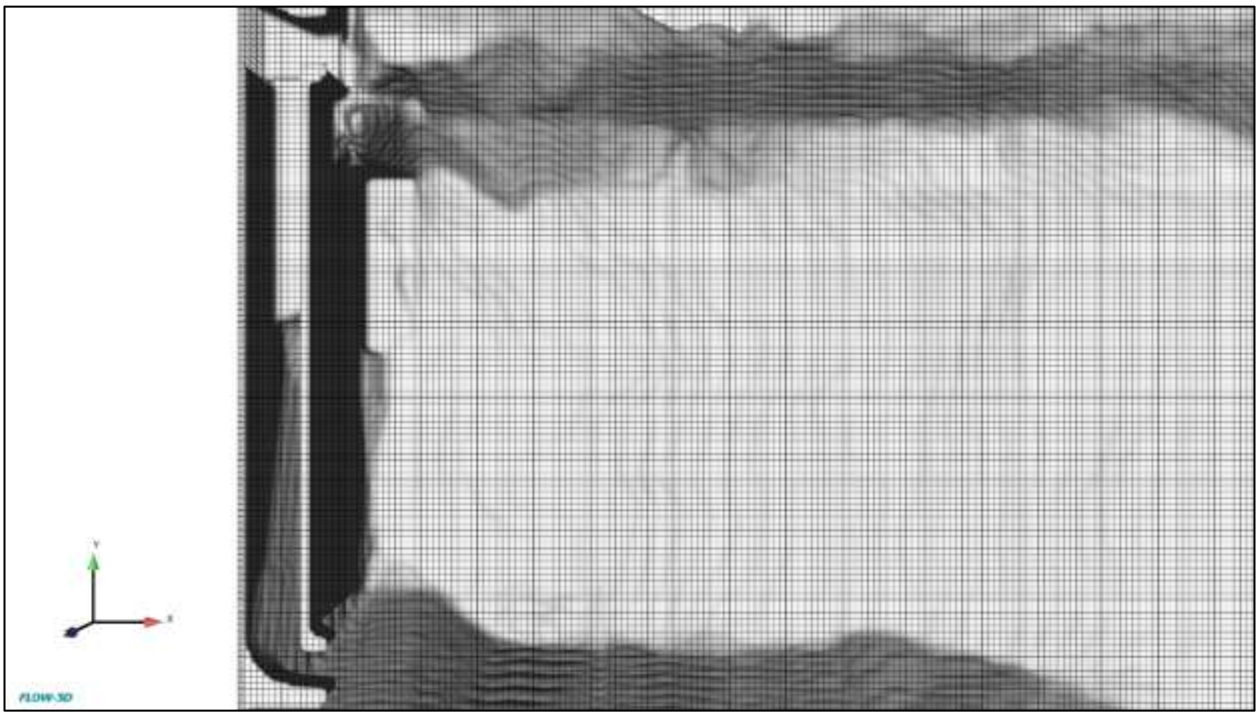

Figure 3-8: Typical Mesh Layout for Tailrace Model

3.3.2 Model Scenarios

Table 3-3 lists the boundary conditions for the tailrace model. The tailrace model used mass sources to add flow from the powerhouse units and the auxiliary water supply.

Table 3-3: CFD model scenarios for tailwater model

Case	Unit 1 Discharge (cfs)	Unit 2 Discharge (cfs)	AWS Discharge (cfs)	Tailwater WSE (ft, NGVD29 [NAVD88])	Notes:
1	3,300	3,300	100	58.36 [57.57]	5% exceedance flow tailwater El. (26,000 cfs)

Case	Unit 1 Discharge (cfs)	Unit 2 Discharge (cfs)	AWS Discharge (cfs)	Tailwater WSE (ft, NGVD29 [NAVD88])	Notes:
2	3,300	3,300	100	53.17 [52.38]	50% exceedance flow tailwater El. (6,800 cfs)

3.4 Model Approach

3.4.1 Boundary Conditions

Boundary conditions for the CFD model were applied through multiple boundary types, as listed below:

VOLUME FLOW INLET

The volume flow inlet allows a specified volume of flow to enter the model and was used at the upstream boundary of the model. A directional vector was applied to the inflow. The water surface was known and was specified for the inflow boundary condition.

PRESSURE INLET/OUTLET

The pressure boundary conditions specify a known pressure and/or WSE for the boundary.

MASS MOMENTUM SOURCE/SINK

The mass momentum source/sink allows a specified volume of flow to enter/exit the model at a specific location not associated with a mesh block boundary. This boundary type was used at the crest weir, auxiliary water supplies (AWS) and powerhouse inlet/outlet. A directional vector and uniform velocity distribution were applied to the inflow.

WALL

The boundary type wall applied the no-slip condition at the outer boundary of the mesh blocks as well as a zero-velocity condition normal to the boundary.

ROUGHNESS HEIGHT

The roughness height for the structures and topography was set to 0.003 ft, consistent with concrete.

3.5 Model Evaluation

Completed model runs were evaluated quantitatively and qualitatively using velocity magnitude and flow streamlines. The CFD model solves the URANS equations, and data presented at a single time step may contain a maximum or minimum value, which may or may not correspond to the anticipated hydraulic characteristics.

FLUX SURFACES

Flux surfaces were used to monitor the volumetric flow through portions of each CFD model. The surfaces were monitored for mass/volume balance of flow through the model.

MONITORING POINTS

Monitoring points were placed within the model to gather point data for the ladder, powerhouse channel, and tailrace. The monitoring points were selected based on their proximity to key model elements.

4 Results

4.1 Fish Ladder

Three fish ladder simulations were identified in the project scope. Plots showing velocity contours and vectors are presented in plan and profile views. Note the profile views are cut through the center of the inner and outer runs of the fish ladder. Streamlines through the fish ladder and in the immediate downstream region are presented in plan view. An isometric view of the water surface colored by velocity is also presented in the vicinity of the downstream weirs.

4.1.1 Case 1

Normal operation flow through the fish ladder was identified for each of the inlet features: diffuser, ladder, and crest gate. Case 1 assigns 300 cfs over the adjacent crest weir, 60 cfs through the diffuser, and 30 cfs through the fish ladder. The sluice gate is closed for Case 2.

Streamlines colored by velocity are shown on Figure 4-1. Figure 4-2 shows velocity vectors and contours. Figure 4-8 and Figure 4-4 show velocity vector and contour profiles for the inside and outside runs of the fish ladder upstream and downstream of the 180º bend in the fish ladder, respectively. Figure 4-5 shows the water surface colored by velocity in the vicinity of the weirs.

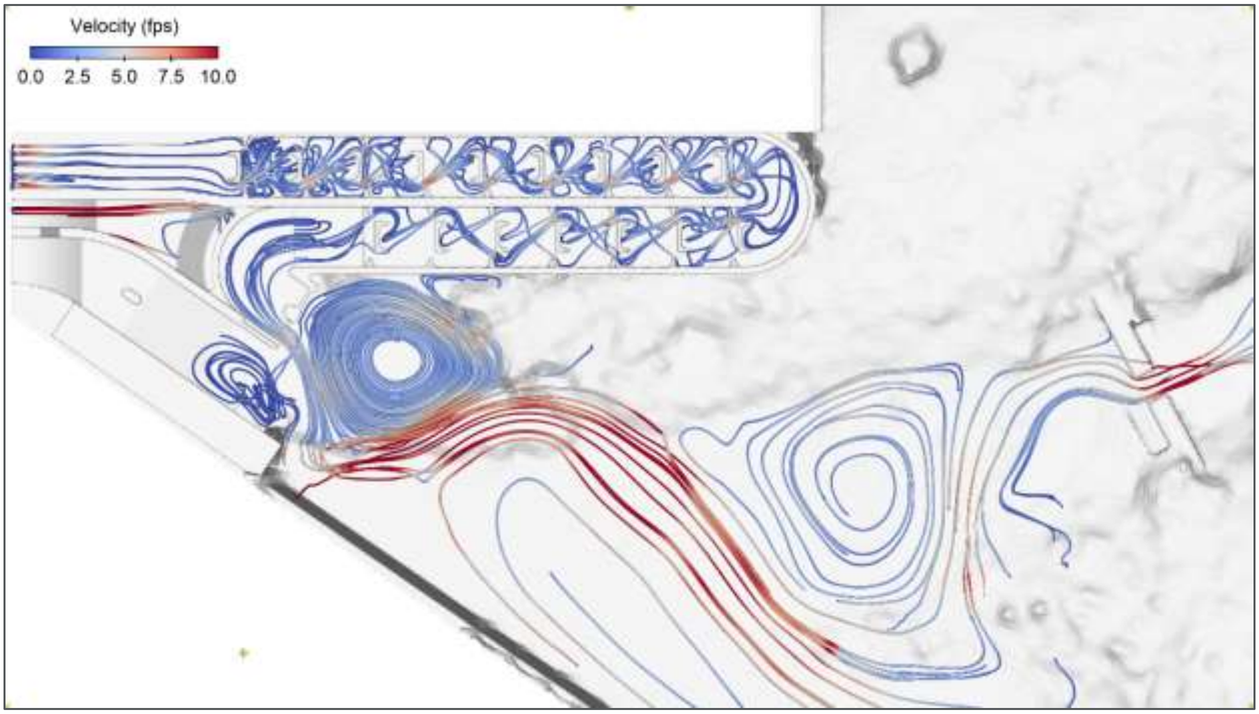

Figure 4-1: Streamlines Colored by Velocity Showing Flow Patterns for Case 1

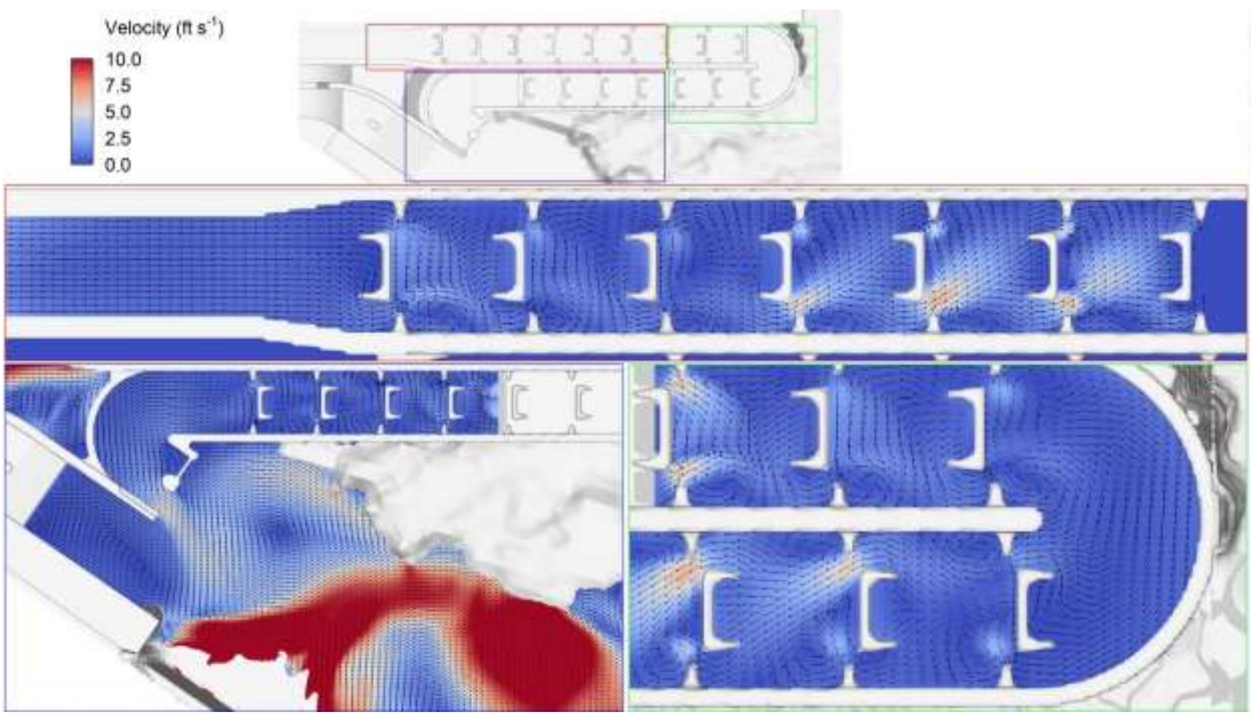

Figure 4-2: Velocity Contours and Vectors for Case 1

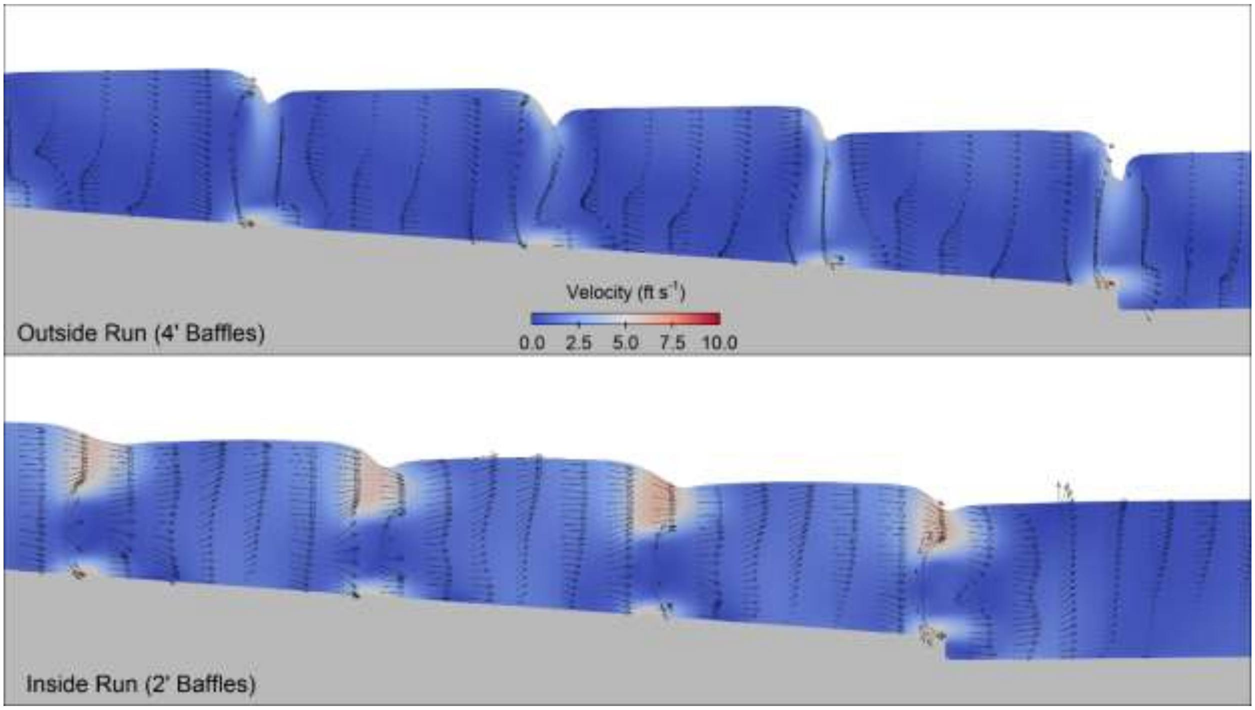

Figure 4-3: Velocity Contour and Vector Profiles Upstream of the 180º Bend

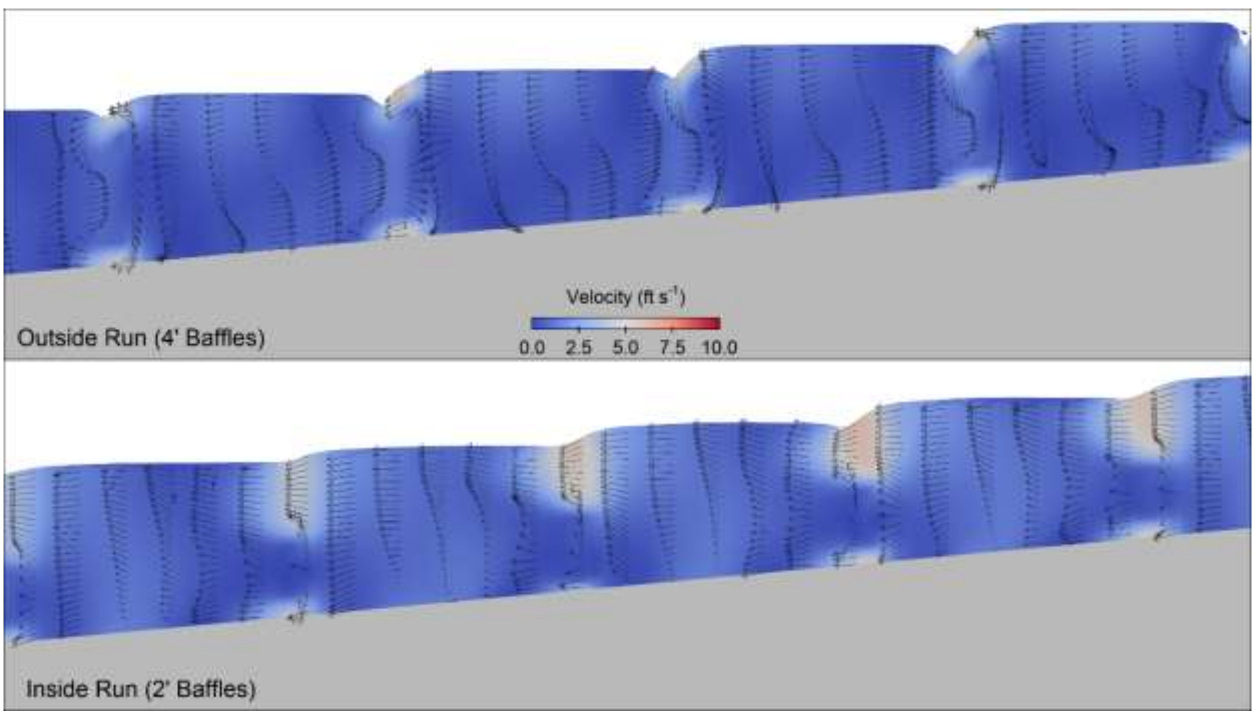

Figure 4-4: Velocity Contour and Vector Profiles Downstream of the 180o Bend

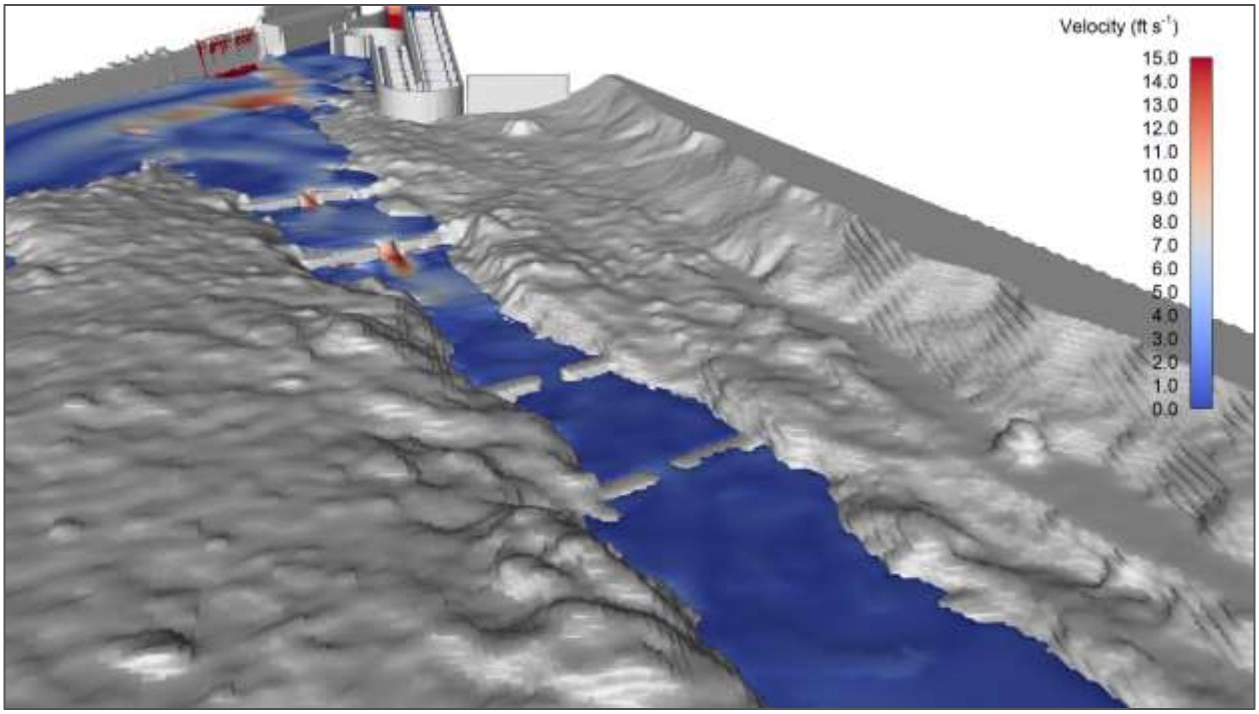

Figure 4-5: Water Surface Colored by Velocity

Flow through the fish ladder showed surface velocities of approximately 6.0 ft per second over the baffles. The vectors and streamlines indicate an area of recirculation on the inside portion of the 180° turn. The depth at the fish ladder entrance was approximately 5.0 to 5.5 ft. Multiple areas of recirculating flow were present downstream of the ladder entrance with a high velocity parallel to the fish ladder entrance. Flow over the bypass weirs does indicate potential surface velocities of about 10.0 ft per second.

4.1.2 Case 2

Normal operation flow through the fish ladder was identified for each of the inlet features: supplemental attraction through the sluice gate, diffuser, and ladder. Case 2 assigns 360 cfs through the sluice gate, 60 cfs through the diffuser, and 30 cfs through the ladder. The crest weir is closed for Case 2.

Streamlines colored by velocity are shown on Figure 4-6. Figure 4-7 shows velocity vectors and contours. Figure 4-8 and Figure 4-9 show velocity vector and contour profiles for the inside and outside runs of the fish ladder upstream and downstream of the 180º bend in the fish ladder, respectively. Figure 4-10 shows the water surface colored by velocity in the vicinity of the weirs.

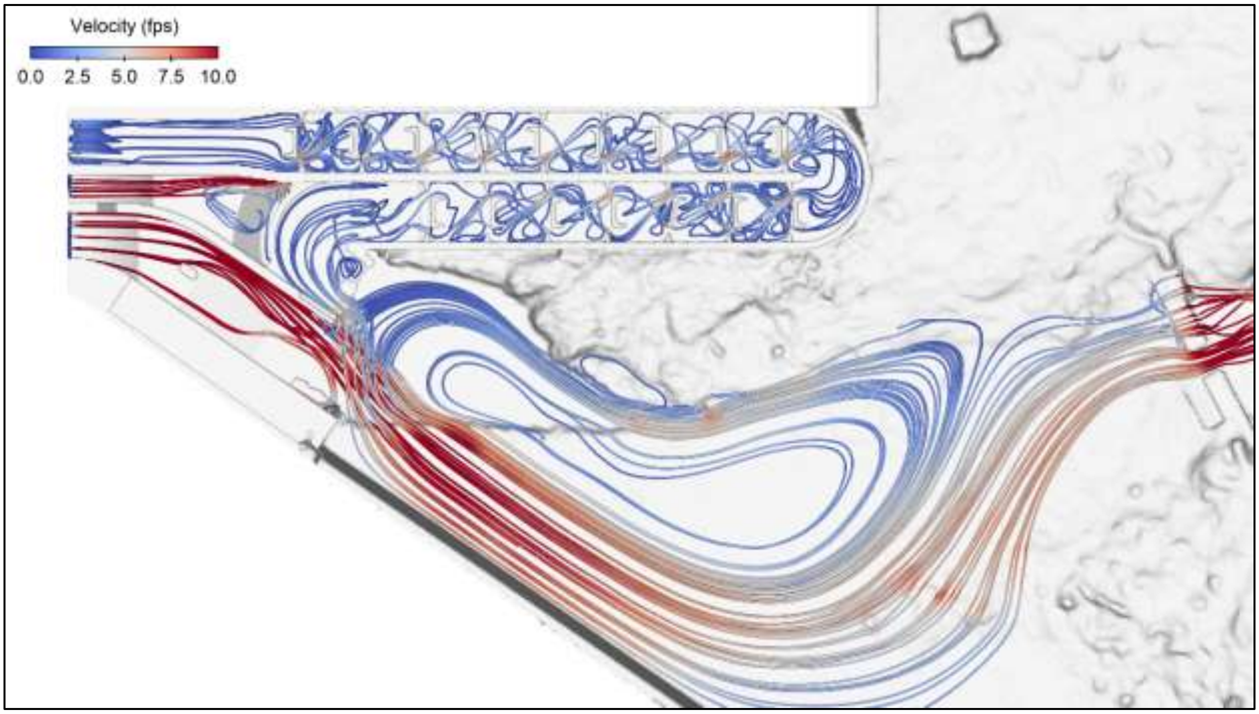

Figure 4-6: Streamlines Colored by Velocity Showing Flow Patterns for Case 2

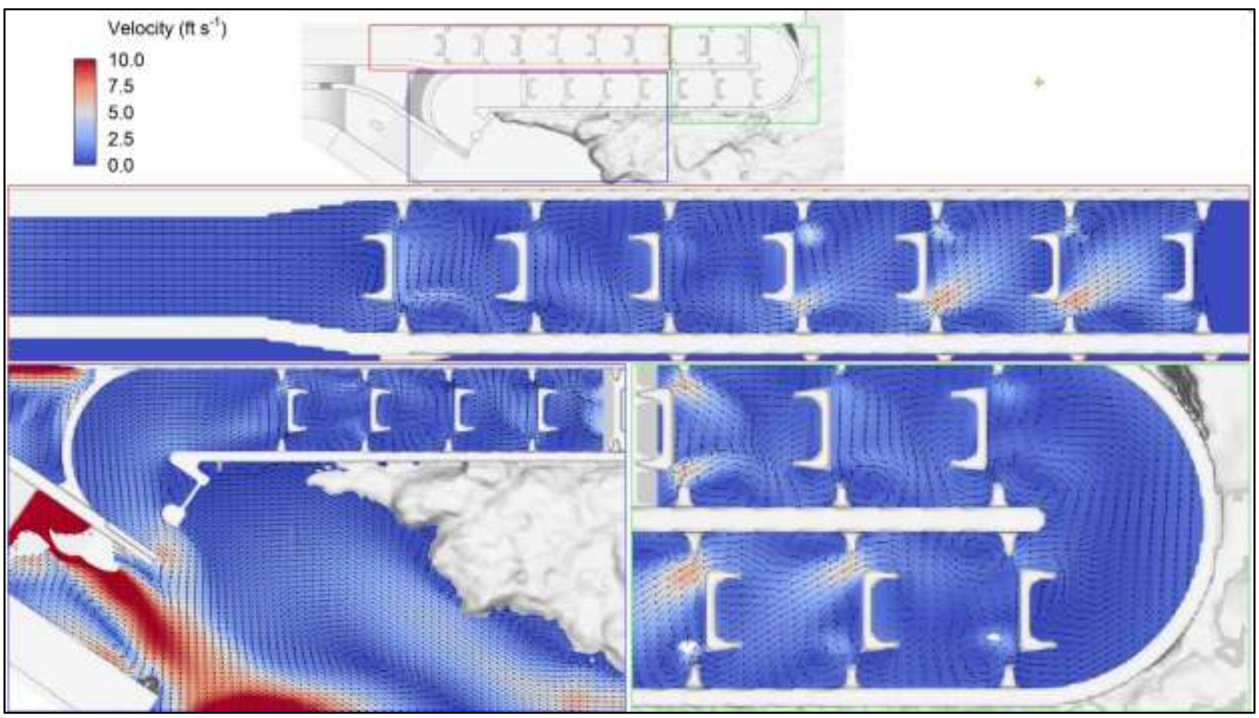

Figure 4-7: Velocity Contours and Vectors for Case 2

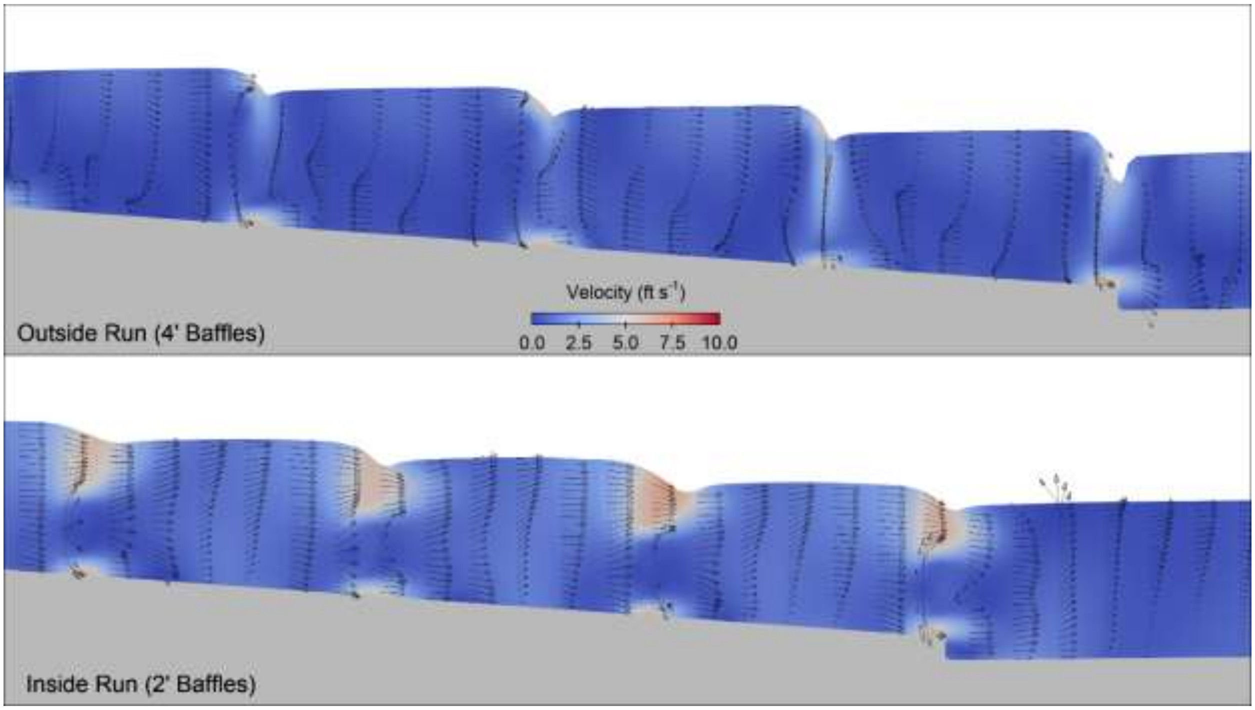

Figure 4-8: Velocity Contour and Vector Profiles Upstream of the 180° Bend

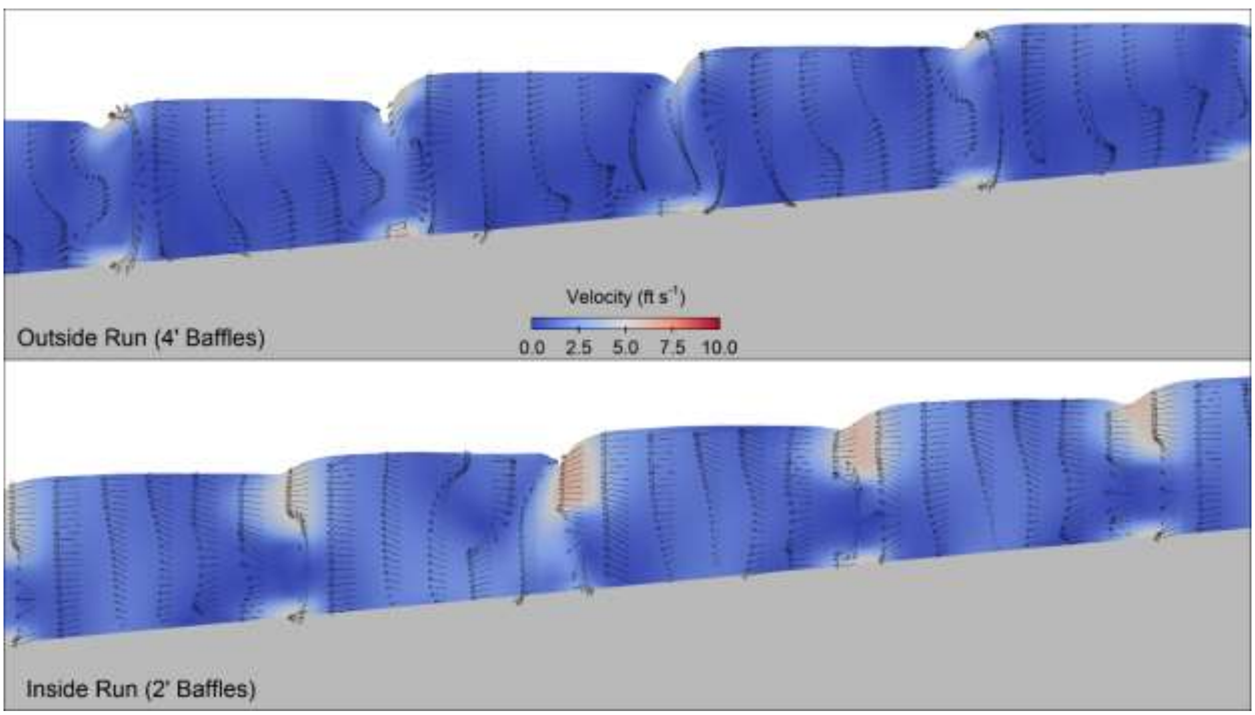

Figure 4-9: Velocity Contour and Vector Profiles Downstream of the 180° Bend

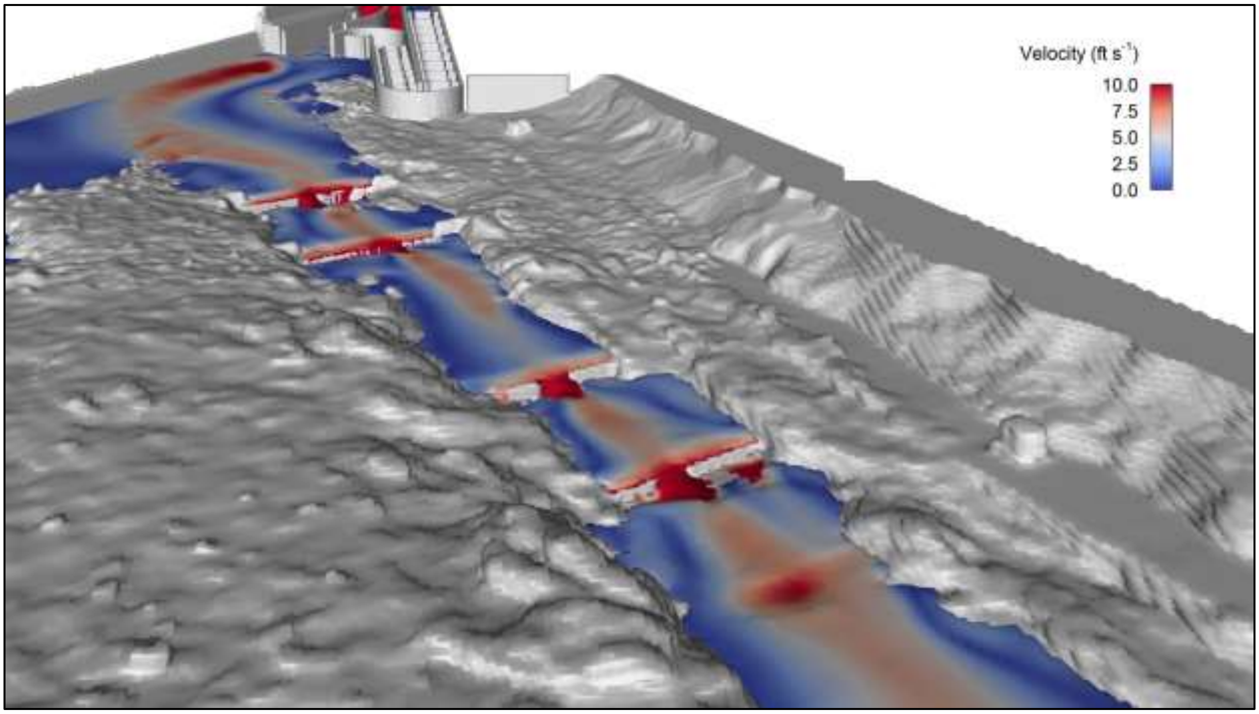

Figure 4-10: Water Surface Colored by Velocity

Flow through the fish ladder showed surface velocities of approximately 6.0 ft per second over the baffles. The vectors and streamlines indicate an area of recirculation on the inside portion of the 180° turn. The depth at the fish ladder entrance was approximately 6.5 ft. Multiple areas of recirculating flow were present downstream of the ladder entrance with a clearly defined flow path from the sluice flow to the first bypass weir. Flow over the bypass weirs does indicate potential surface velocities of about 10.0 ft per second.

4.1.3 Case 3

Case 3 assigns simulates the 5% exceedance flow in the Merrimack River of approximately 26,000 cfs. The powerhouse handles 6,600 cfs, leaving the remaining 19,400 cfs to be passed through the fish ladder and over the spillway crest. The normal operating flow of 30 cfs and 60 cfs were passed through the fish ladder and diffuser gates, respectively. 19,310 cfs is passed equally over the approximately 960 ft spillway crest.

Streamlines colored by velocity are shown on Figure 4-11. Figure 4-12 shows velocity vectors and contours. Figure 4-13 and Figure 4-14 show velocity vector and contour profiles for the inside and outside runs of the fish ladder upstream and downstream of the 180º bend in the fish ladder, respectively. Figure 4-15 shows the water surface colored by velocity in the vicinity of the weirs.

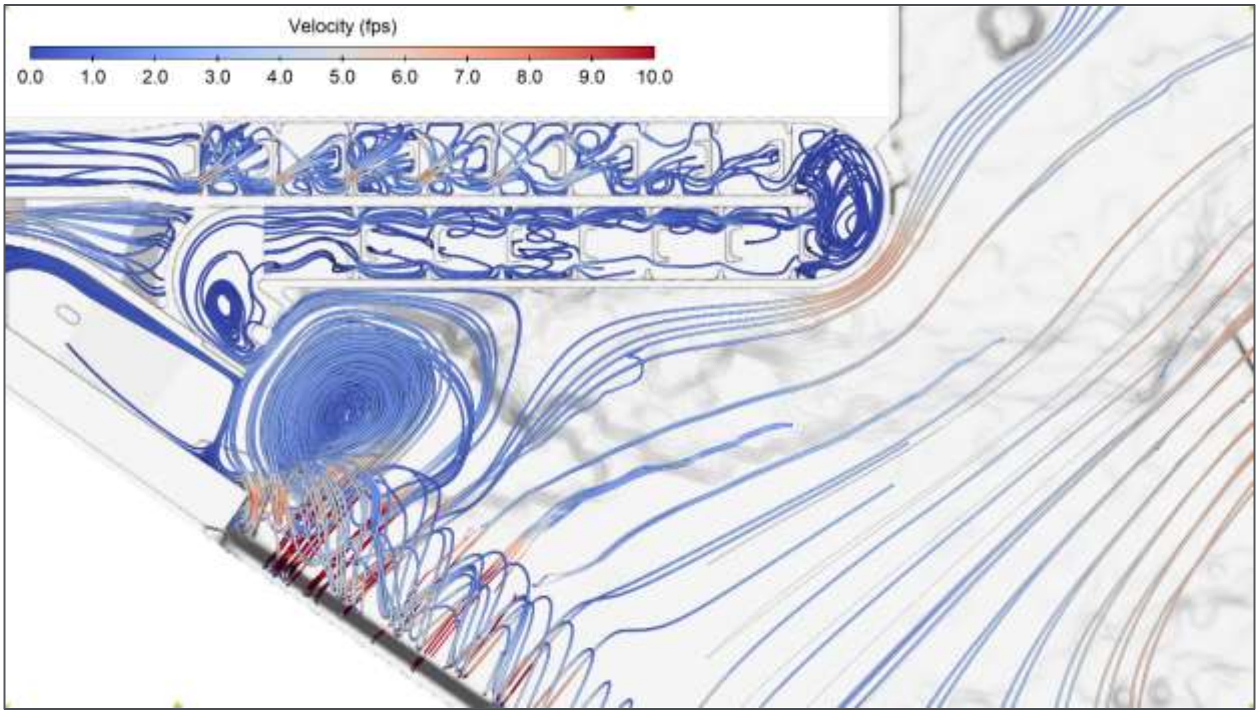

Figure 4-11: Streamlines Colored by Velocity Showing Flow Patterns for Case 3

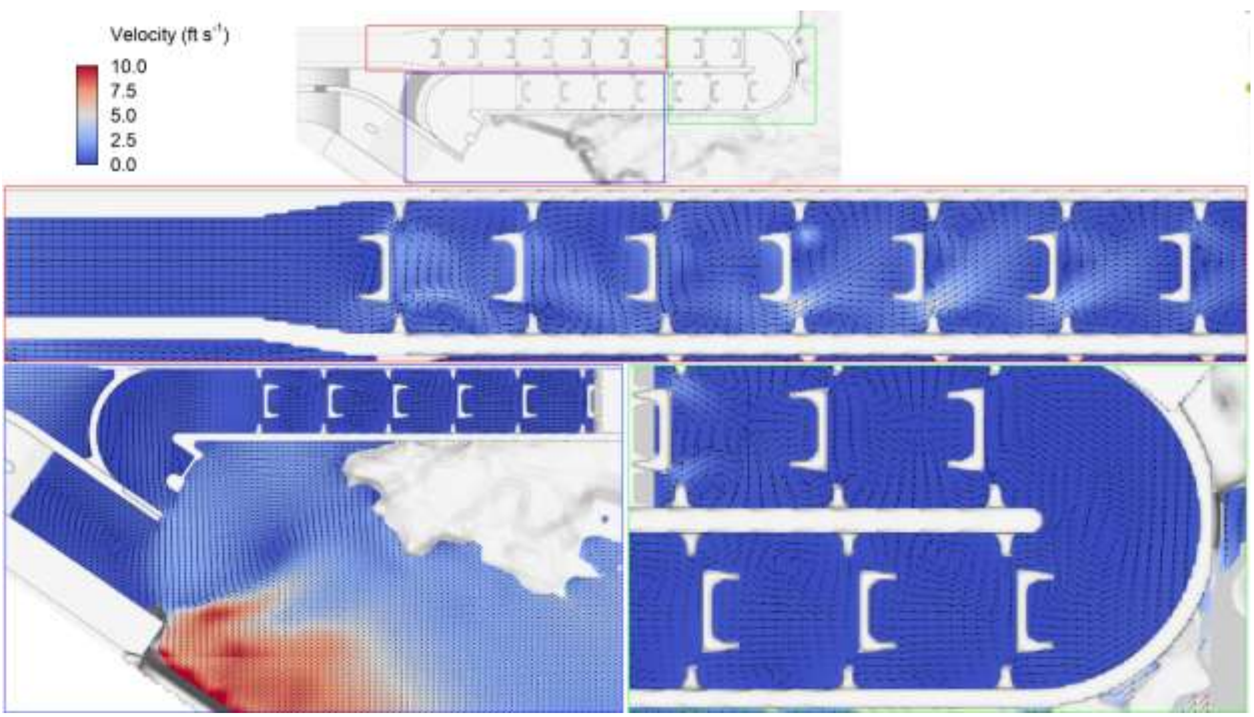

Figure 4-12: Velocity Contours and Vectors for Case 3

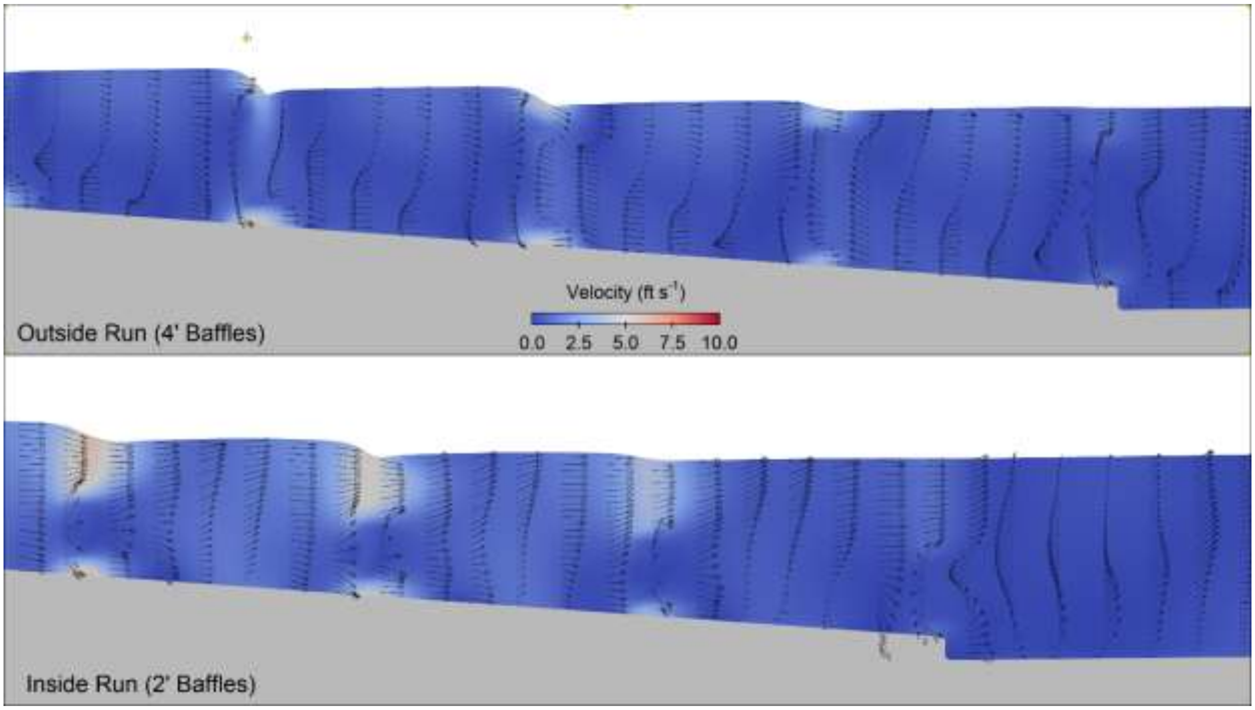

Figure 4-13: Velocity Contour and Vector Profiles Upstream of the 180° Bend

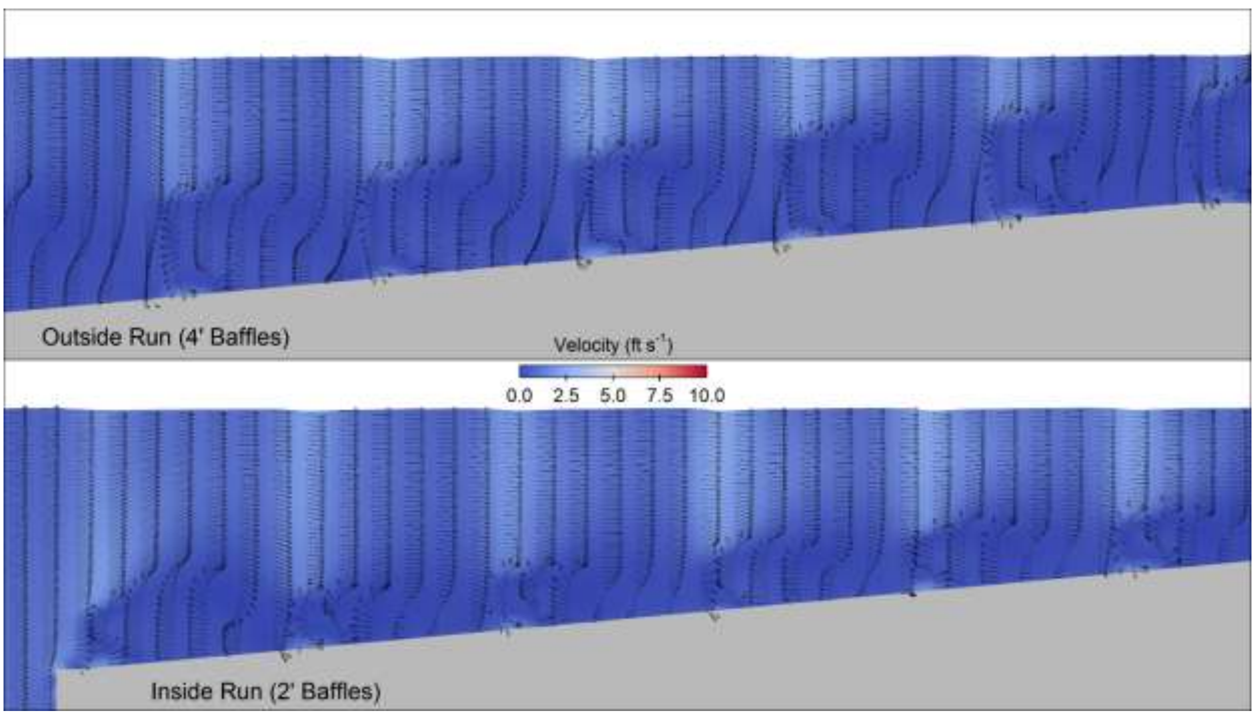

Figure 4-14: Velocity Contour and Vector Profiles Downstream of the 180° Bend

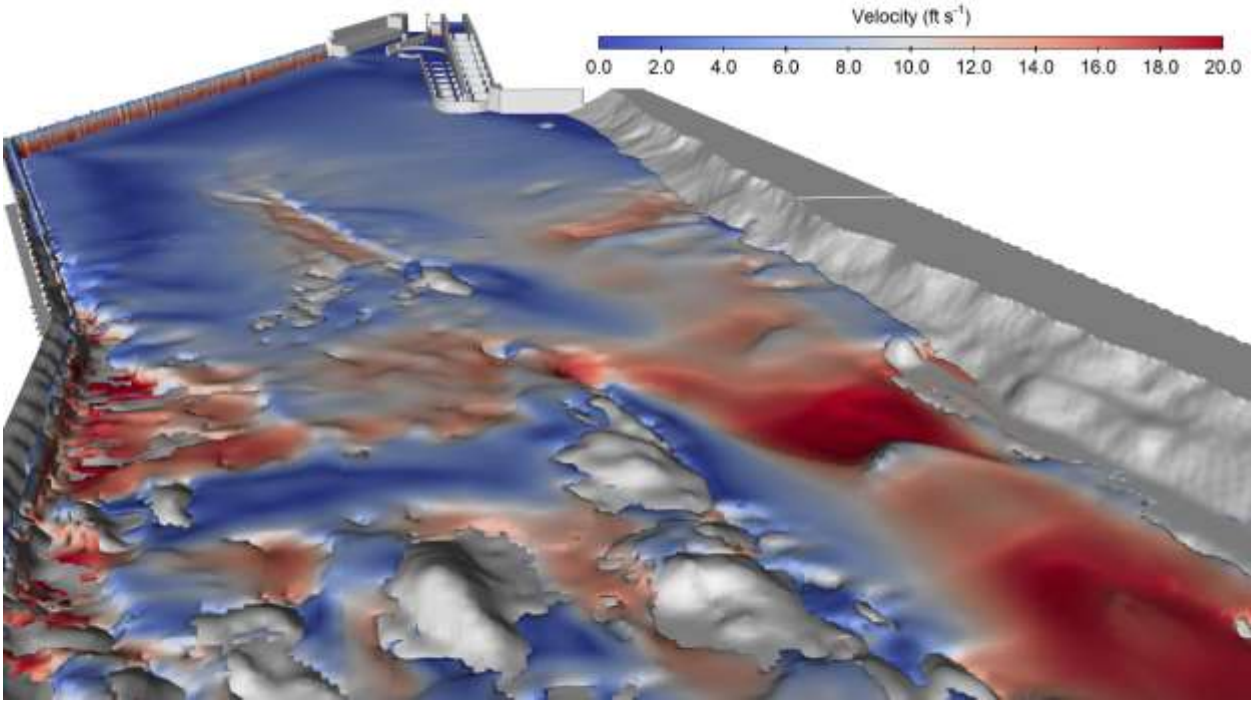

Figure 4-15: Water Surface Colored by Velocity

Flow through the fish ladder showed surface velocities of approximately 6.0 ft per second over the baffles. The vectors and streamlines indicate an area of recirculation on the inside portion of the 180° turn. The depth at the fish ladder entrance was approximately 12.5 ft. Multiple areas of recirculating flow were present downstream of the ladder entrance with a clearly defined flow path from the sluice flow to the first bypass weir. Flow over the bypass weirs does indicate potential surface velocities of about 10.0 ft per second.

4.1.4 Case 4

Case 4 is an update to Case 2 with the fish ladder flow being adjusted to 47 cfs. Case 4 assigns 360 cfs through the sluice gate, 60 cfs through the diffuser, and 47 cfs through the ladder. The crest weir is closed for Case 4.

Streamlines colored by velocity are shown in Figure 4-16 and stream lines colored by source are shown in Figure 4-17. Figure 4-18 shows velocity vectors and contours. Figure 4-19 and Figure 4-20 show velocity vector and contour profiles for the inside and outside runs of the fish ladder upstream and downstream of the 180º bend in the fish ladder, respectively. The water surface profile and depths in the fish ladder are shown in

Figure 4-21. Figure 4-22 shows the water surface colored by velocity in the vicinity of the weirs. The depth and 3ft of depth contour are shown in Figure 4-23. The depth and water surface profile through the bypass weirs is shown in Figure 4-24.

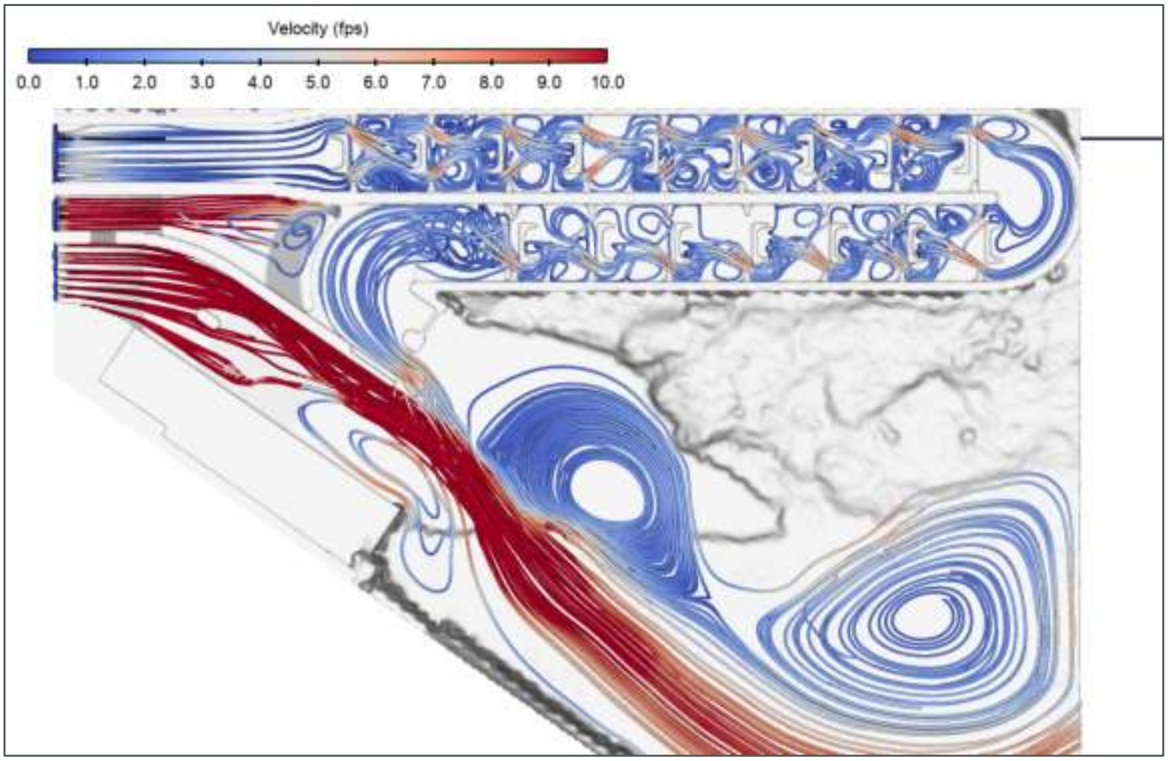

Figure 4-16: Streamlines Colored by Velocity Showing Flow Patterns for Case 4

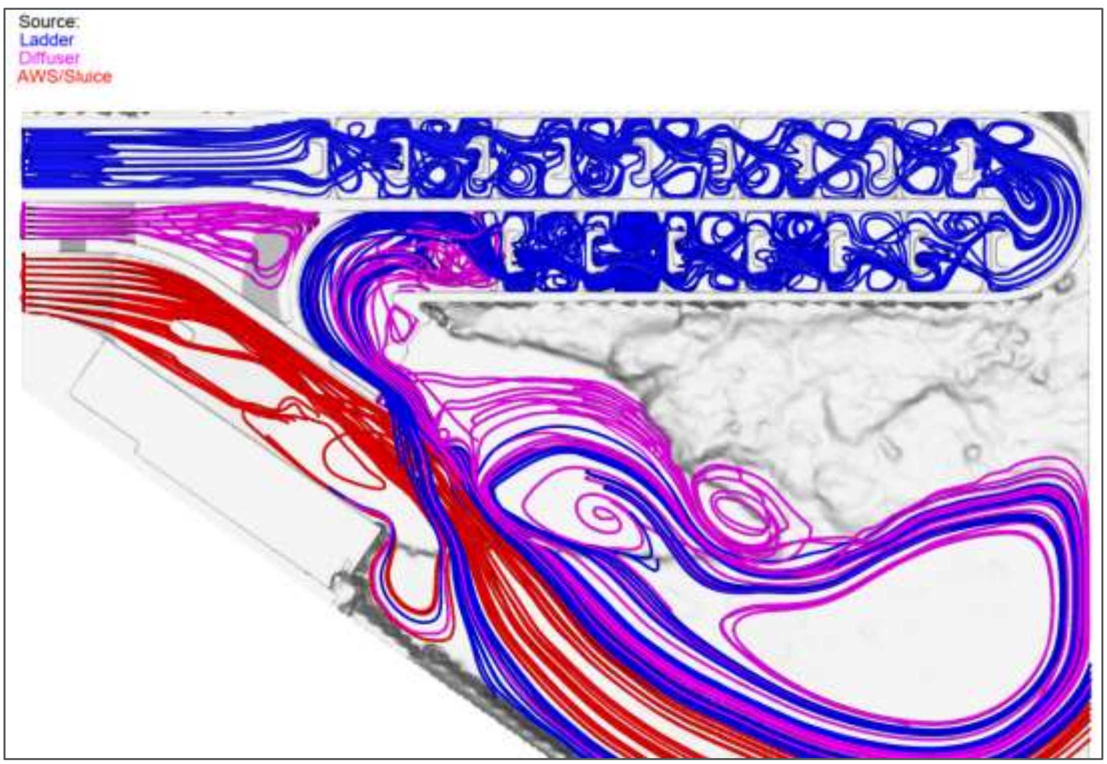

Figure 4-17: Streamlines Colored by Source Showing Flow Patterns for Case 4

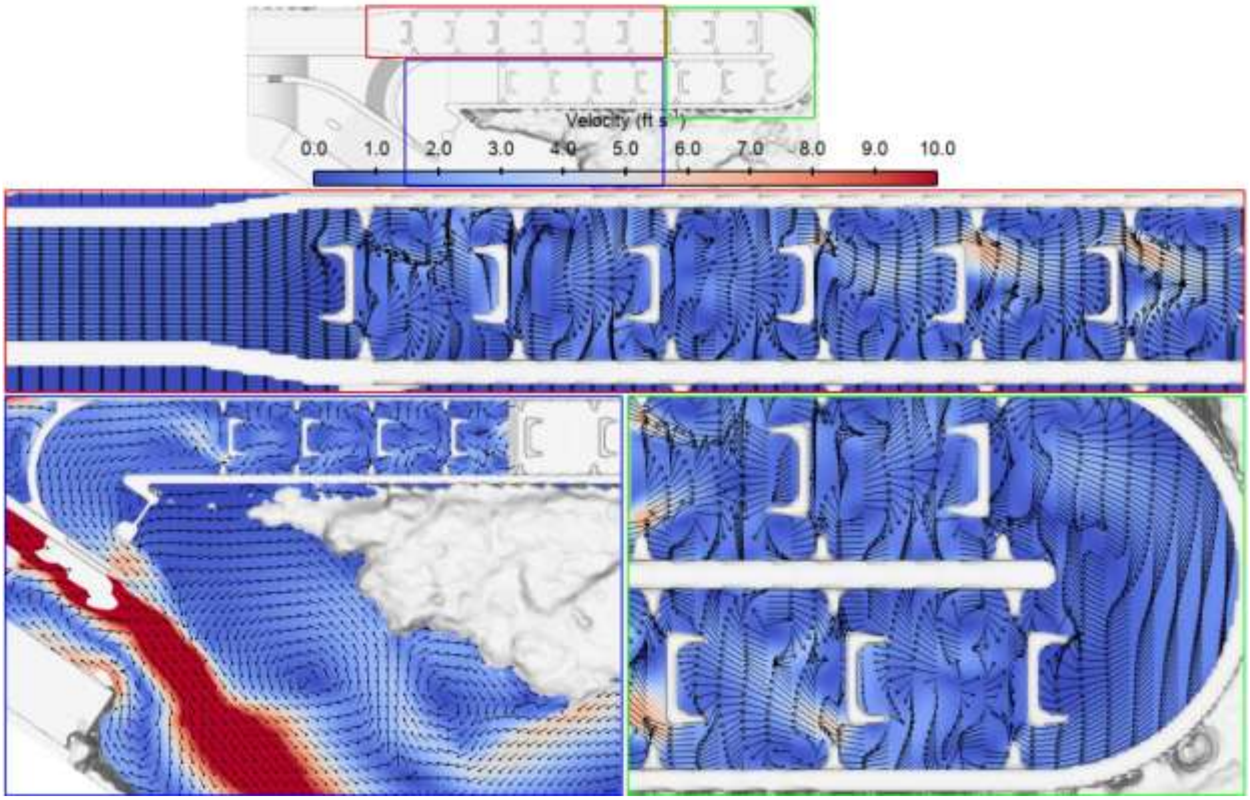

Figure 4-18: Velocity Contours and Vectors for Case 4

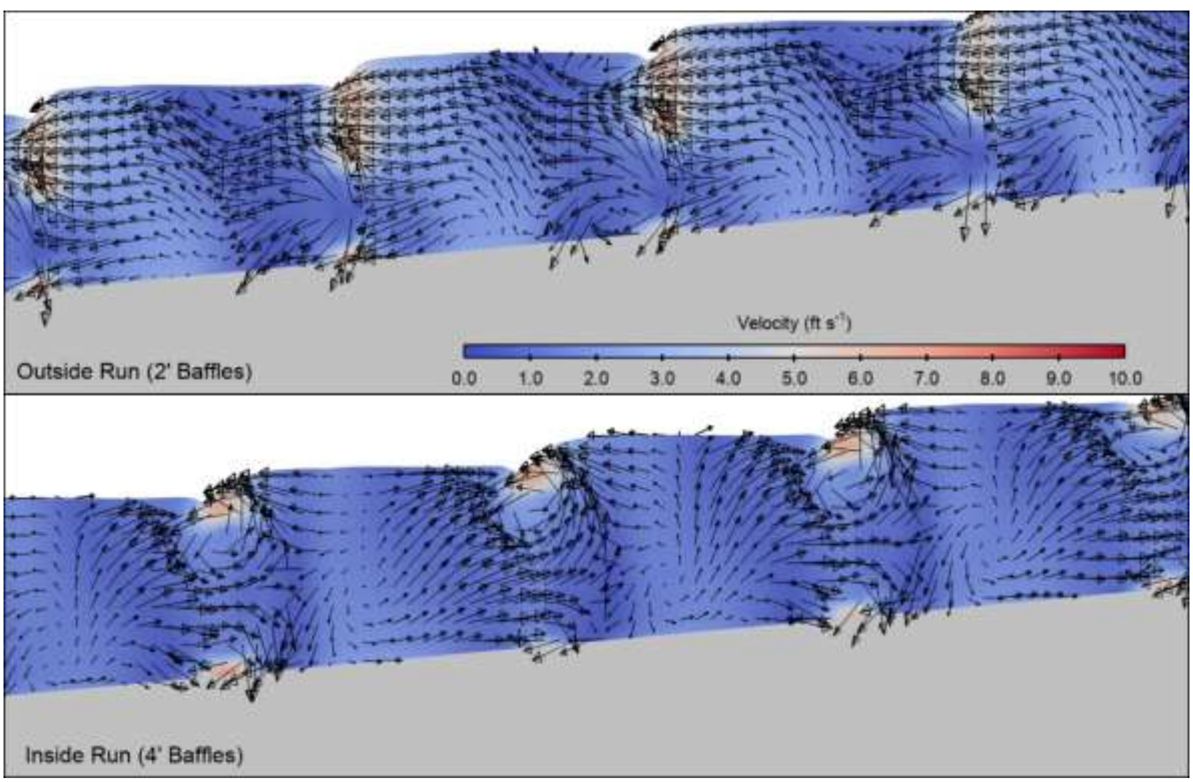

Figure 4-19: Velocity Contour and Vector Profiles Upstream of the 180º Bend

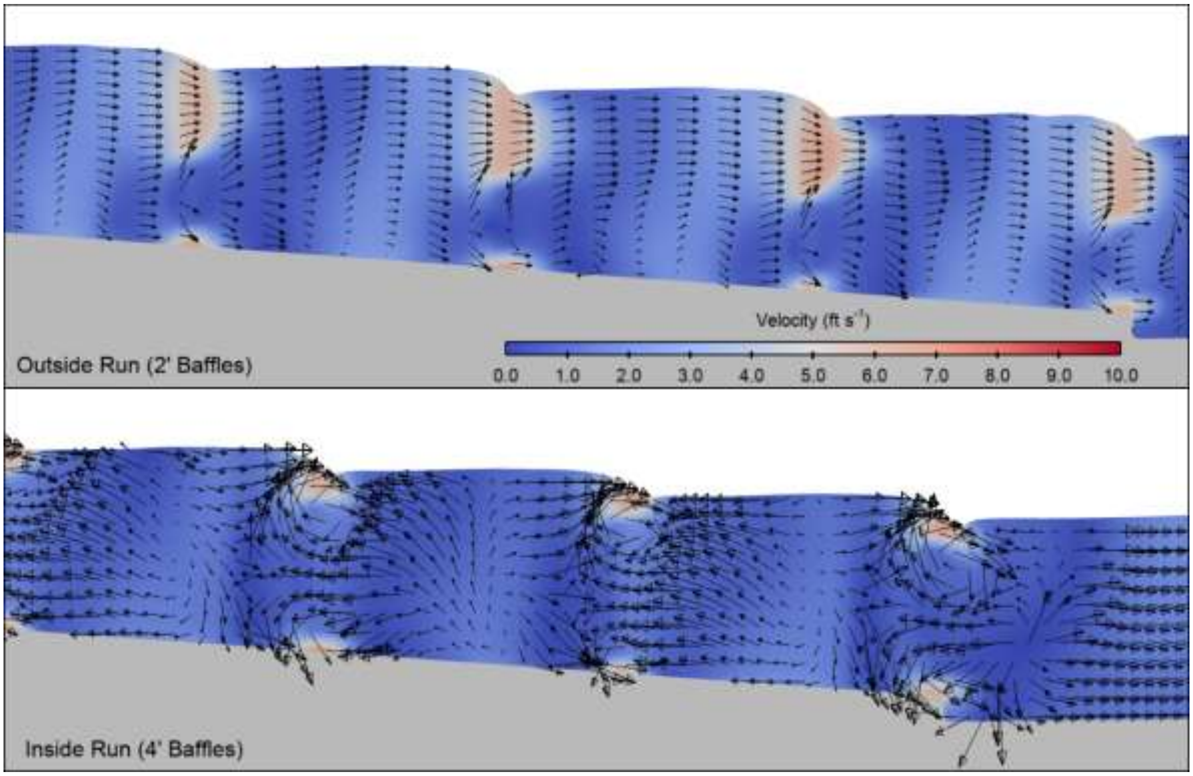

Figure 4-20: Velocity Contour and Vector Profiles Downstream of the 180º Bend

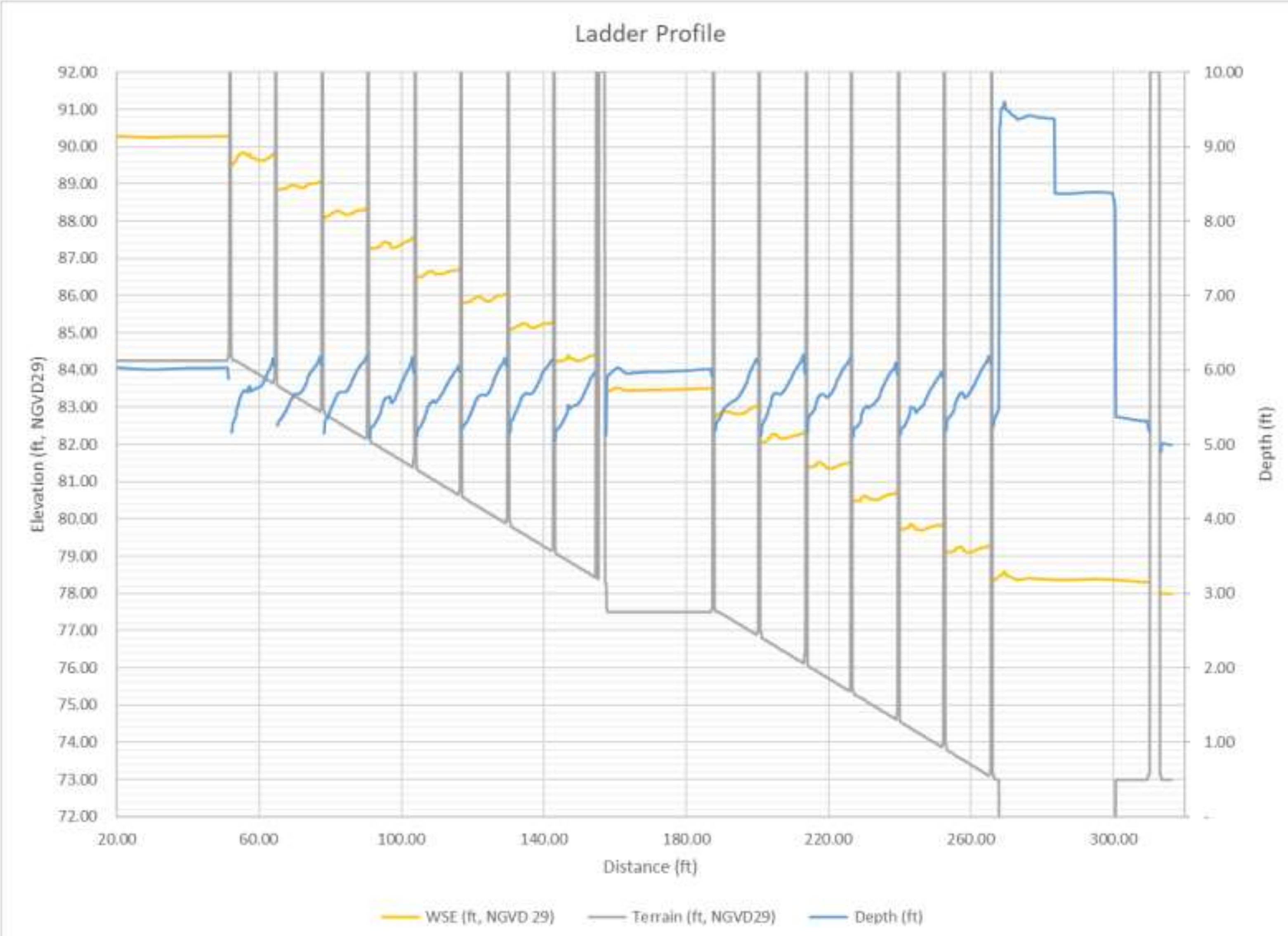

Figure 4-21: Water Surface Profiles and Depths through the Fish Ladder

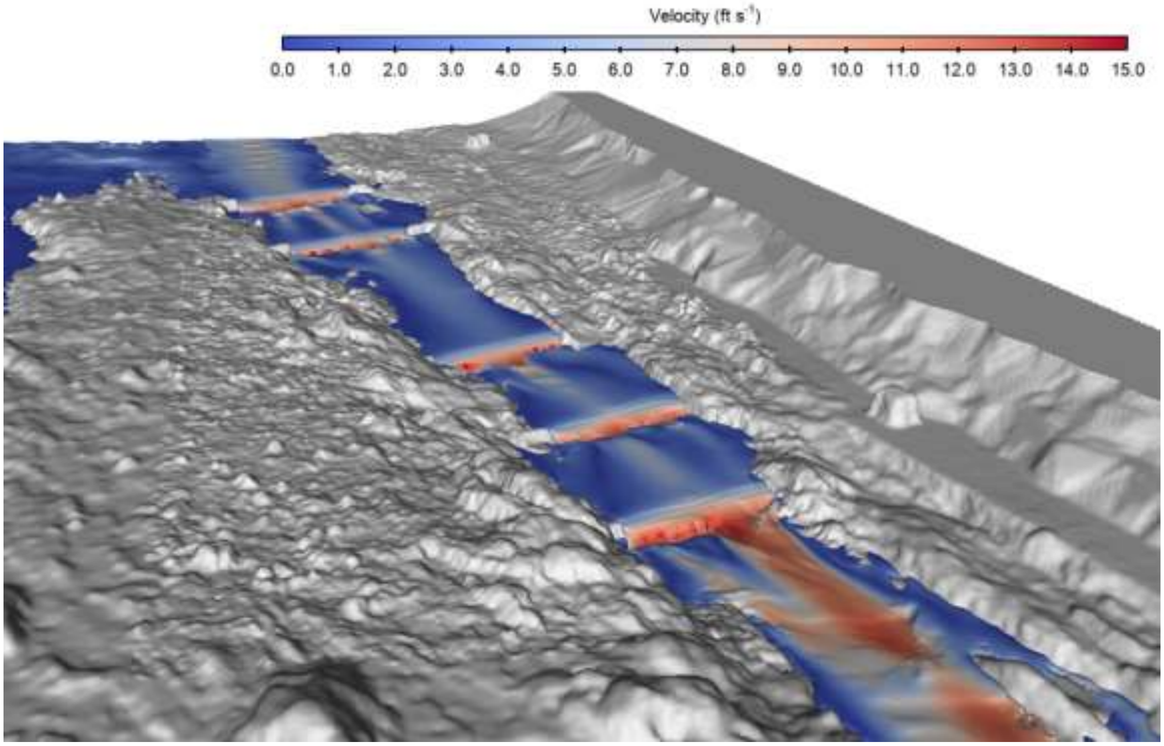

Figure 4-22: Water Surface Colored by Velocity

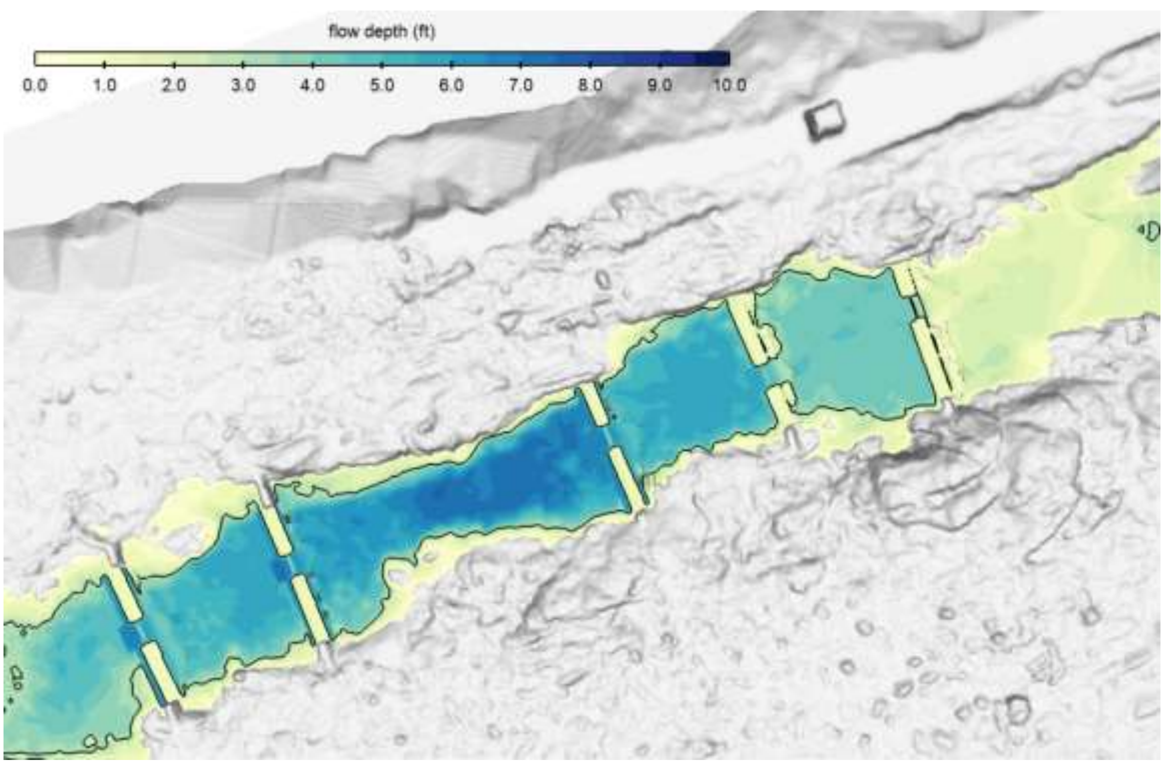

Figure 4-23: Water Depth and 3-foot contour through the Bypass Weirs

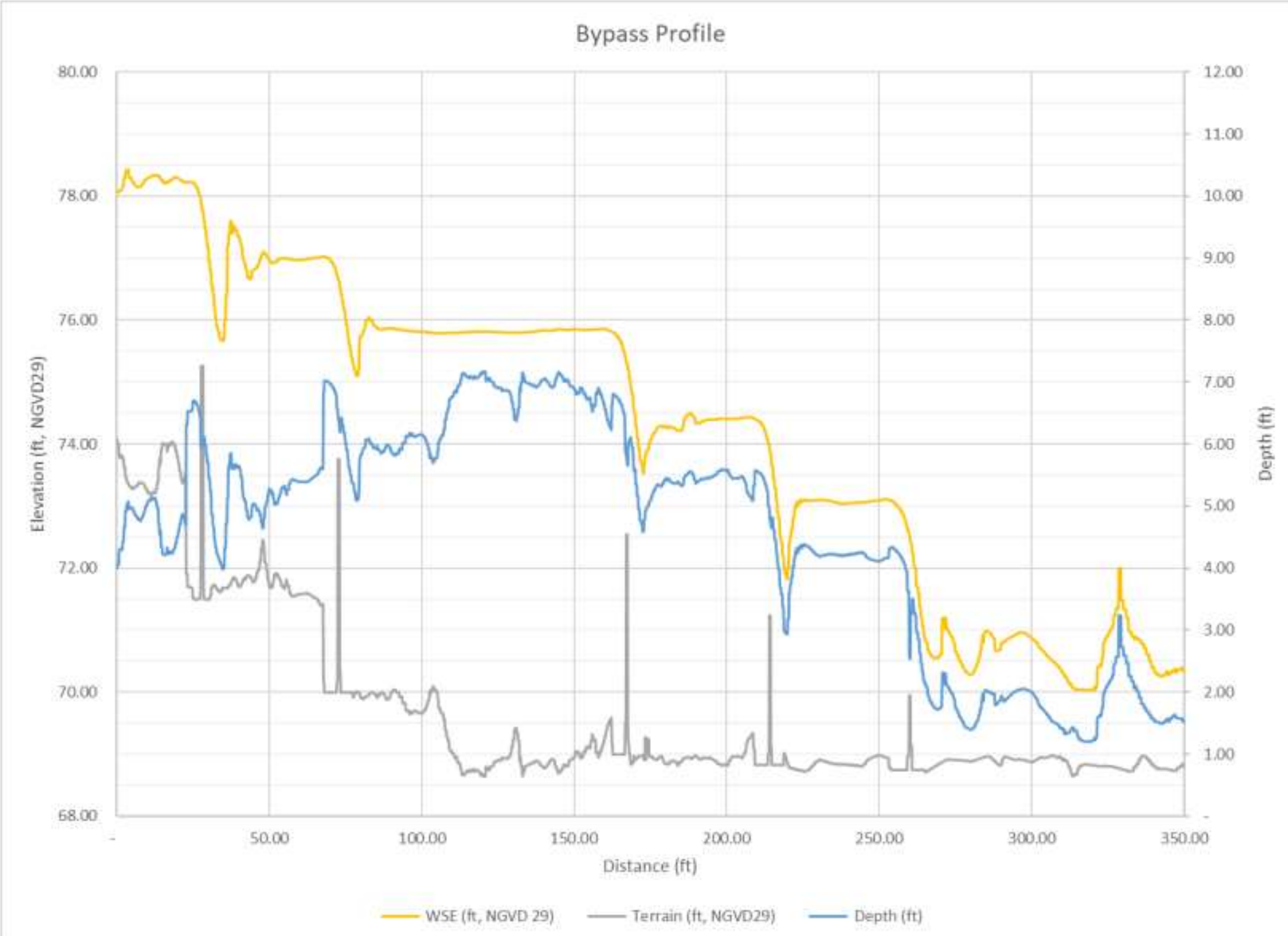

Figure 4-24: Water Surface Profiles and Depths through the Bypass Weirs

Flow through the fish ladder showed surface velocities of approximately 6.0 ft per second over the baffles. The vectors and streamlines indicate an area of recirculation on the inside portion of the 180° turn. The depth at the fish ladder entrance was approximately 5 ft. The water surface profile through the fish ladder showed pool to pool drops of approximately 1 ft with a depth of about 5.5 to 6 ft. Multiple areas of recirculating flow were present downstream of the ladder entrance with a clearly defined flow path from the sluice flow to the first bypass weir. Flow over the bypass weirs does indicate potential surface velocities of about 10.0 to 12.0 ft per second.

4.2 E.L. Field Powerhouse Forebay Model

The three simulations were analyzed and plots showing velocity contours and vectors at multiple depths were exported. The figures depicting streamlines are colored by velocity and model outlet. The red colored streamlines exit the model domain through the AWS and the cyan lines would discharge through the powerhouse.

4.2.1 Case 1 – Full Station Capacity

The 5 percent exceedance model assigns 3,300 cfs through each unit for a total powerhouse flow of 6,600 cfs. The AWS is operating at 130 cfs under this scenario. Figure 4-25 shows the velocity contours and vectors at the water surface (El. 91 ft, NGVD29) and depths of 1 ft (El. 90 ft, NGVD29), 6 ft (El. 85 ft, NGVD29), and 11 ft (El. 85 ft, NGVD29). The streamlines colored by velocity are shown on Figure 4-26. Figure 4-27 shows streamlines colored by model outlet. Figure 4-28 through Figure 4-33 show velocity contour or vectors on multiple sections in the forebay.

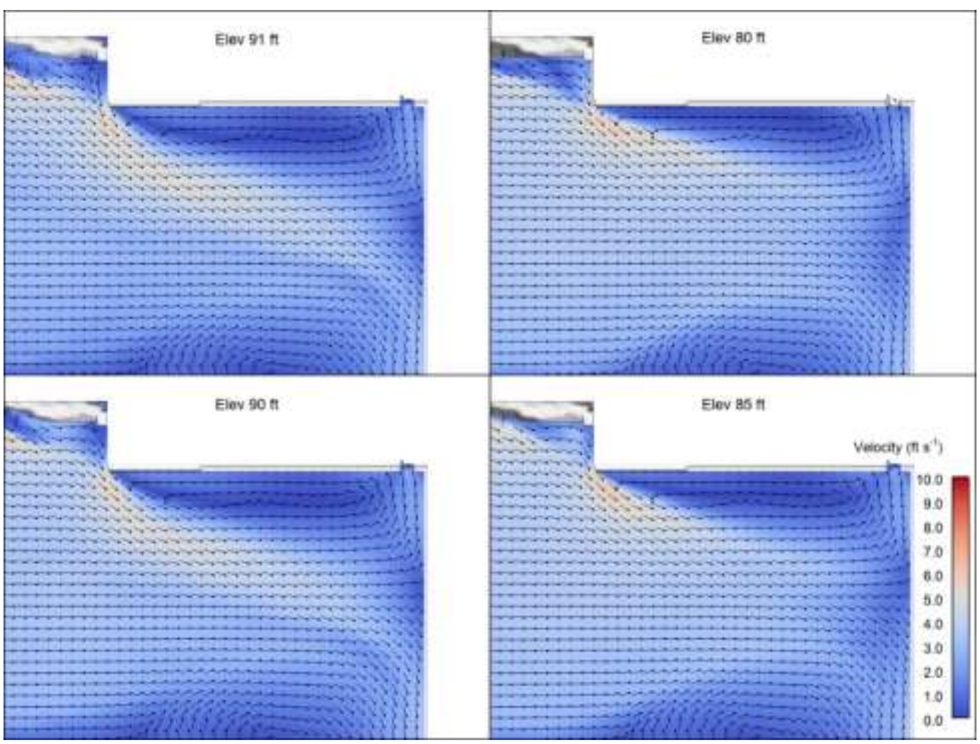

Figure 4-25: Velocity Contours and Vectors for Case 1

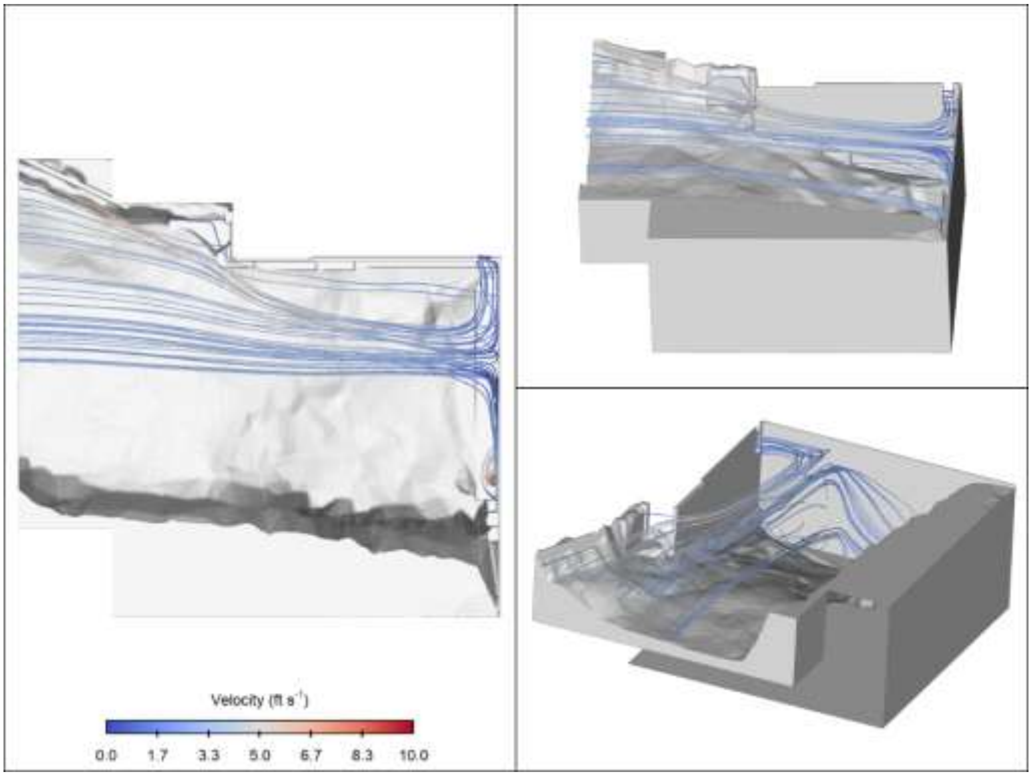

Figure 4-26: Streamlines Colored by Velocity Showing Flow Patterns

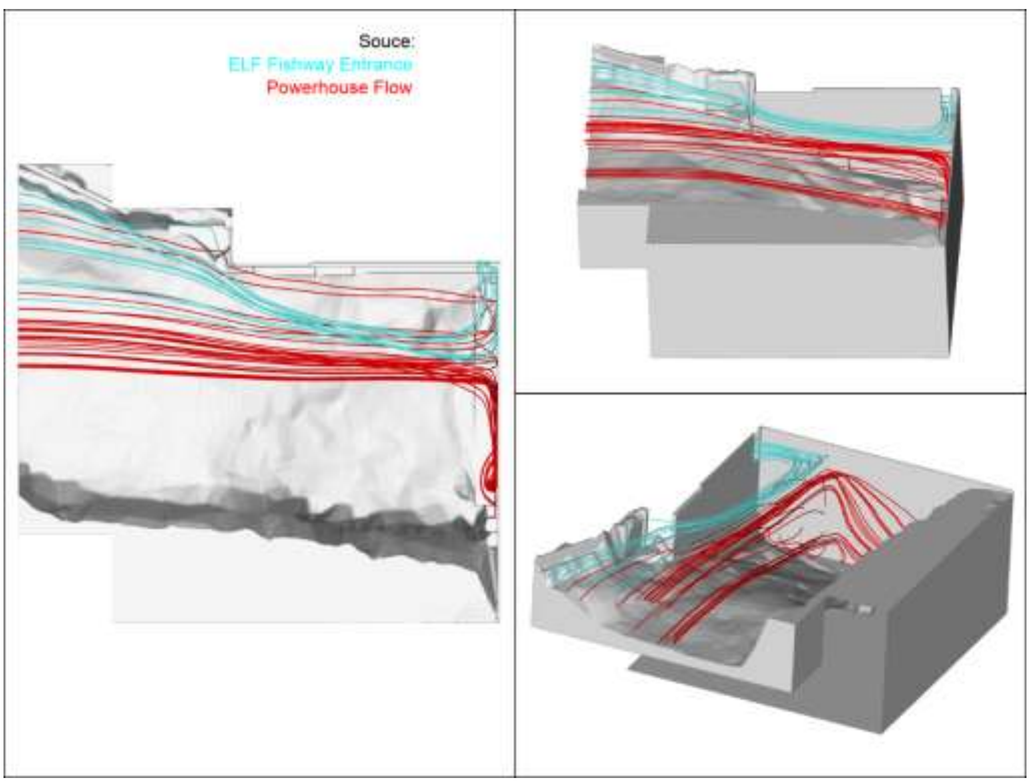

Figure 4-27: Streamlines Colored by Model Outlet Showing Flow Patterns for Case 1

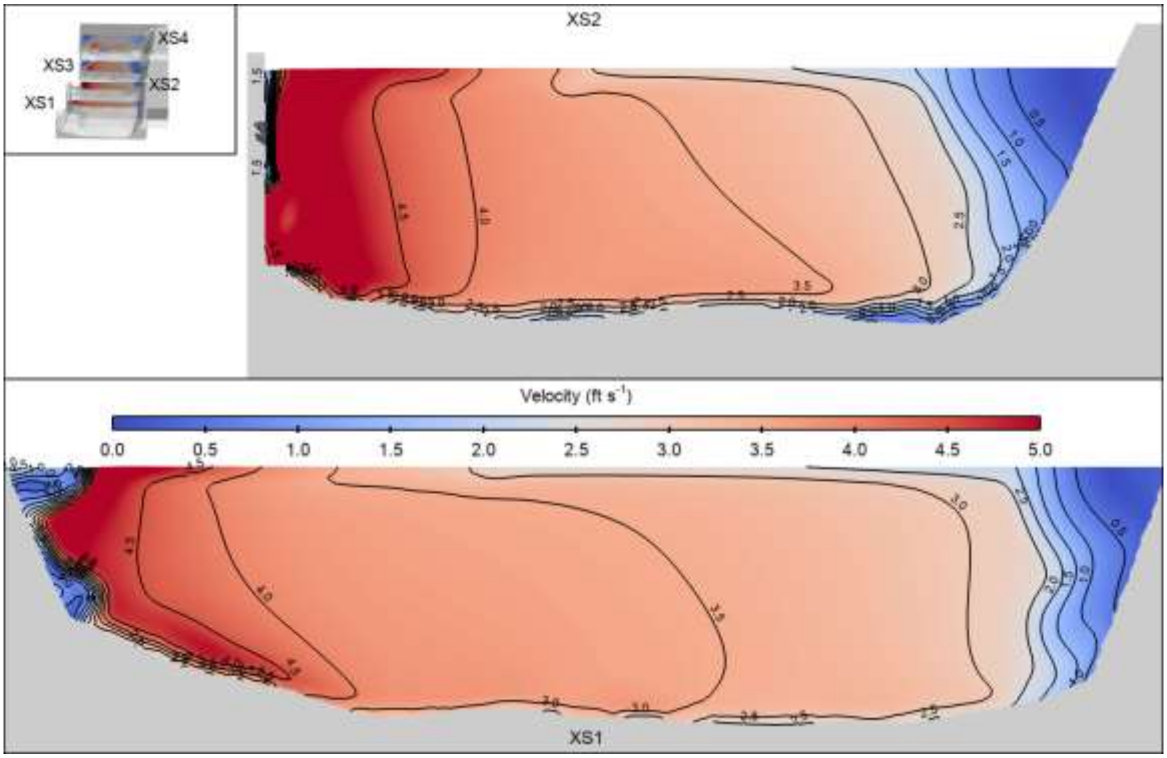

Figure 4-28: Velocity Contours on Cross-Sections for Case 1

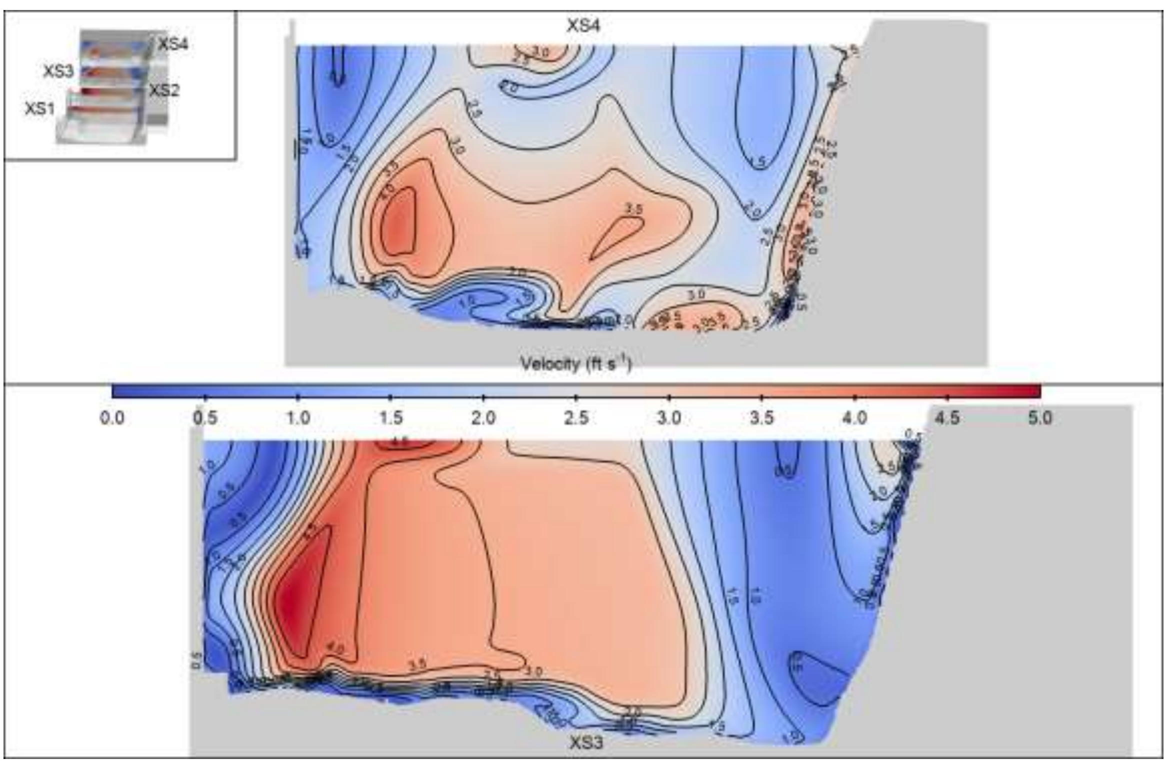

Figure 4-29: Velocity Contours on Cross-Sections for Case 1

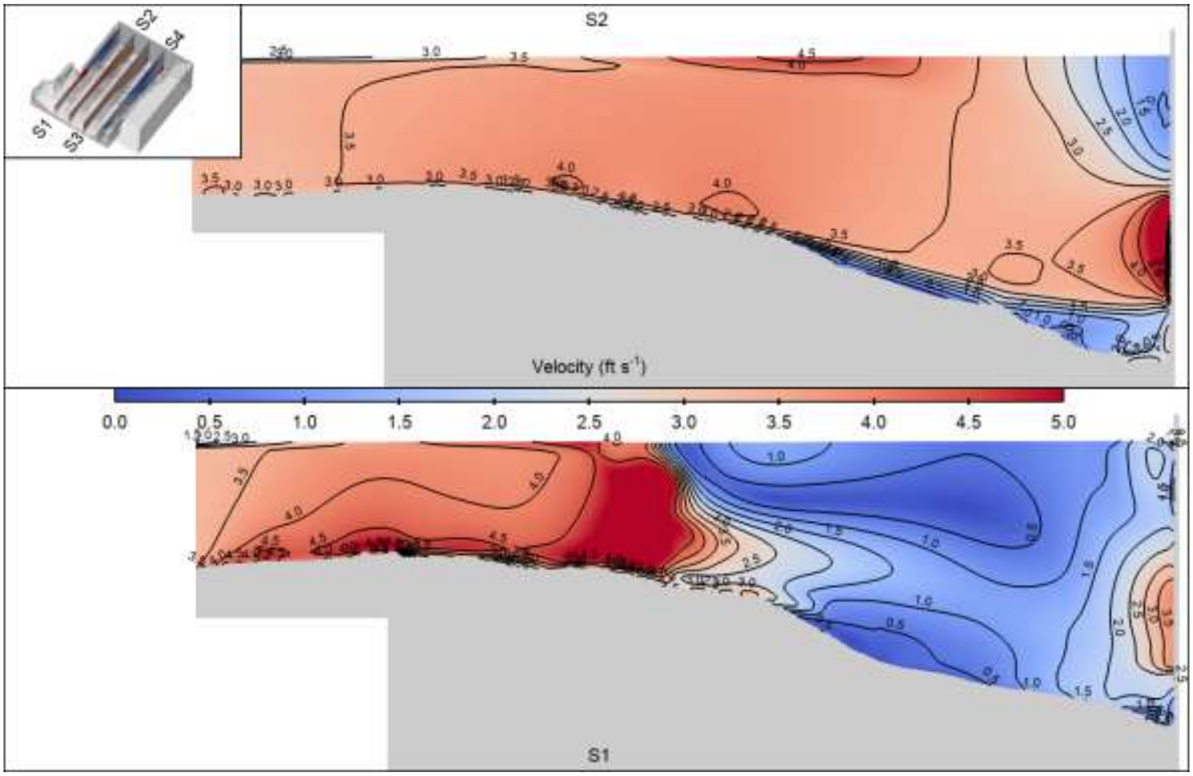

Figure 4-30: Velocity Contours on Sections for Case 1

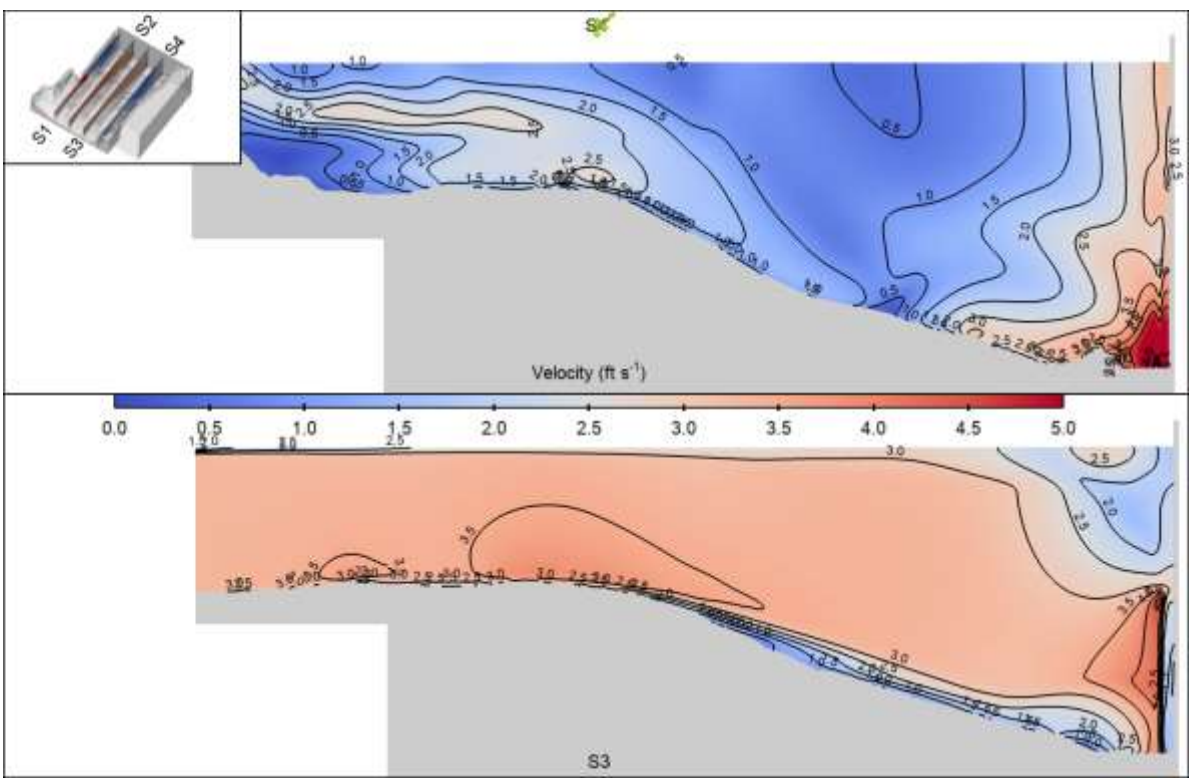

Figure 4-31: Velocity Contours on Sections for Case 1

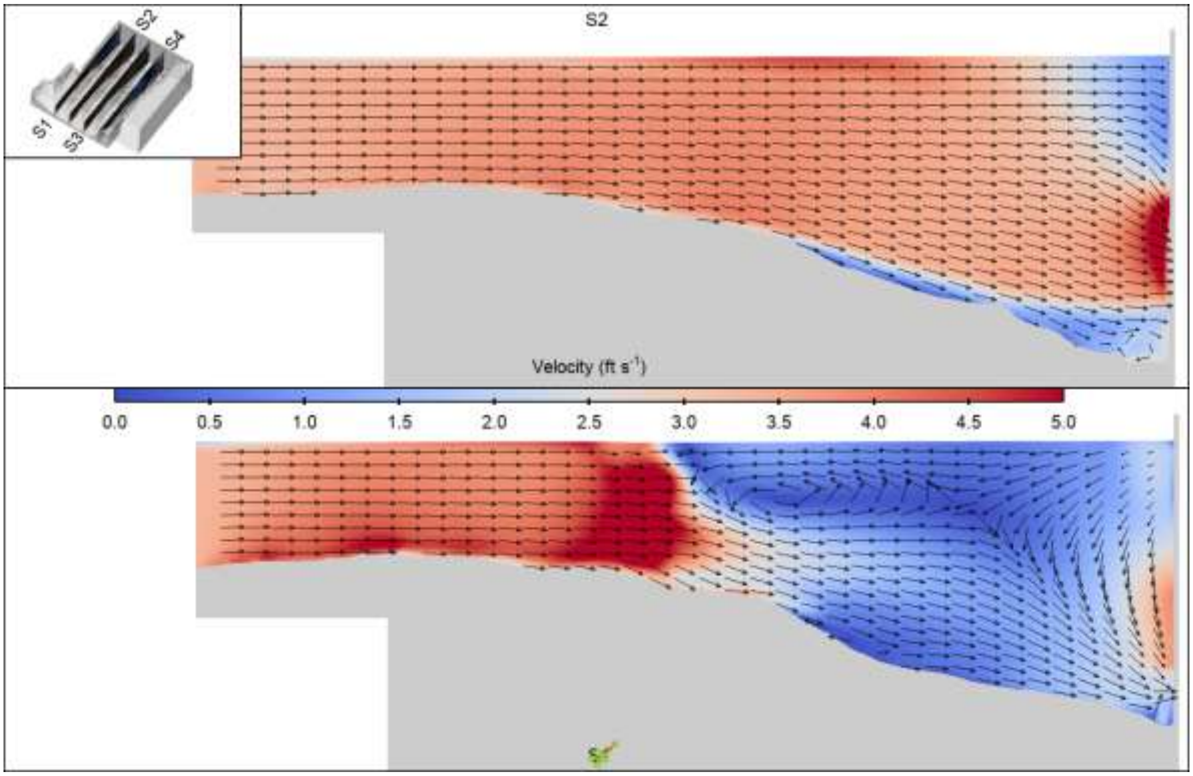

Figure 4-32: Velocity Vectors on Sections for Case 1

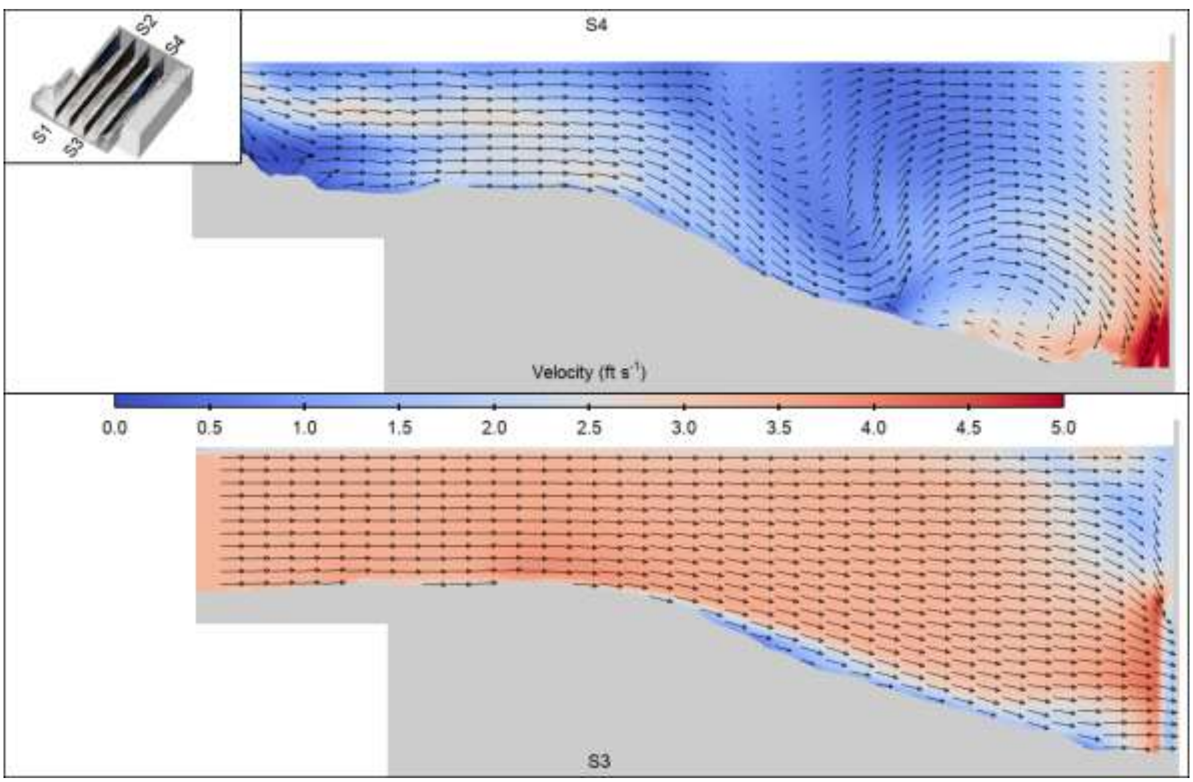

Figure 4-33: Velocity Vectors on Sections for Case 1

The velocity patterns show a large recirculation area along the fishway structure. Flow approaches the fishway entrance parallel to the powerhouse structure. The streamlines indicate flow entering the fishway originates from the left bank (viewing downstream) of the powerhouse channel.

4.2.2 Case 2 – 75% Exceedance

The 75 percent exceedance model assigns 1,310 cfs through each unit for a total powerhouse flow of 2,620 cfs. The AWS is operating at 130 cfs under this scenario. Figure 4-34 shows the velocity contours and vectors at the water surface (El. 91 ft, NGVD29) and depths of 1 ft (El. 90 ft, NGVD29), 6 ft (El. 85 ft, NGVD29), and 11 ft (El. 85 ft, NGVD29). The streamlines colored by velocity are shown on Figure 4-35. Figure 4-36 shows streamlines colored by model outlet. Figure 4-37 through Figure 4-42 show velocity contour or vectors on multiple sections in the forebay.

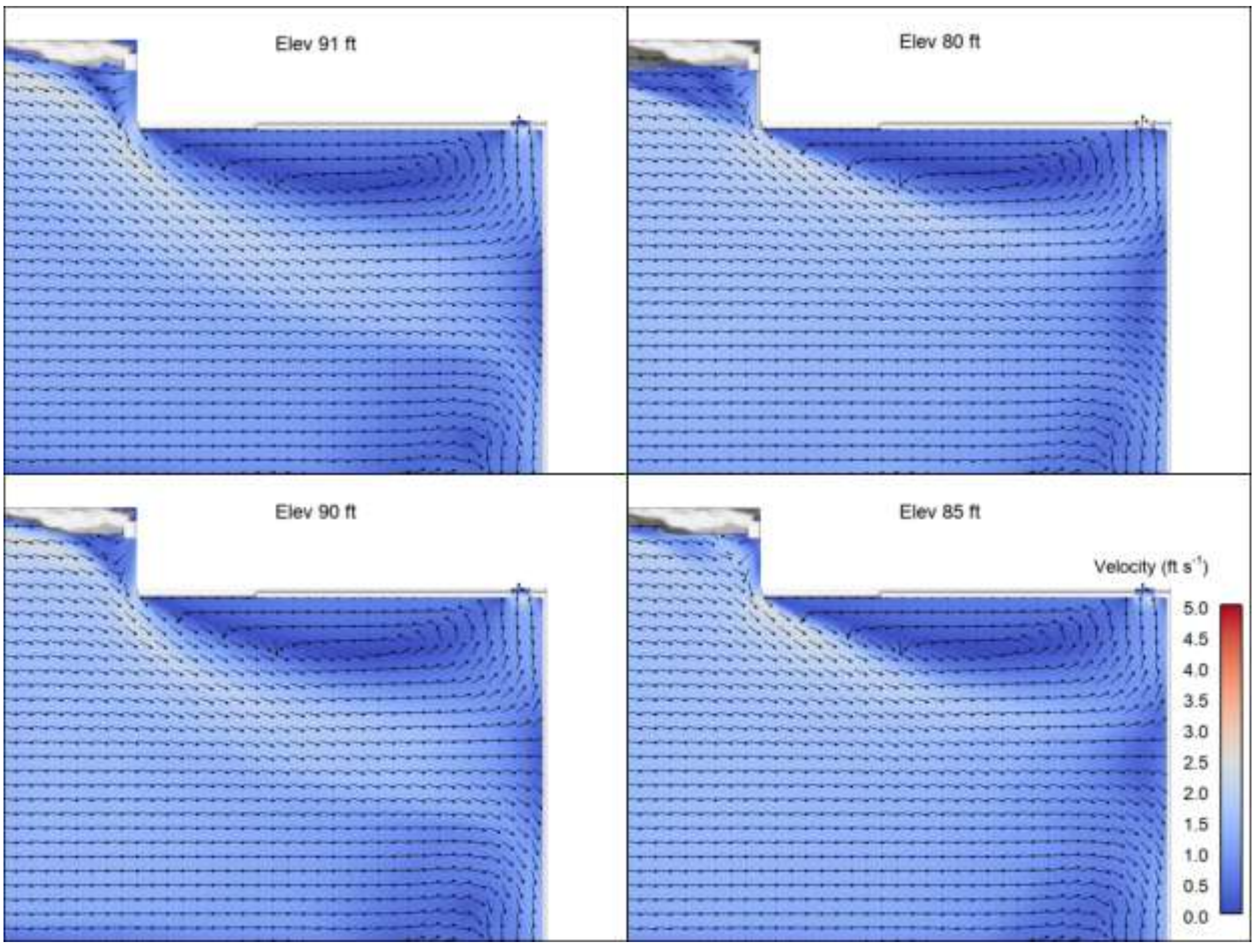

Figure 4-34: Velocity Contours and Vectors for Case 2

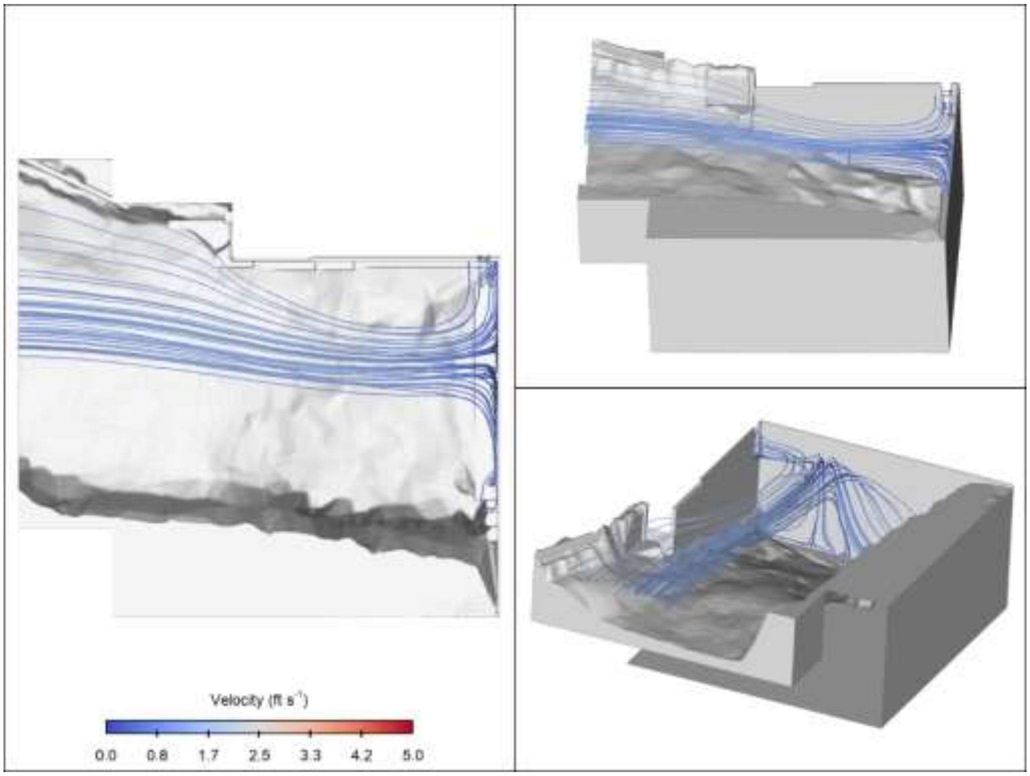

Figure 4-35: Streamlines Colored by Velocity Showing Flow Patterns for Case 2

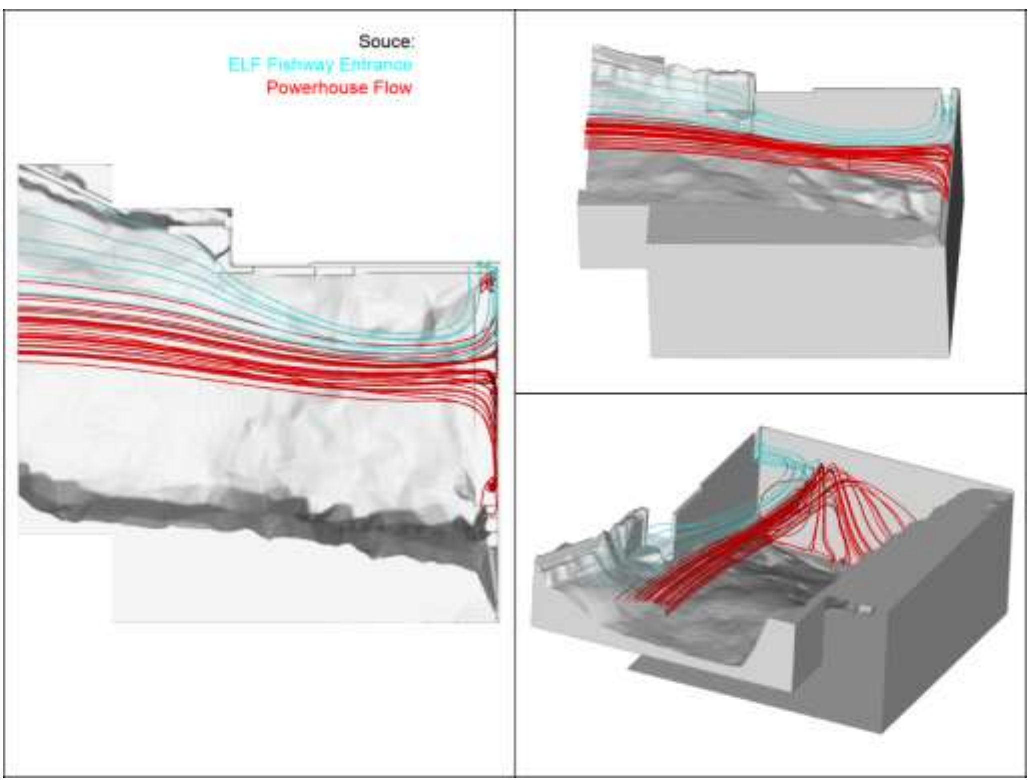

Figure 4-36: Streamlines Colored by Model Outlet Showing Flow Patterns for Case 2

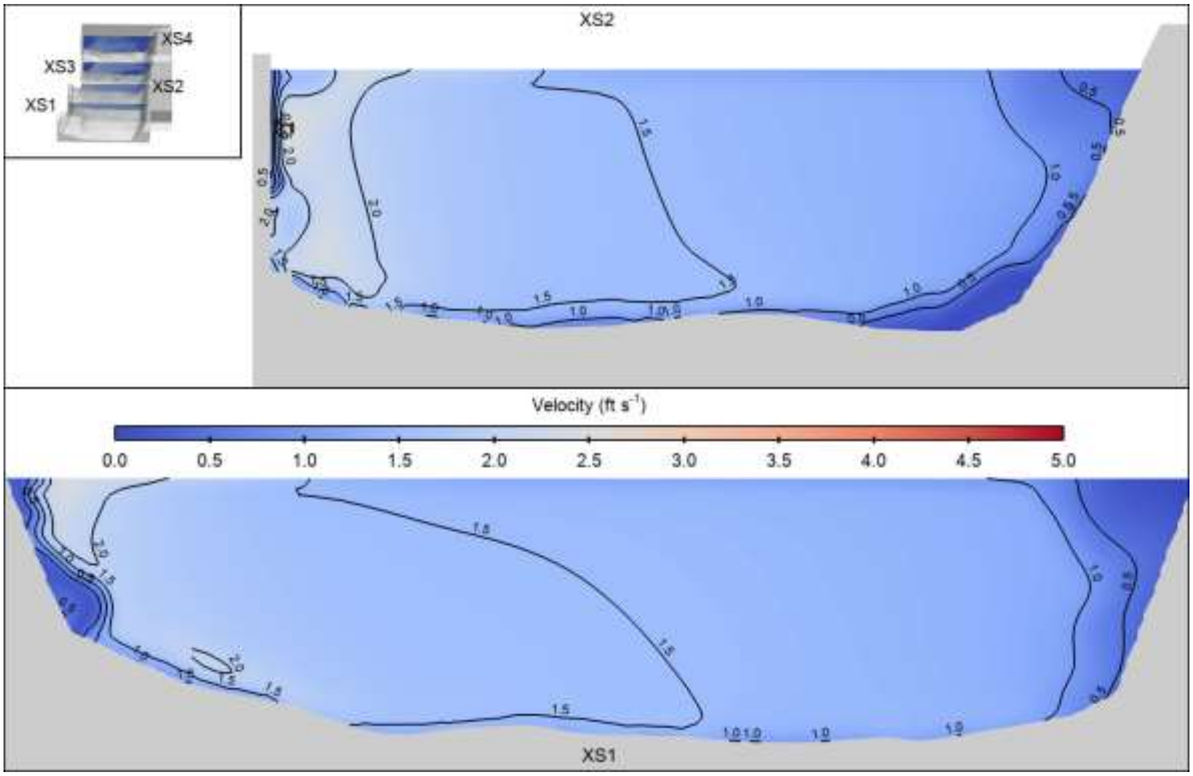

Figure 4-37: Velocity Contours on Cross-Sections for Case 2

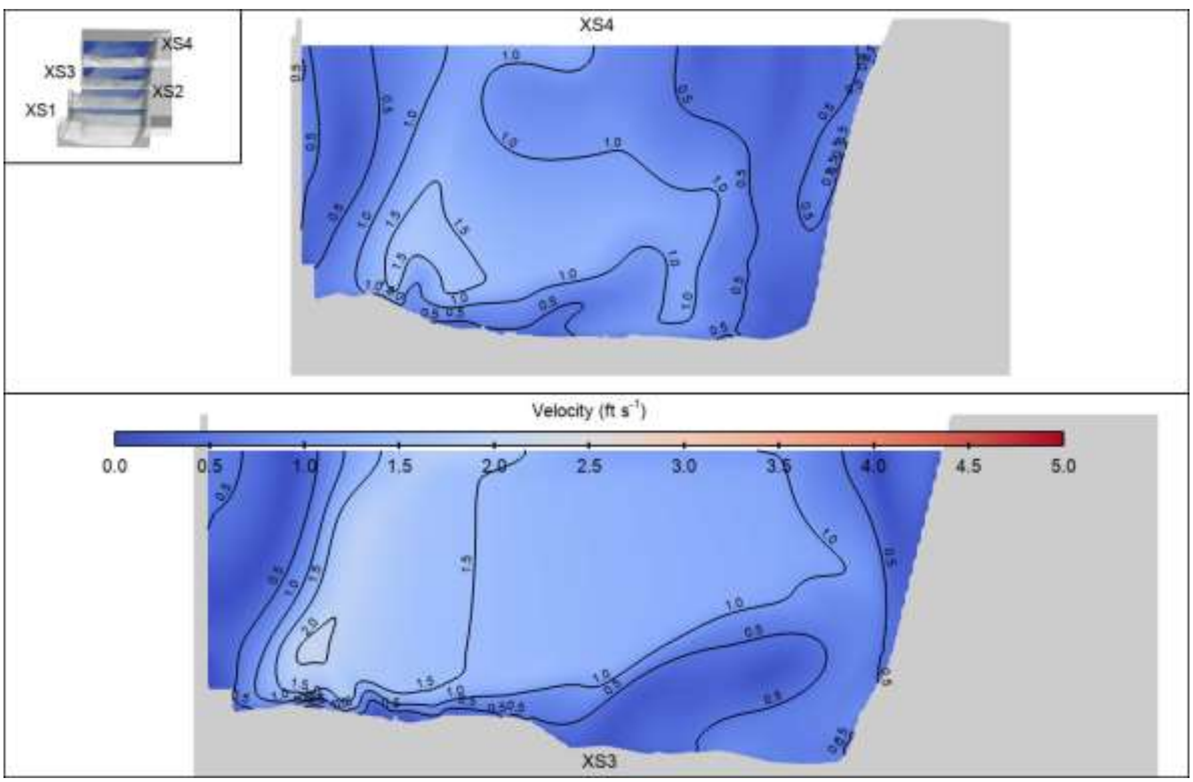

Figure 4-38: Velocity Contours on Cross-Sections for Case 2

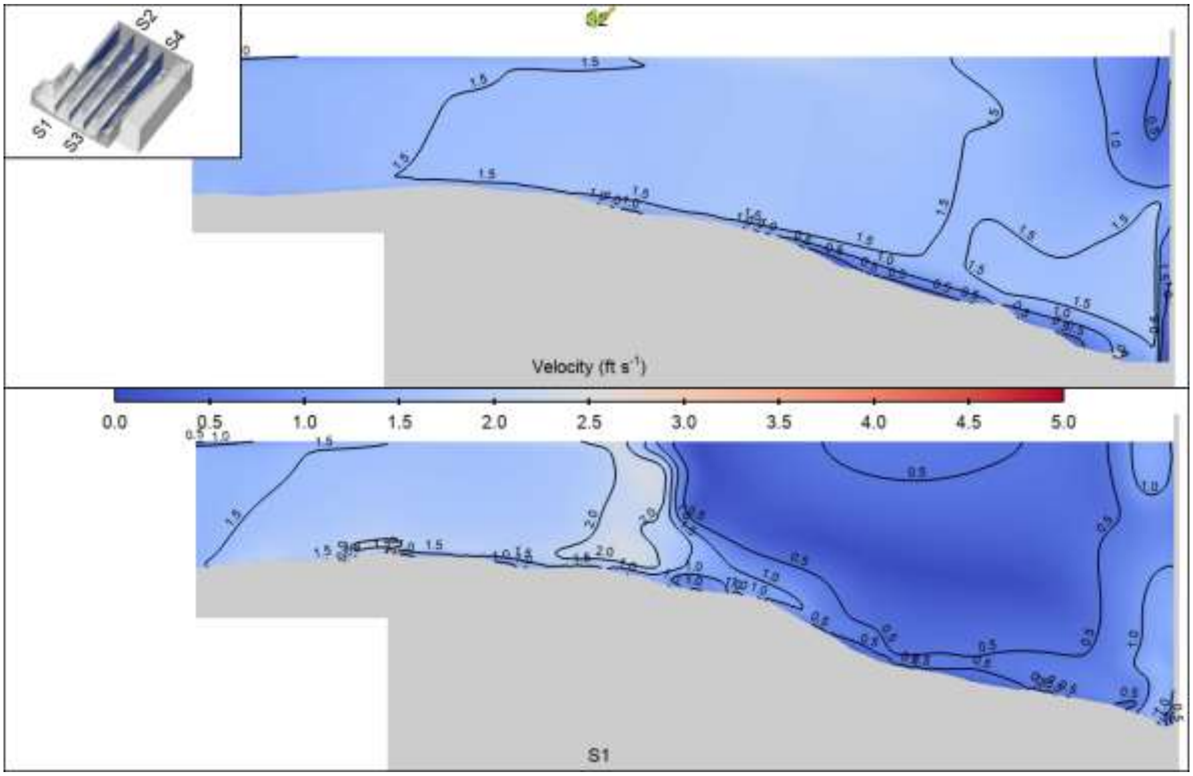

Figure 4-39: Velocity Contours on Sections for Case 2

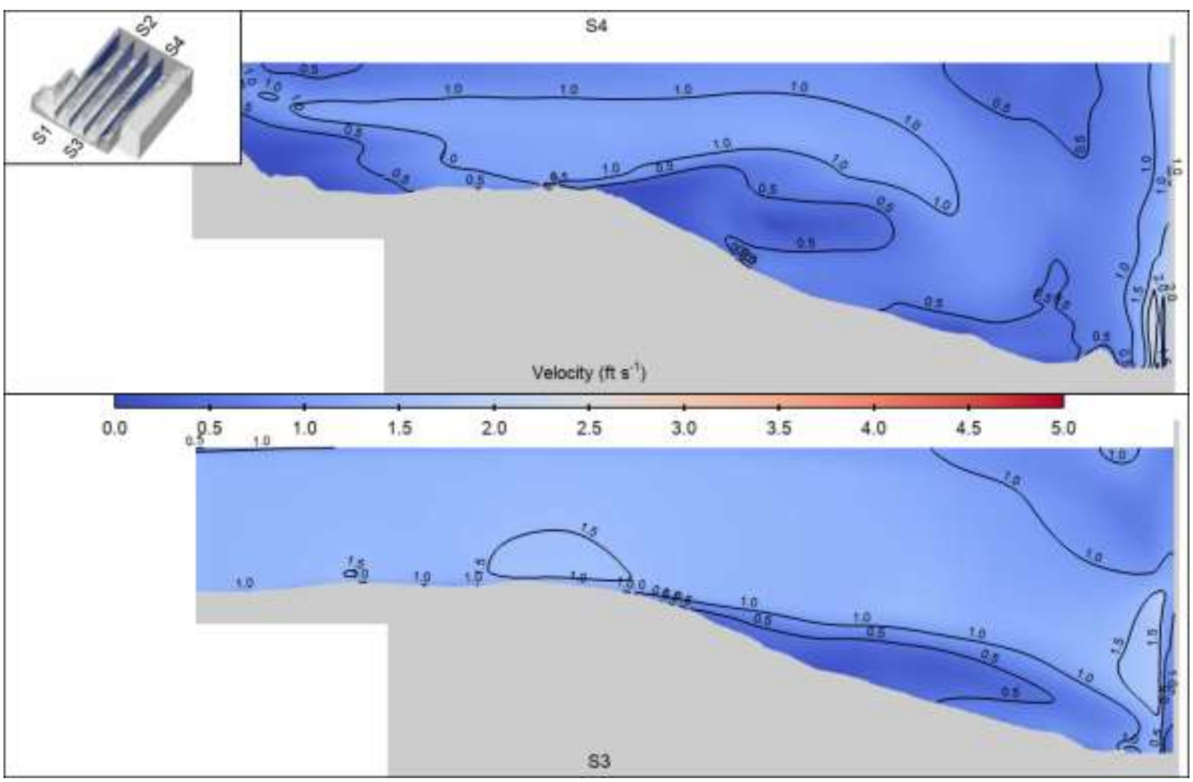

Figure 4-40: Velocity Contours on Sections for Case 2

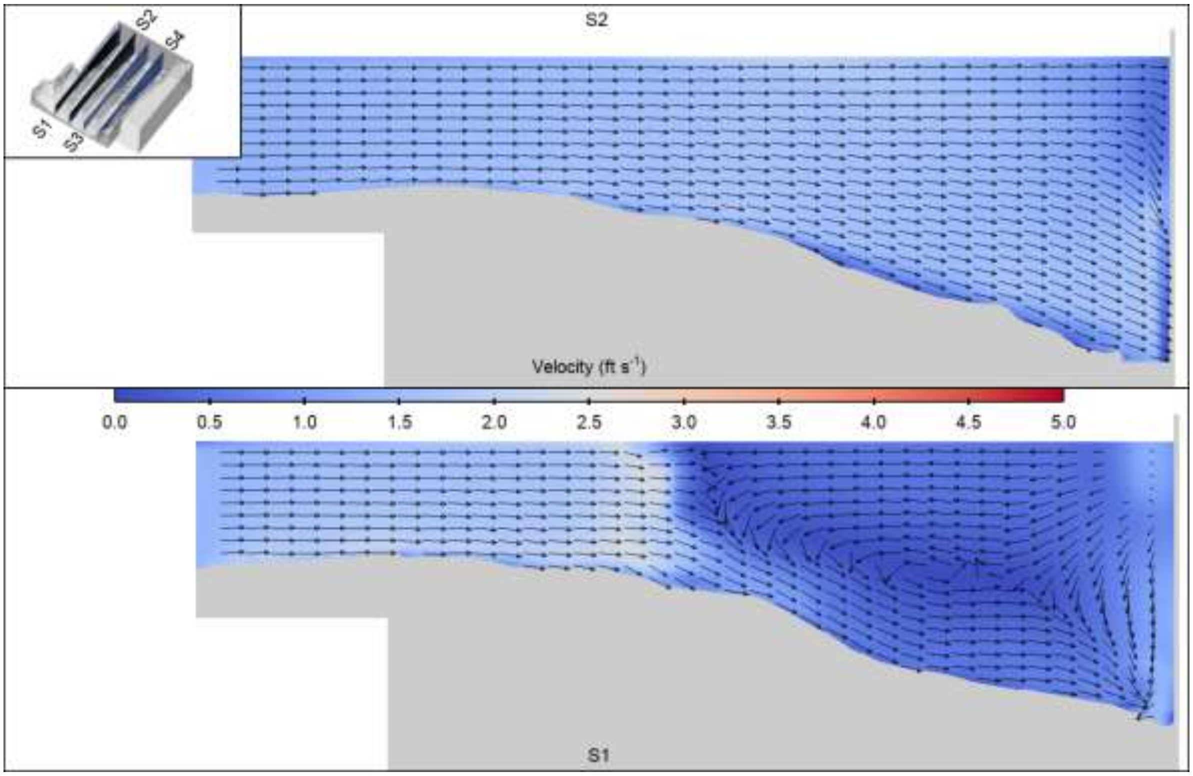

Figure 4-41: Velocity Vectors on Sections for Case 2

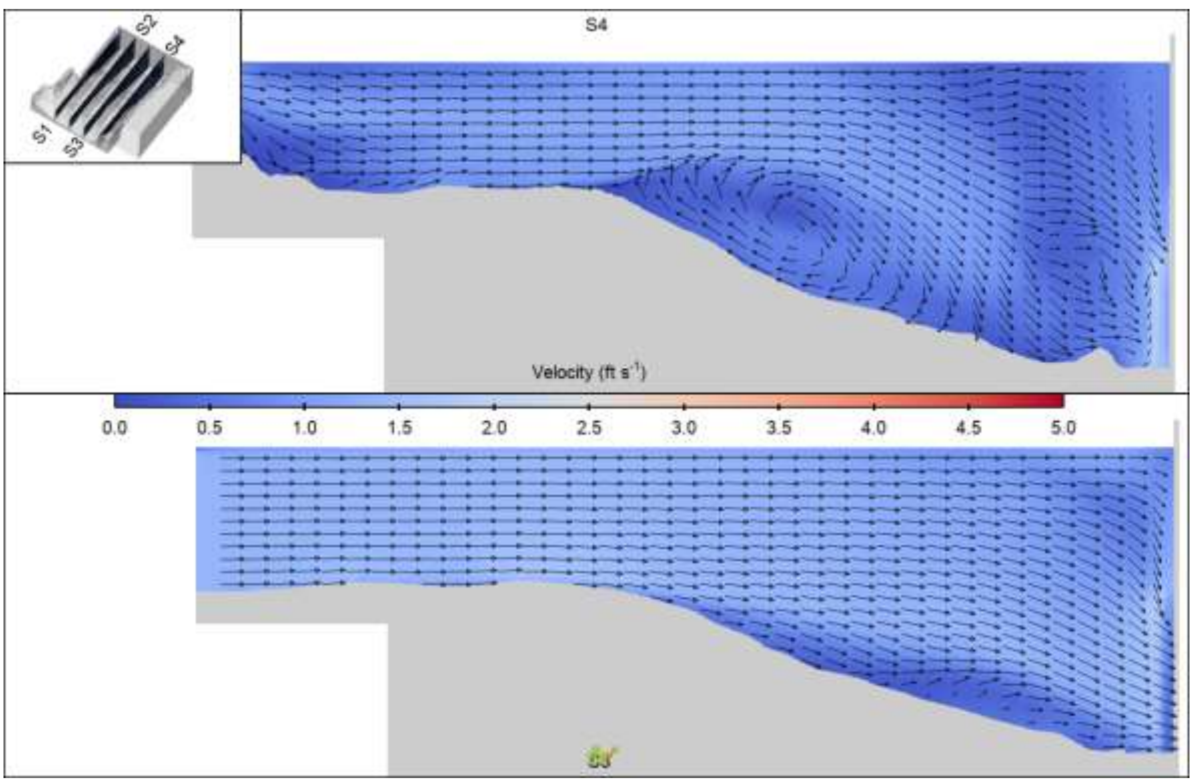

Figure 4-42: Velocity Vectors on Sections for Case 2

Similar to Case 1, The velocity patterns show a large recirculation area along the fishway structure. Flow approaches the fishway entrance parallel to the powerhouse structure. The streamlines indicate that flow entering the fishway originates from the left bank (viewing downstream) of the powerhouse channel.

4.2.3 Case 3 – Minimum Unit Flow

The 75 percent exceedance model assigns 600 cfs through each unit for a total powerhouse flow of 1,200 cfs. The AWS is operating at 130 cfs under this scenario. Figure 4-43 shows the velocity contours and vectors at the water surface (El. 91 ft, NGVD29) and depths of 1 ft (El. 90 ft, NGVD29), 6 ft (El. 85 ft, NGVD29), and 11 ft (El. 85 ft, NGVD29). The streamlines colored by velocity are shown on Figure 4-44. Figure 4-45 shows streamlines colored by model outlet. Figure 4-46 through Figure 4-51 show velocity contour or vectors on multiple sections in the forebay.

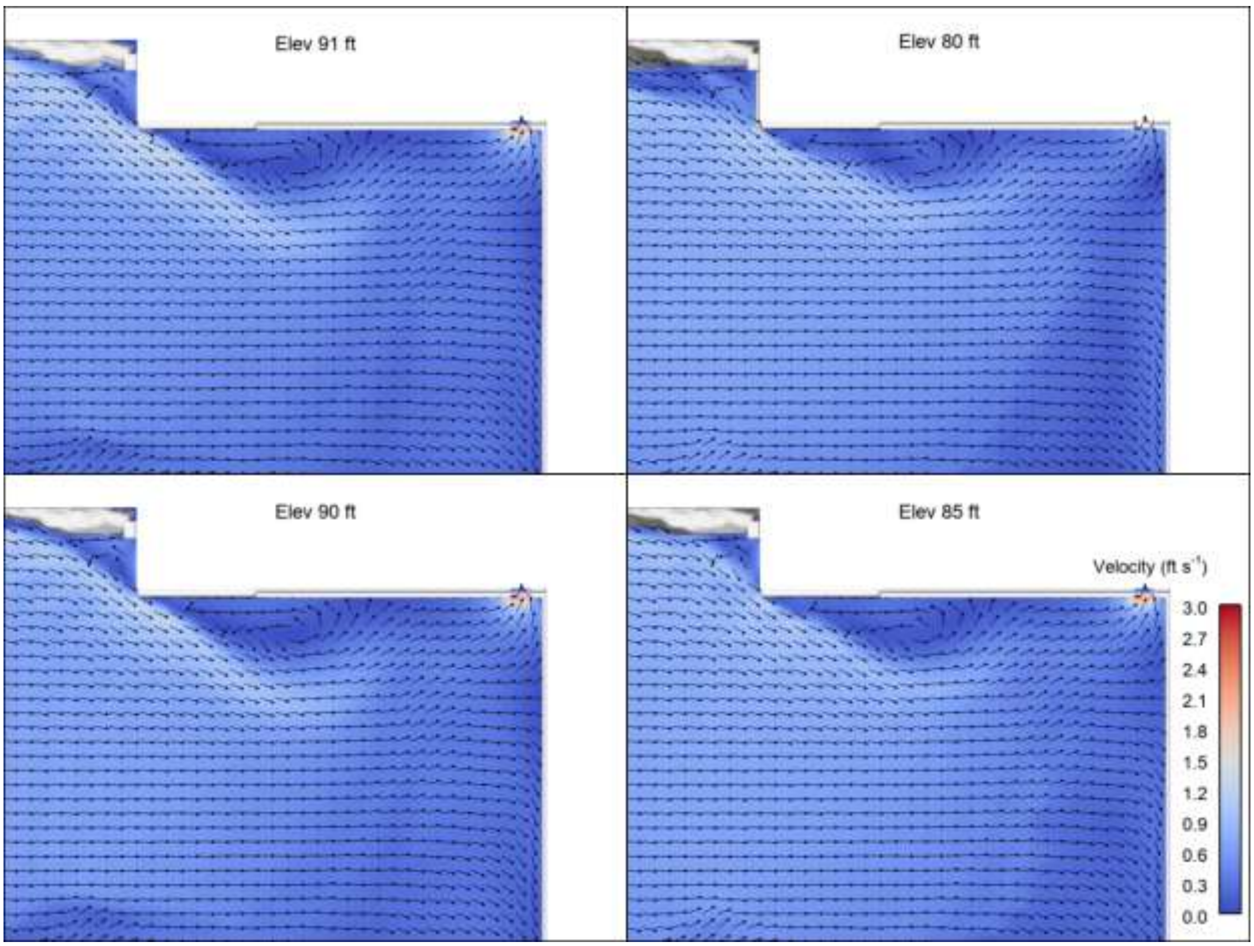

Figure 4-43: Velocity Contours and Vectors for Case 3

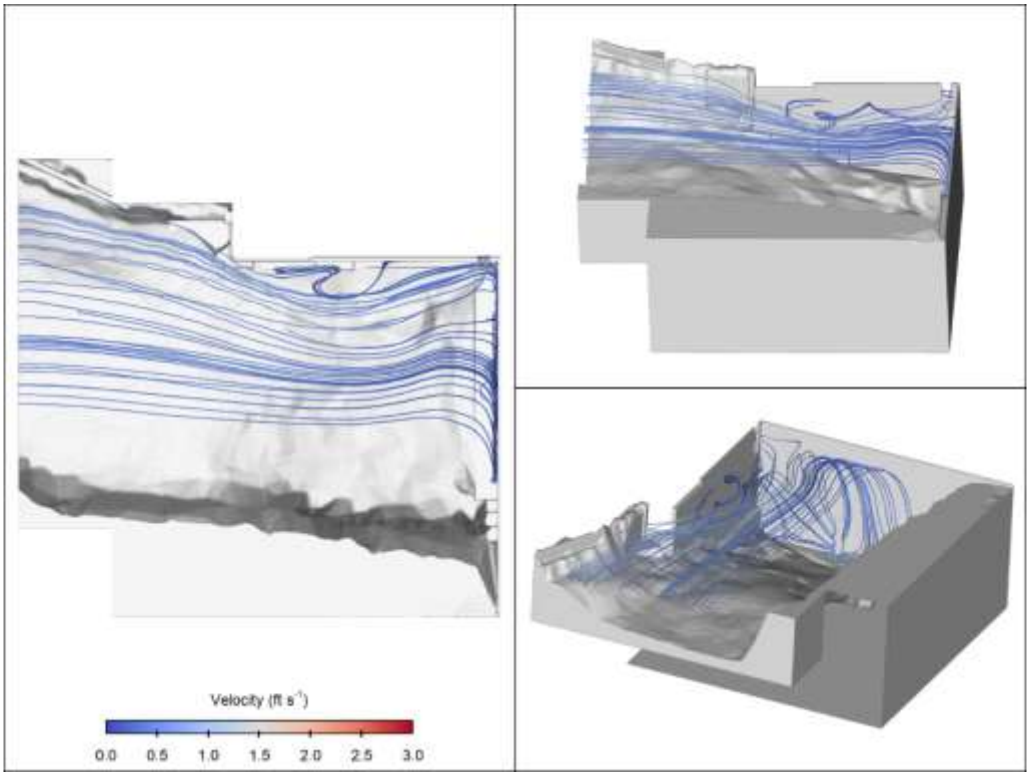

Figure 4-44: Streamlines Colored by Velocity Showing Flow Patterns for Case 3

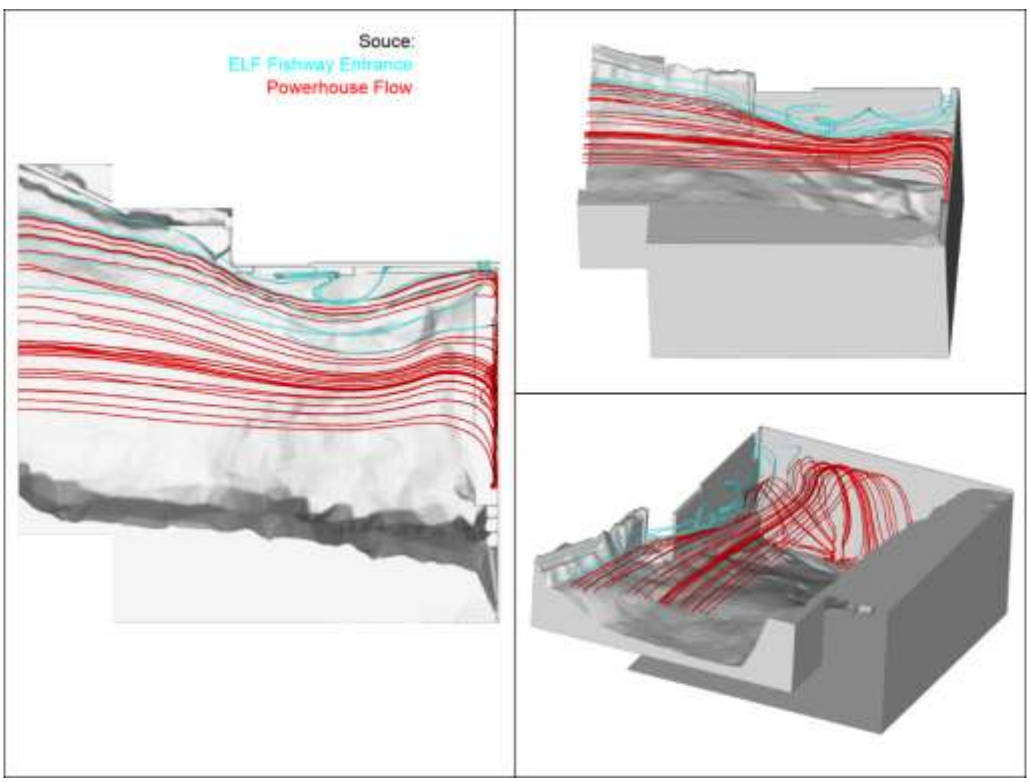

Figure 4-45: Streamlines Colored by Model Outlet Showing Flow Patterns for Case 3

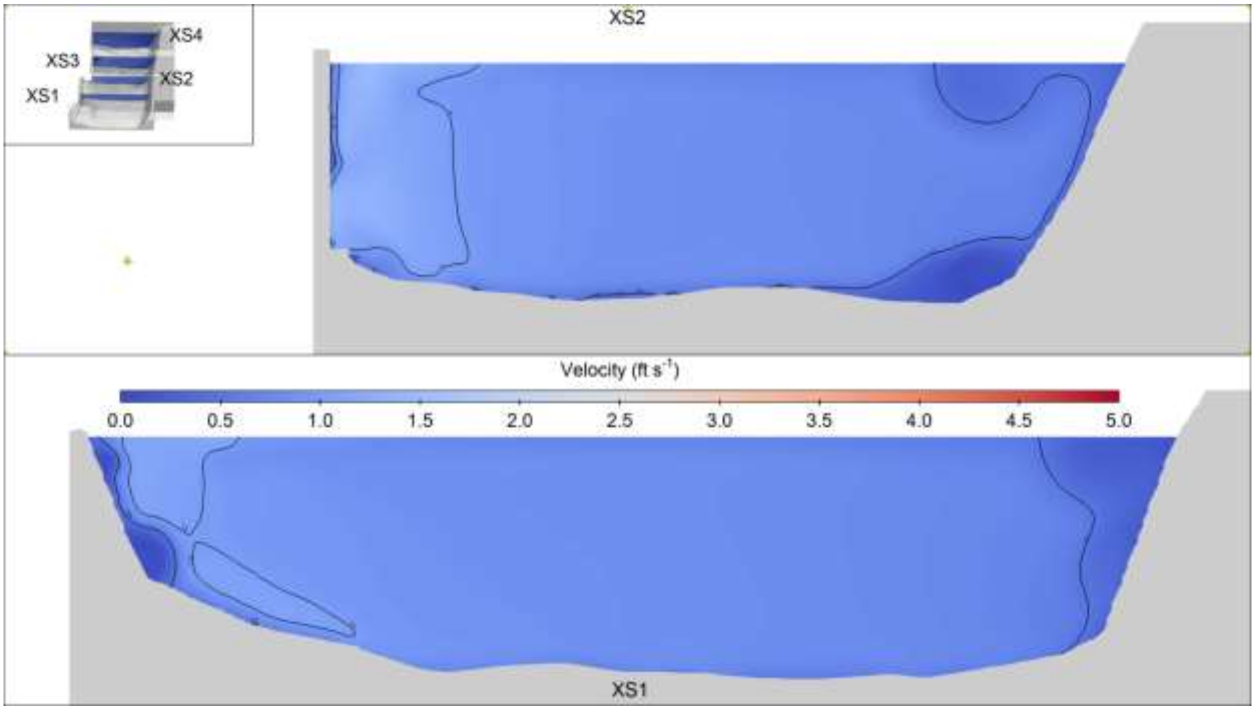

Figure 4-46: Velocity Contours on Cross-Sections for Case 3

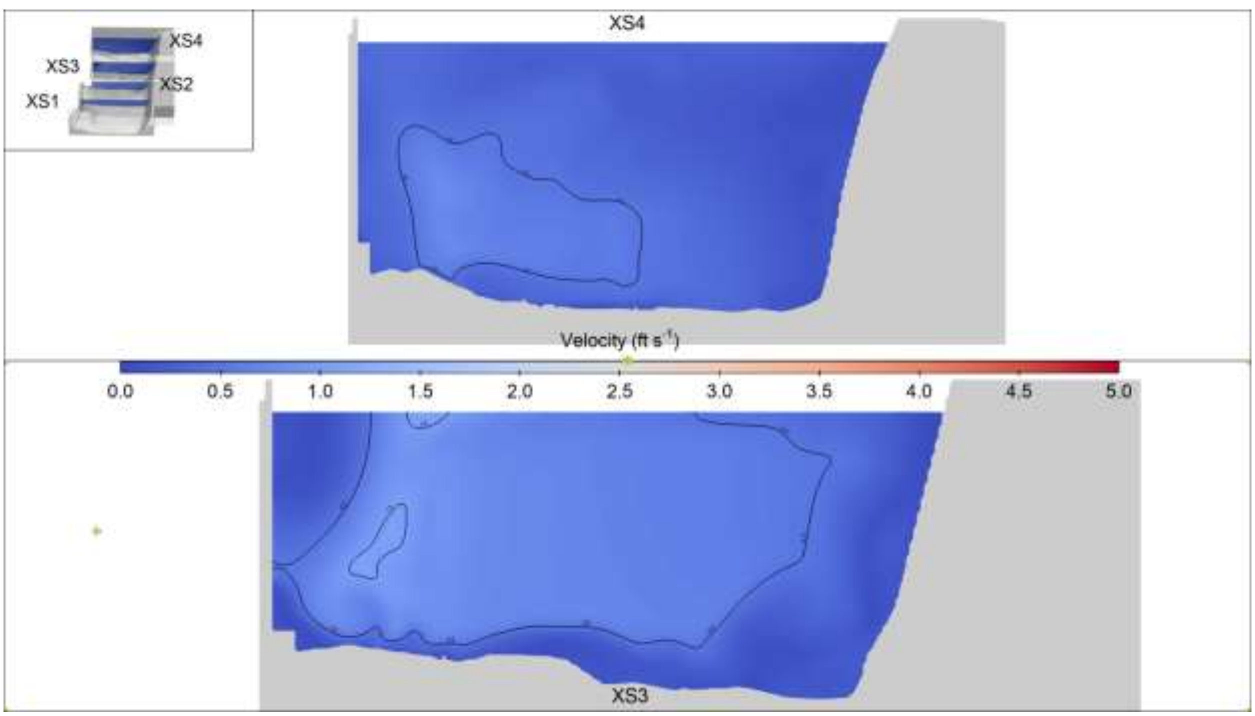

Figure 4-47: Velocity Contours on Cross-Sections for Case 3

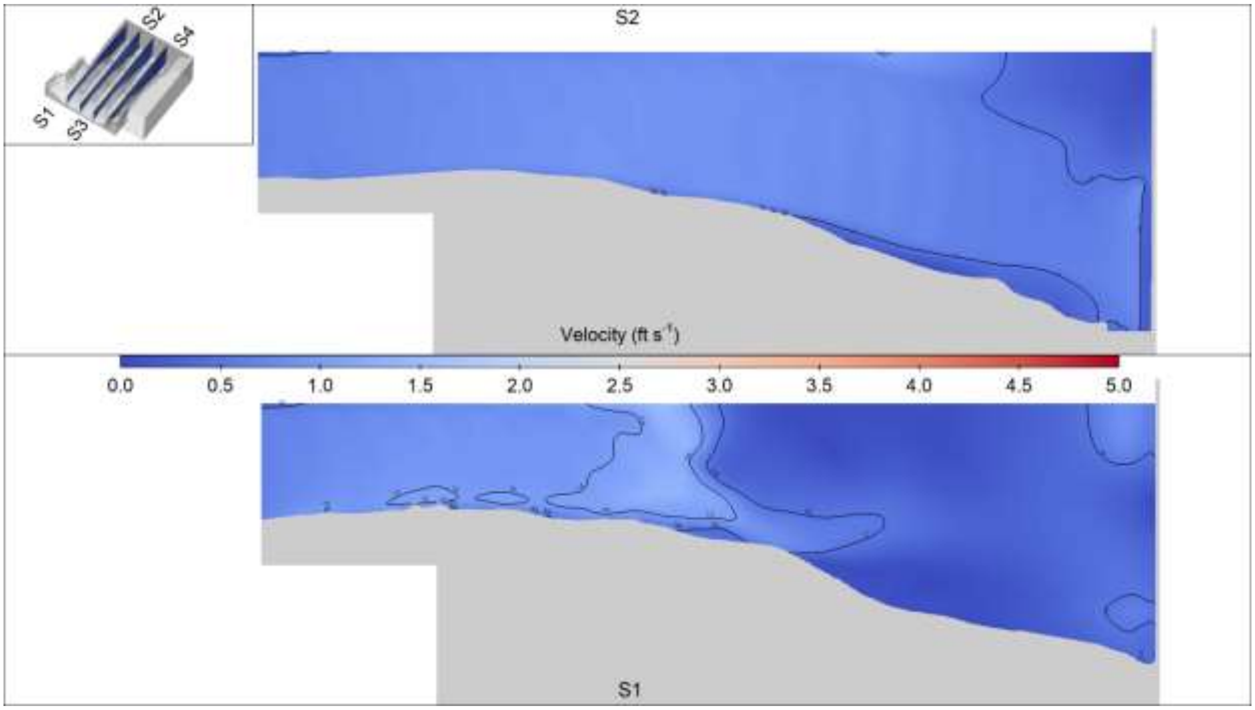

Figure 4-48: Velocity Contours on Sections for Case 3

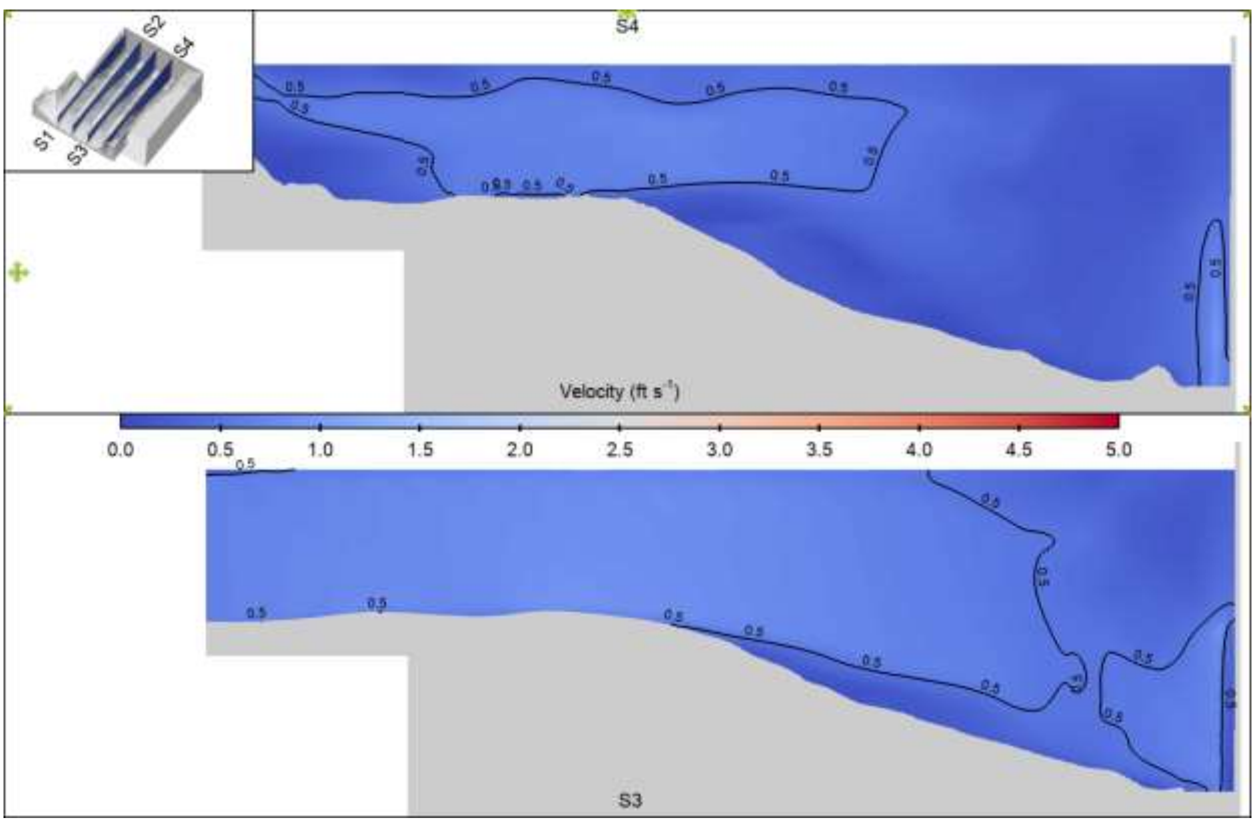

Figure 4-49: Velocity Contours on Sections for Case 3

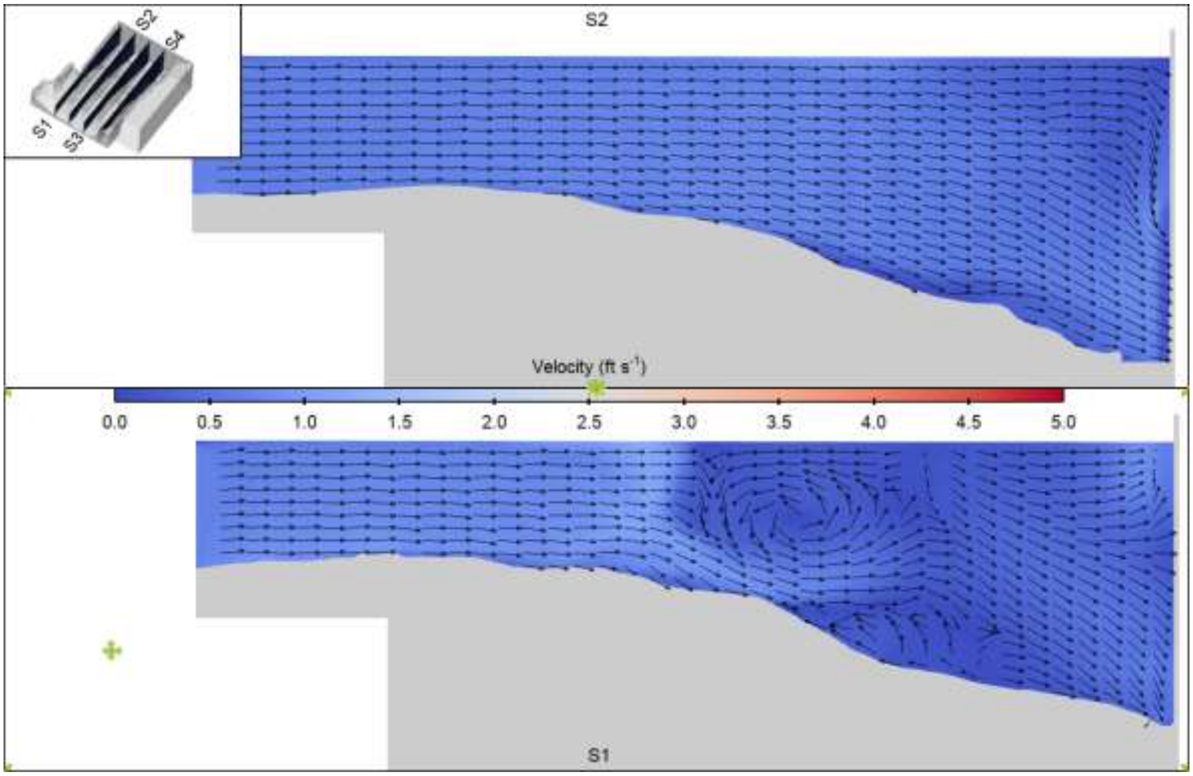

Figure 4-50: Velocity Vectors on Sections for Case 3

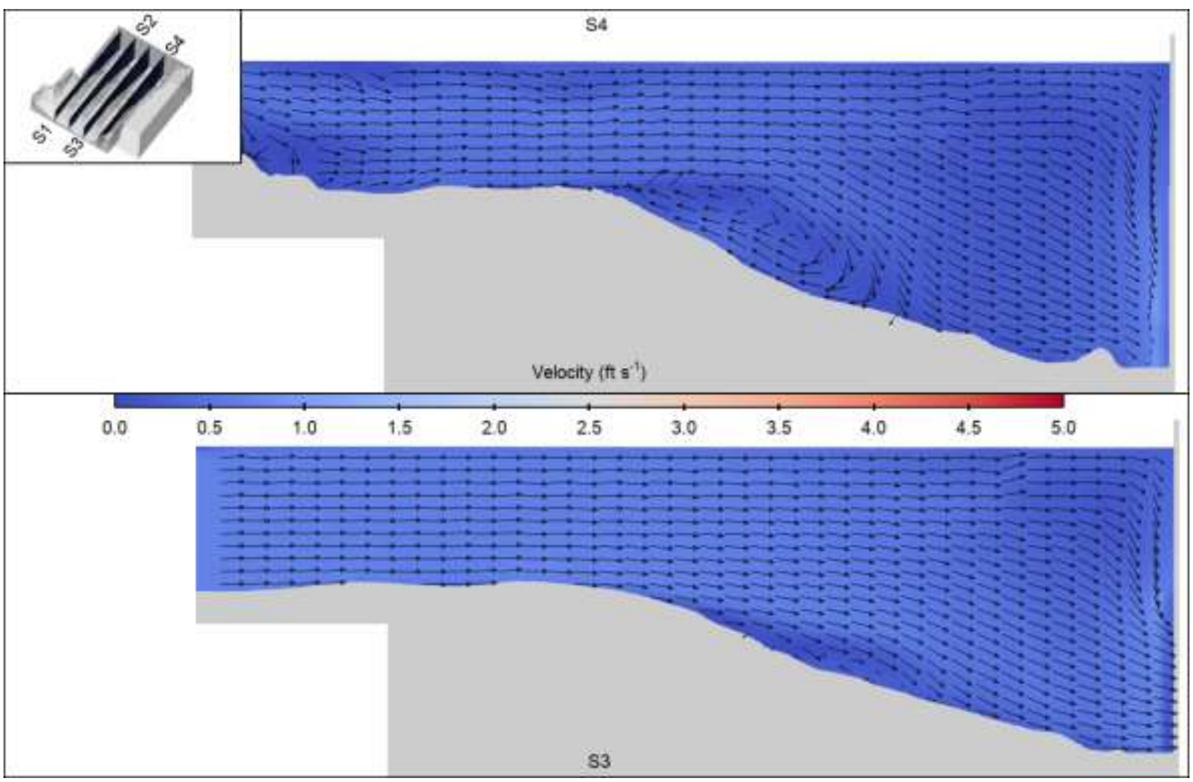

Figure 4-51: Velocity Vectors on Sections for Case 3

The velocity patterns show a recirculation area along the fish elevator structure. Flow approaches the fishway entrance parallel to the powerhouse structure. The streamlines indicate flow entering the fishway originates from the left bank (viewing downstream) of the powerhouse channel.

4.3 E.L. Field Powerhouse Tailrace Model

Two simulations for the tailrace model were analyzed and plots showing velocity contour and vectors at multiple depths were exported. The figures depicting streamlines are colored by velocity and model inlet. The red colored streamlines enter the model domain through the AWS and the cyan lines would discharge through the powerhouse.

4.3.1 Case 1 – 5% Exceedance Tailwater Level

The 5 percent exceedance model assigns 3,300 cfs through each unit for a total powerhouse flow of 6,600 cfs. The AWS is operating at 100 cfs under this scenario. The downstream tailwater condition was determined from the 1983 Tailwater Curve provided to HDR via Email on 2/12/2021. The river flow for the 5% exceedance is approximately 26,000 cfs. At this flow, the tailwater at the downstream end of the tailrace is approximately 58.36 ft. Figure 4-52 shows the velocity contours and vectors at the water surface and depths of 2 ft, 5 ft, and 10 ft. Multiple profiles showing velocity contours and vectors are shown on Figure 4-53. The streamlines colored by velocity are shown on Figure 4-54. Figure 4-55 shows streamlines colored by model outlet. The blue colored streamlines enter the model domain through the AWS and the red lines would discharge through the powerhouse.

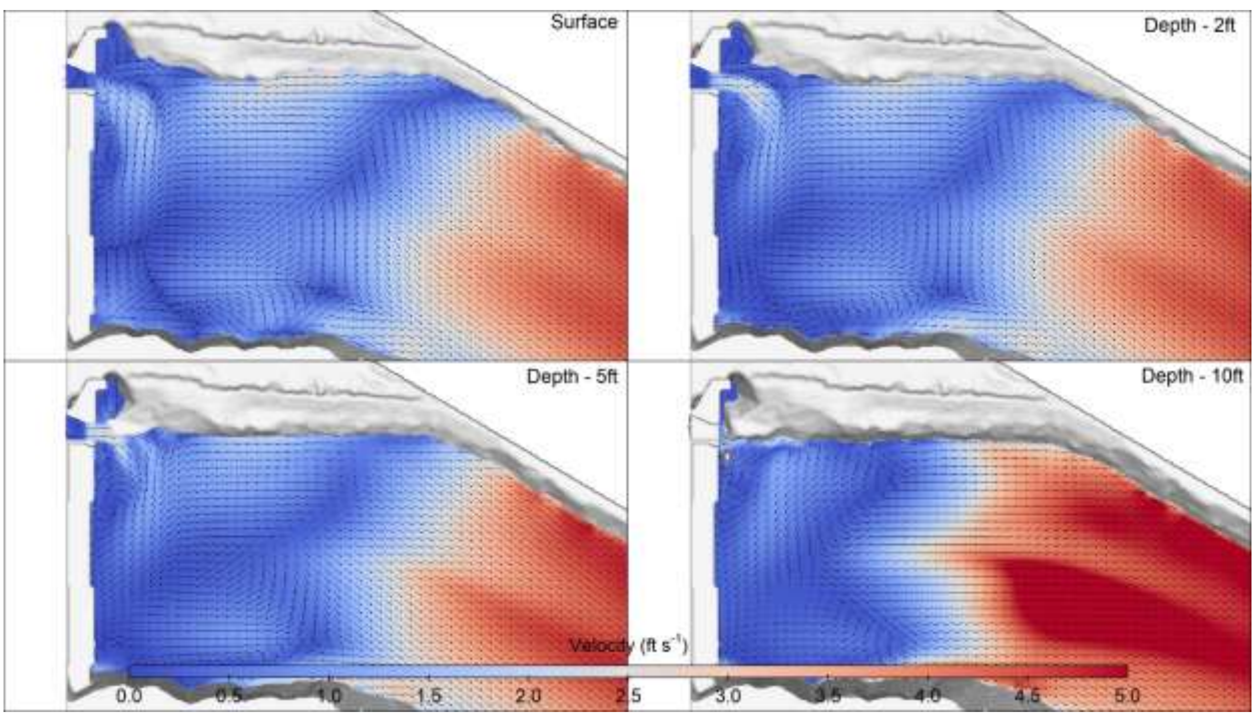

Figure 4-52: Velocity Contours and Vectors at Depth for Case 1

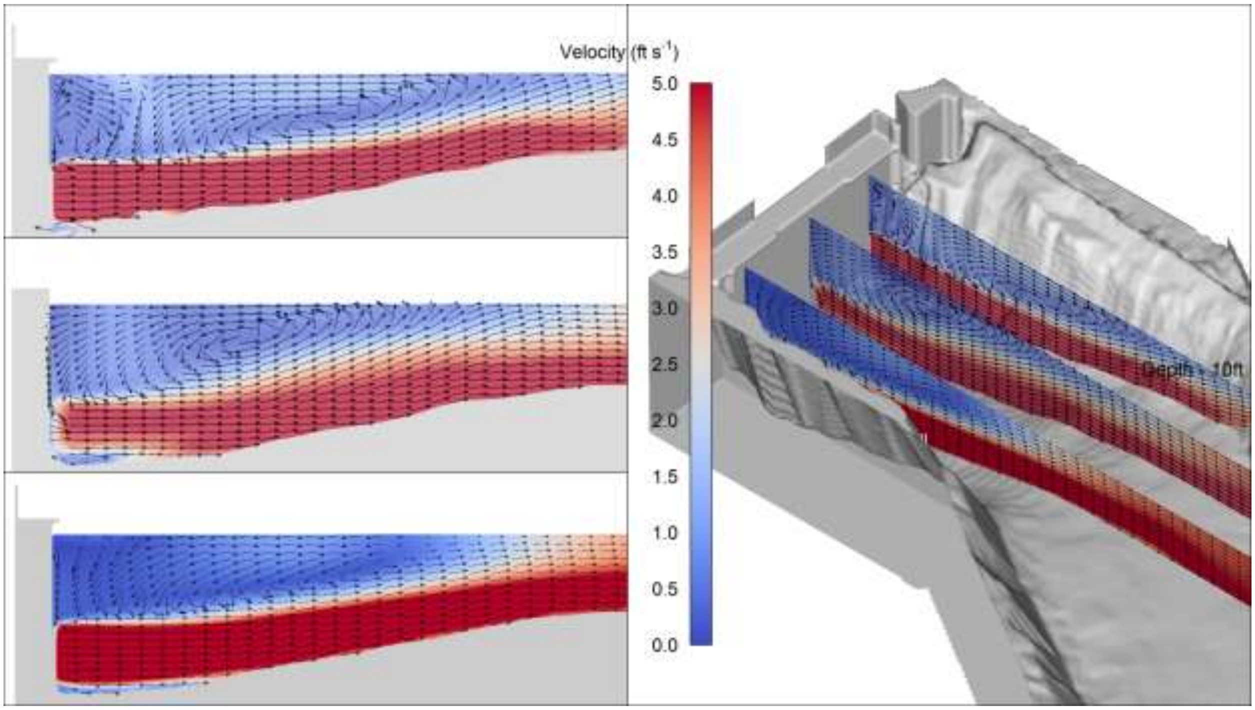

Figure 4-53: Velocity contours and Vector Profiles for Case 1

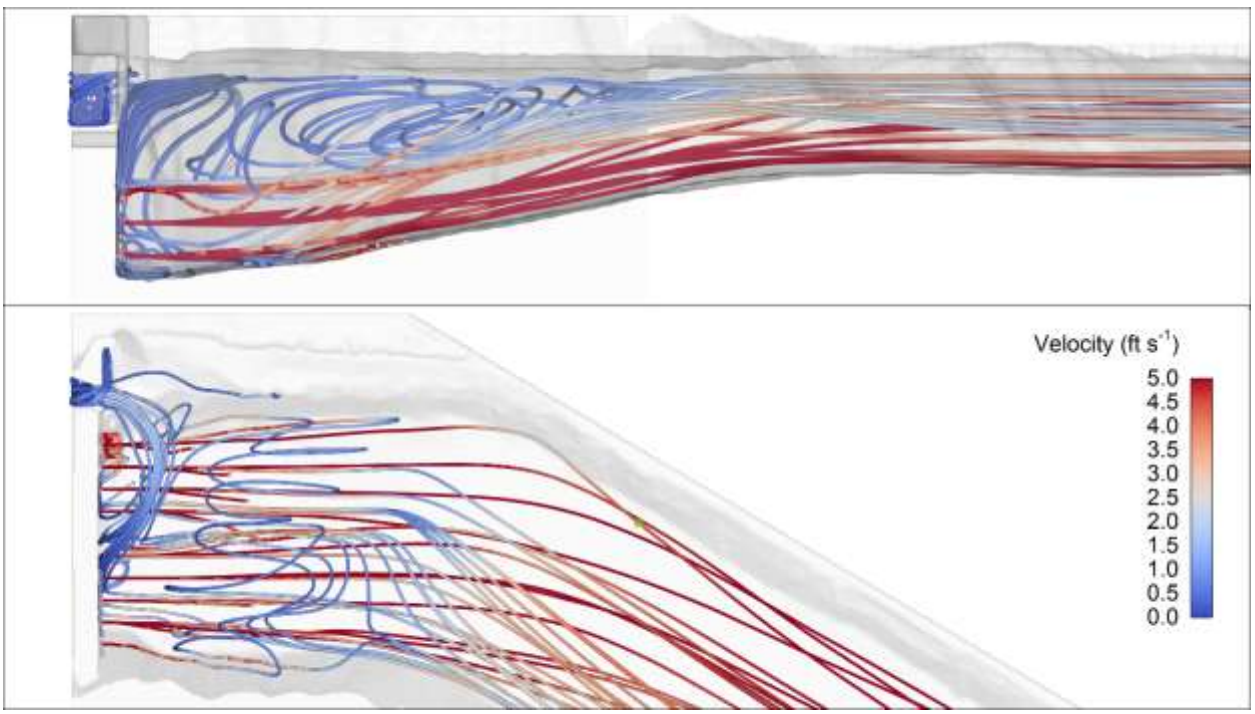

Figure 4-54: Streamlines Colored by Velocity Showing Flow Patterns for Case 1

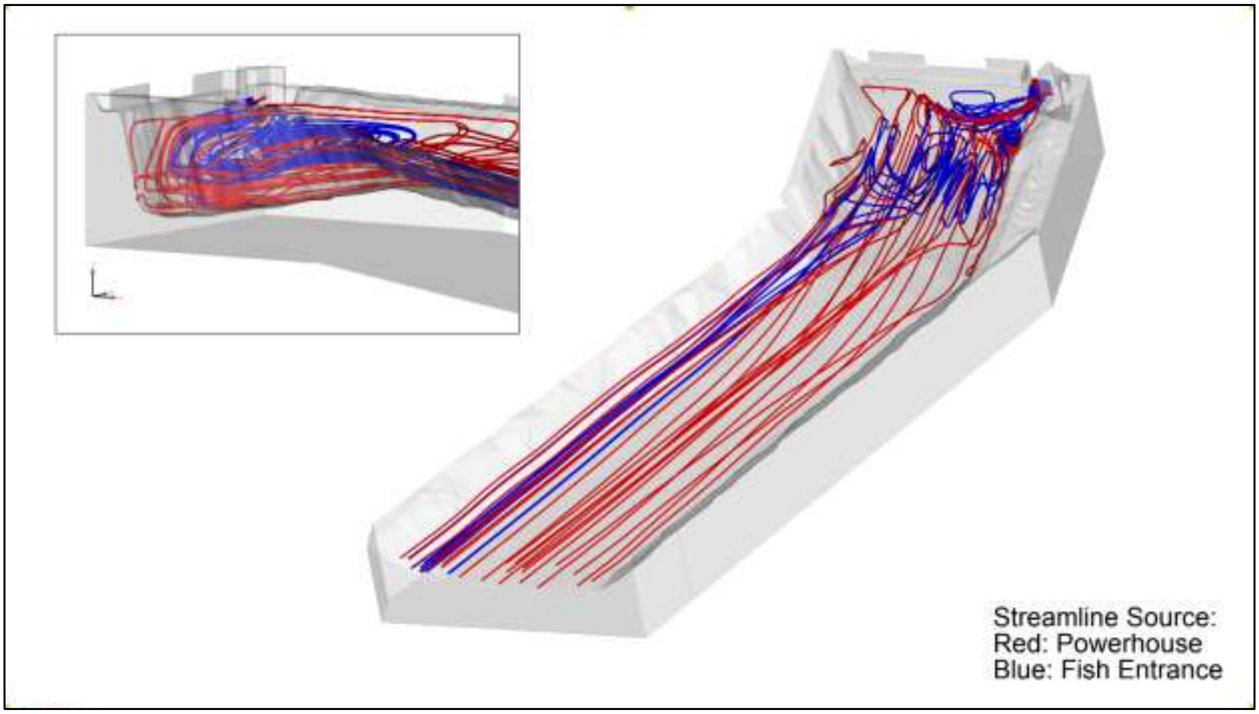

Figure 4-55: Streamlines Colored by Model Outlet Showing Flow Patterns for Case 1

The velocity patterns show a recirculation area above the powerhouse discharges. The recirculation creates two areas where flow is separated near the powerhouse discharge. The flow from the AWS mixes with the powerhouse discharge. The entrance to the AWS shows velocities below 4.0 ft per second. The rock outcrop downstream of the outlet directs flow to the center of the tailrace allowing the flow to mix with the powerhouse discharge.

4.3.2 Case 2 – 50% Exceedance Tailwater Level

The 50 percent exceedance model assigns 3,300 cfs through each unit for a total powerhouse flow of 6,600 cfs. The downstream tailwater condition was determined from the 1983 Tailwater Curve provided to HDR via Email on 2/12/2021. The river flow for the 50% exceedance is approximately 6,755 cfs. At this flow, the tailwater at the downstream end of the tailrace is approximately 53.2 ft. The AWS is operating at 100 cfs under this scenario. Figure 4-56 shows the velocity contours and vectors at the water surface and depths of 2 ft, 5 ft, and 10 ft. Multiple profiles showing velocity contours and vectors are shown in Figure 4-57. The streamlines colored by velocity are shown in Figure 4-58. Figure 4-59 shows streamlines colored by model outlet. The blue colored streamlines enter the model domain through the AWS and the red lines would discharge through the powerhouse.

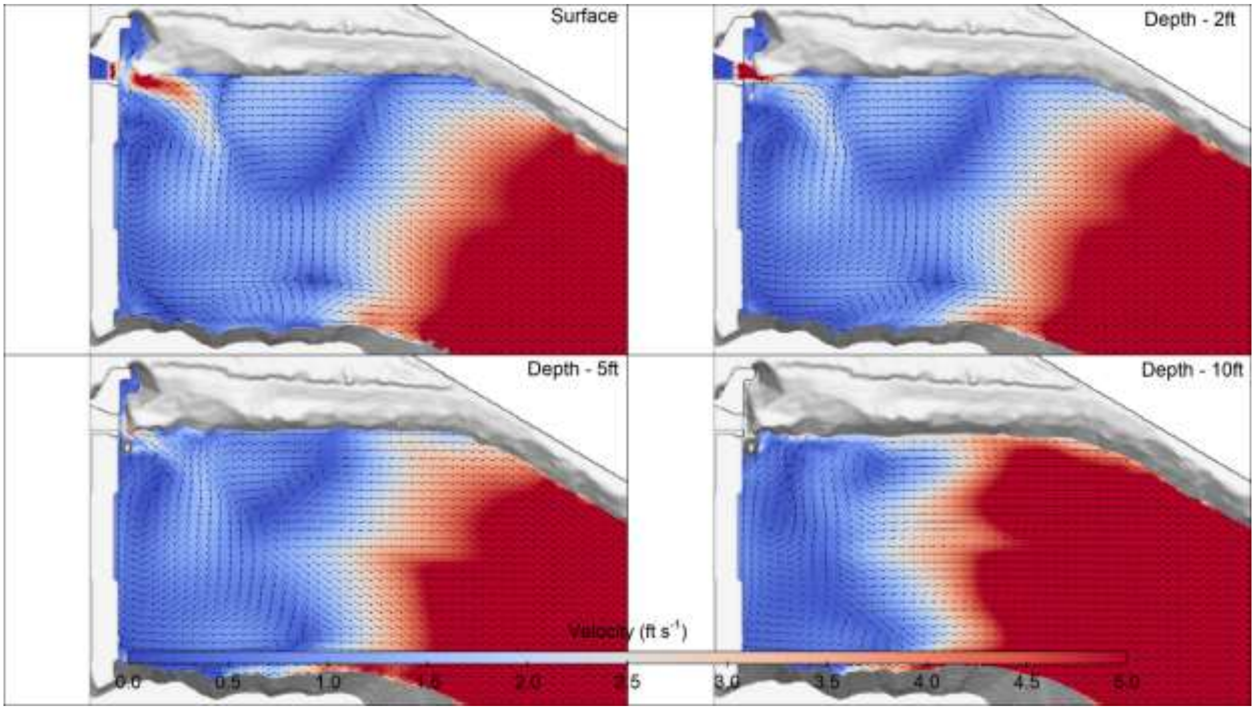

Figure 4-56: Velocity Contours and Vectors at Depth for Case 2

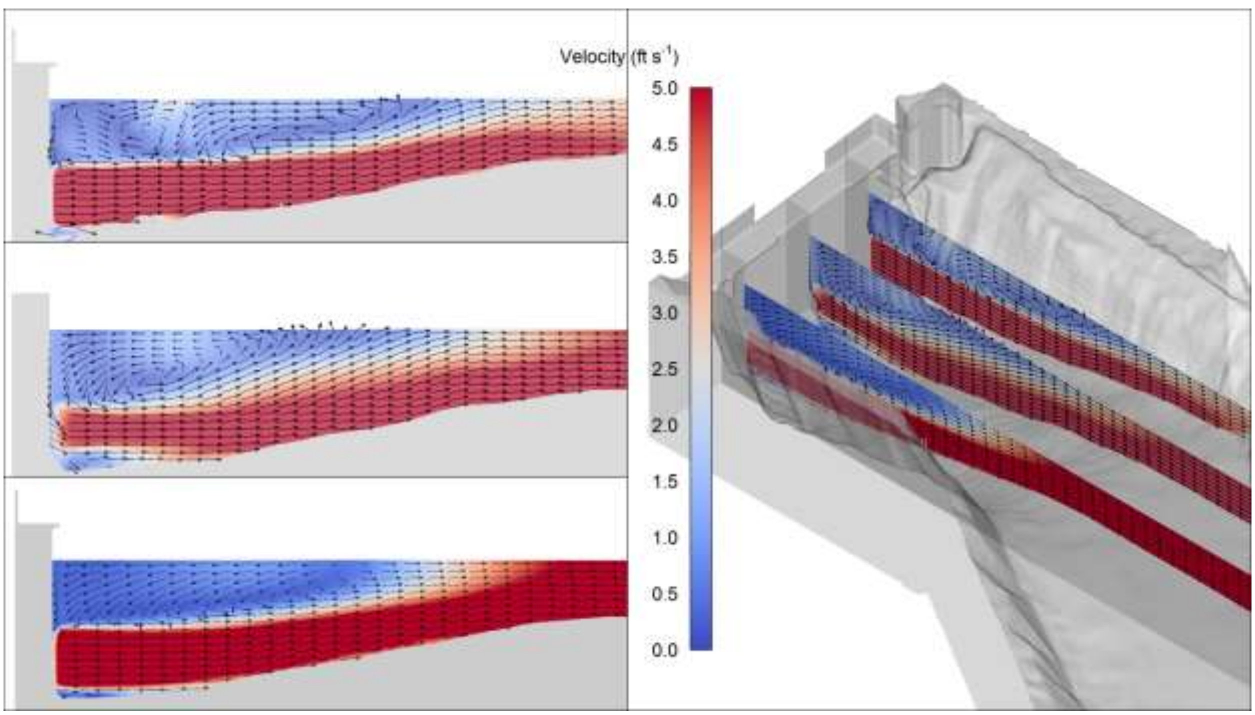

Figure 4-57: Velocity Contours and Vector Profiles for Case 2

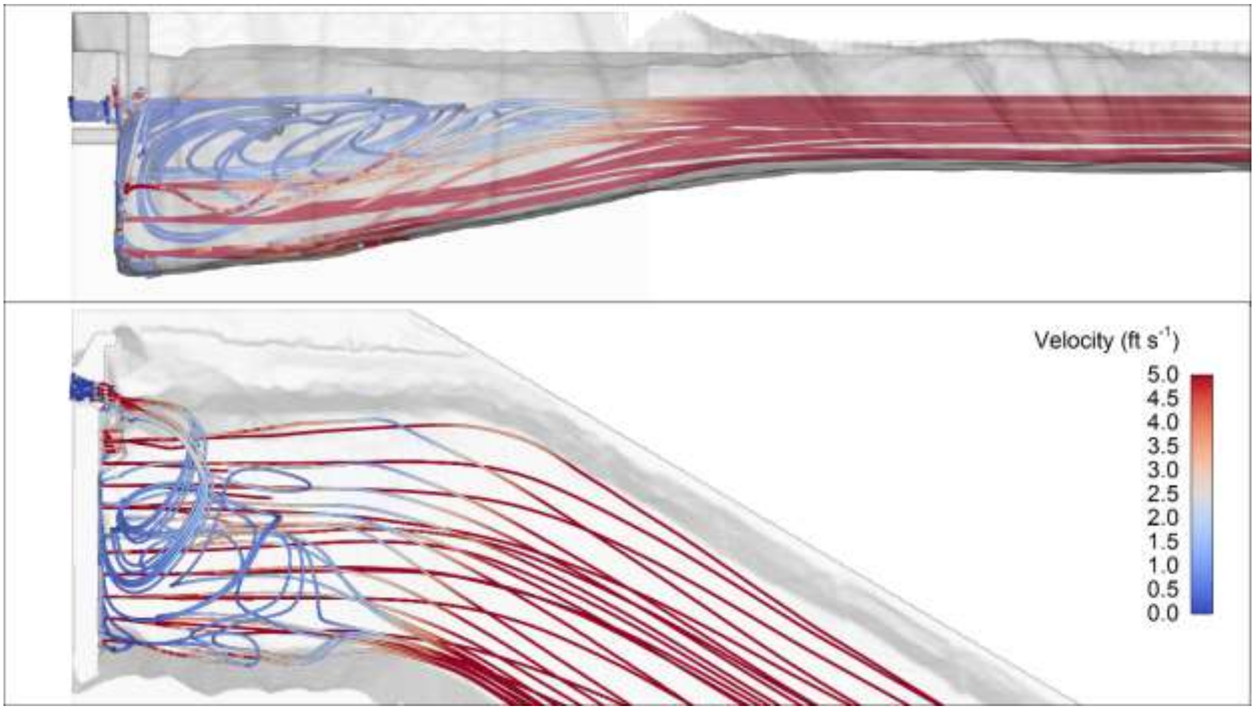

Figure 4-58: Streamlines Colored by Velocity Showing Flow Patterns for Case 2

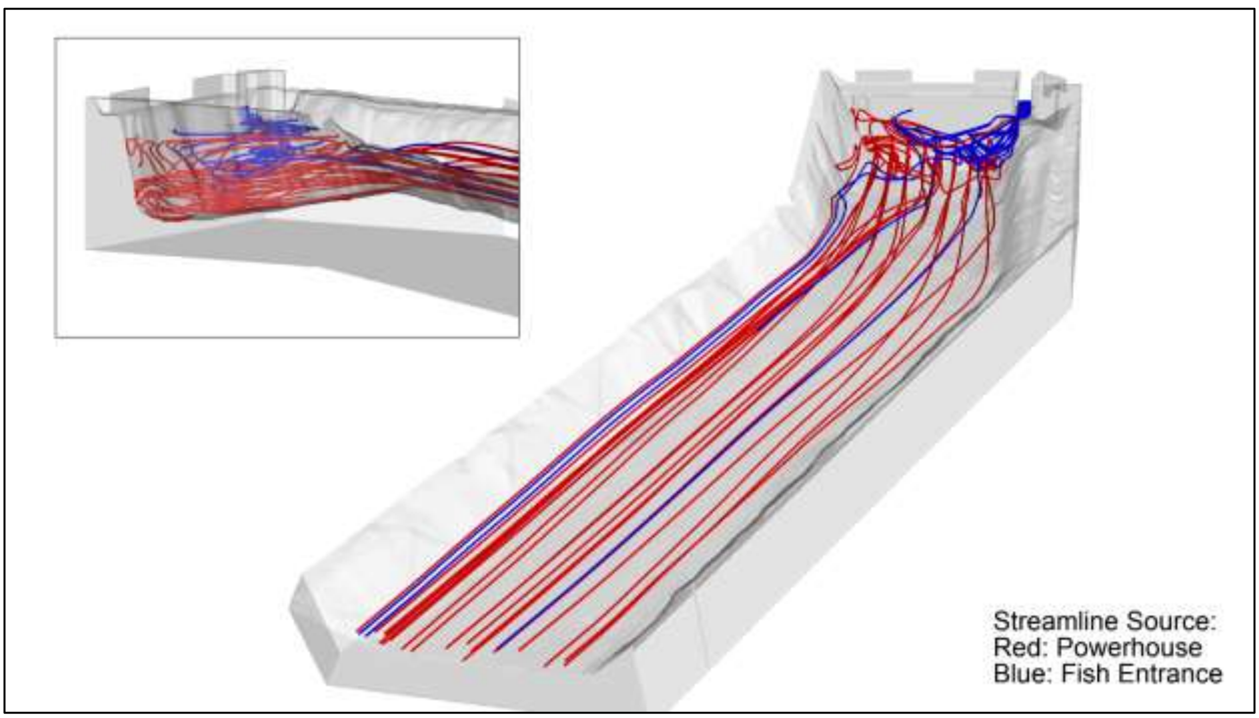

Figure 4-59: Streamlines Colored by Model Outlet Showing Flow Patterns for Case 2

The velocity patterns show a recirculation area above the powerhouse discharges. The recirculation creates two areas of flow separation near the powerhouse discharge. The flow from the AWS mixes with the powerhouse discharge. The velocity near the AWS outlet is above 5.0 ft per second and the flow path was limited to the top 5 ft of the water column during modeling.

5 Conclusions

Three CFD models were developed to help assess the hydraulic conditions near the Pawtucket Dam fish ladder and the E.L. Field Powerhouse forebay and tailrace. Multiple simulations were run for each model and the results were graphically presented.

The Pawtucket Fish Ladder simulations showed increased velocities over the baffles placed in the fish ladder slots and a recirculation area was present in the 180° near the inside wall. The fish ladder entrance showed an area of flow recirculation or cross flow for the modeled events. The bypass weirs showed increased velocities at the gaps.

The E.L. Field Powerhouse forebay CFD model showed a large recirculation area upstream of the AWS. Flow entering the AWS was limited to the upper portion of the water column and originated along the left bank (viewing downstream) of the channel. The AWS flow would travel from the left bank to the upstream powerhouse face before turning parallel to the powerhouse face and entering the AWS.

The E.L. Field Powerhouse tailrace CFD model showed a mixed flow at the downstream boundary. Flow exiting the river side AWS was immediately mixed with the powerhouse discharge. Flow from the powerhouse would change direction and flow toward the powerhouse, mixing the AWS flow into the water column. The low tailrace elevation shows a limited flow path and velocities exceeding 5.0 ft per second at the AWS outlet.

6 References

Flow Science, Inc. (2019). *User Manual: FLOW-3D Documentation, Release 12.0.0.* Albuquerque: Flow Science, Inc.

Hirt, C. W., & Nichols, D. B. (1981). Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *Journal of Computational Physics*, 201-225.

Orszag, S. A., & Yakhot, V. (1986). Renormalization Group Analysis of Turbulence. *Internation Congress of Mathmaticians* (pp. 1395-1399). Berkley: Princeton University.

US Fish and Wildlife. (2014, November 20). Lowell Weir Survey_141120.xls.