Three-Dimensional Computational Fluid Dynamics Model Study Report

Lawrence Hydroelectric Project

(FERC No. 2800)

April 27, 2026

Prepared by:

HDR

This page is intentionally left blank.

Contents

Tables

Figures

List of Acronyms

3D	three dimensional
AWS	auxiliary water supply
CFD	Computational Fluid Dynamics
CFR	Code of Federal Regulations
cfs	cubic feet per second
DEM	digital elevation model
EA	Environmental Assessment
Essex	Essex Company, LLC
FERC	Federal Energy Regulatory Commission (or Commission)
FPA	Federal Power Act
Fps	foot per second
ft	feet
GIS	Geographic Information System
ILP	Integrated Licensing Process
LiDAR	Light Detection and Ranging
MADMF	Massachusetts Department of Marine Fisheries
MassWildlife	Massachusetts Division of Fisheries and Wildlife
NHFG	New Hampshire Fish and Game
NMFS	National Marine Fisheries Service
Project	Lawrence Hydroelectric Project (or Lawrence Project)
RNG	Renormalized Group
RSP	Revised Study Plan
URANS	Unsteady Reynolds Averaged-Navier Stokes
USC	United States Code
USFWS	U.S. Fish and Wildlife Service
WSE	water surface elevation

1.0 Introduction and Background

1.1 Background

Essex Company, LLC (Essex), a subsidiary of Patriot Hydro, LLC, is the Licensee, owner, and operator of the Lawrence Hydroelectric Project (FERC No. 2800) (Project or Lawrence Project). The Project was licensed by the Federal Energy Regulatory Commission (FERC or Commission) in December 1978, and the license expires on November 30, 2028. The Lawrence Project is located on the Merrimack River in the City of Lawrence in Essex County, Massachusetts.

The Project is currently licensed by the Commission under the authority granted to FERC by Congress through the Federal Power Act (FPA), 16 United States Code (USC) §791(a), et seq., to license and oversee the construction and operation of non-federal hydroelectric projects on jurisdictional waters and/or federal lands. In accordance with FERC's regulations at 18 Code of Federal Regulations (CFR) §16.9(b), Essex must file an application for a new license for the Project on or before November 30, 2026. Accordingly, Essex is pursuing a new license for the Project pursuant to the Commission's Integrated Licensing Process (ILP), as described at 18 CFR Part 5 of the Commission's regulations.

FERC's November 28, 2023 Scoping Document 2 identified a variety of aquatic resource issues to be analyzed in the Environmental Assessment for the Project relicensing. The Massachusetts Department of Marine Fisheries (MADMF), Massachusetts Division of Fisheries and Wildlife (MassWildlife), New Hampshire Fish and Game (NHFG), National Marine Fisheries Service (NMFS), and the U.S. Fish and Wildlife Service (USFWS) submitted formal study requests for a Three-Dimensional Computational Fluid Dynamics (3D CFD) Model of the Project's fish passage facilities, approaches, and routes.

The goal of this study is to determine the flow field conditions that exist in and around the Lawrence Project's upstream and downstream migratory fish passage routes. This is anticipated to aid in the interpretation of conditions for the guidance of migrating fish to and through the fish passage facilities. The objectives of this study are to:

1. Develop and calibrate 3D models of areas pertinent to fish passage structures including the Essex Powerhouse forebay and downstream bypass, guidance boom, tailrace, and fish lift.
2. Simulate various operational conditions using each model.
3. Produce a series of color contour maps depicting flow fields relating to attraction and hydraulics; and
4. Develop a 2D flow model for the downstream reach (i.e., the Merrimack River from the downstream face of the Project's dam to the downstream side of the Union Street Bridge).

This report describes numerical methods, model geometry, mesh development, boundary conditions, and results. The Study area includes a portion of the upstream reservoir, Essex Dam, fish lift, downstream fish bypass, powerhouse forebay, tailrace, and the downstream reach.

2.0 CFD Model Development

2.1 Model Description

FLOW-3D HYDRO is a commercially available software developed and supported by Flow Science, Inc. that can solve the 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. FLOW-3D POST 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 HYDRO are provided in the software user's guide (Flow Science, Inc. 2025). The software solves fully URANS equations on structured grids. Model-fitted meshes were developed in three configurations:

1. **Essex Powerhouse Forebay and Downstream Fish Bypass Model (Upstream/Forebay Model):** the upstream model, consisting of the reservoir, powerhouse forebay, dam, guidance boom, fish lift flow entrance (diffuser, fish lift), and downstream fish bypass.
2. **Essex Powerhouse Tailrace Model (Tailrace Model):** the tailrace model consisting of the powerhouse draft tubes, tailrace, fish bypass, auxiliary water supply (AWS) diffuser and fish lift entrance gates.
3. **Essex Fishway Model/Downstream:** A downstream model, consisting of the same geometry as the tailrace model and approximately 1,500 feet of the downstream reach.

The north canal entrance is present in the model, however there is no flow into the canal in the model.

2.1.1.1 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.

2.1.1.2 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).

2.1.1.3 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 report 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 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 the Normandeau Associates, Inc (Normandeau) provided in coordinate system UTM Zone 19N, NAD 83 2011, units meters, North American Vertical Datum of 1988 (NAVD 88), units meters.
- Bathymetry data collected by Normandeau from two areas; 1) the impoundment 2) the downstream each below the dam (as part of the downstream model). 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) LAS format 2011. Data provided in coordinate system Universal Transverse Mercator (UTM) Zone 19N, NAD 83, units meters, vertical datum NAVD 88, units meters.

Project drawings were used to develop 3D representation of the fish passage structures and other pertinent Project facilities and compiled into a full computer-aided drawing (CAD) representation for each of the model areas. The CAD files were then used to build 3D hydraulic models.

3.0 Mesh and Geometry 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. 2025).

3.1 Upstream/Forebay Model

3.1.1 Model Validation

A traditional mesh sensitivity study was not needed for the Project, as measured velocity field data is available. Case 1 was used for model verification. The mesh was developed and the results were compared between the model and measured data. This comparison is discussed in additional detail, including the measured velocity field data and results of the comparison, in Section 4.1.

The upstream model was developed using three mesh blocks: an upstream channel block, ending approximately 500 feet upstream of the dam; a dam mesh block, extending from the edge of the powerhouse to the left bank of the river; and the powerhouse/forebay mesh block, extending from the draft tubes upstream approximately 450 feet upstream of the powerhouse. The blocks used orthogonal cells with 2’x2’x2’ spacing for the upstream channel and dam and 1’x1’x1’ blocks for the powerhouse/forebay.

3.1.2 Geometry and Mesh

As a part of model verification, the Merrimack River bathymetry and Lawrence Hydro Dam features were added upstream and downstream of the fish passage structures. The combined bathymetry, dam, powerhouse, and fish passage geometry is shown on Figure 3-1.

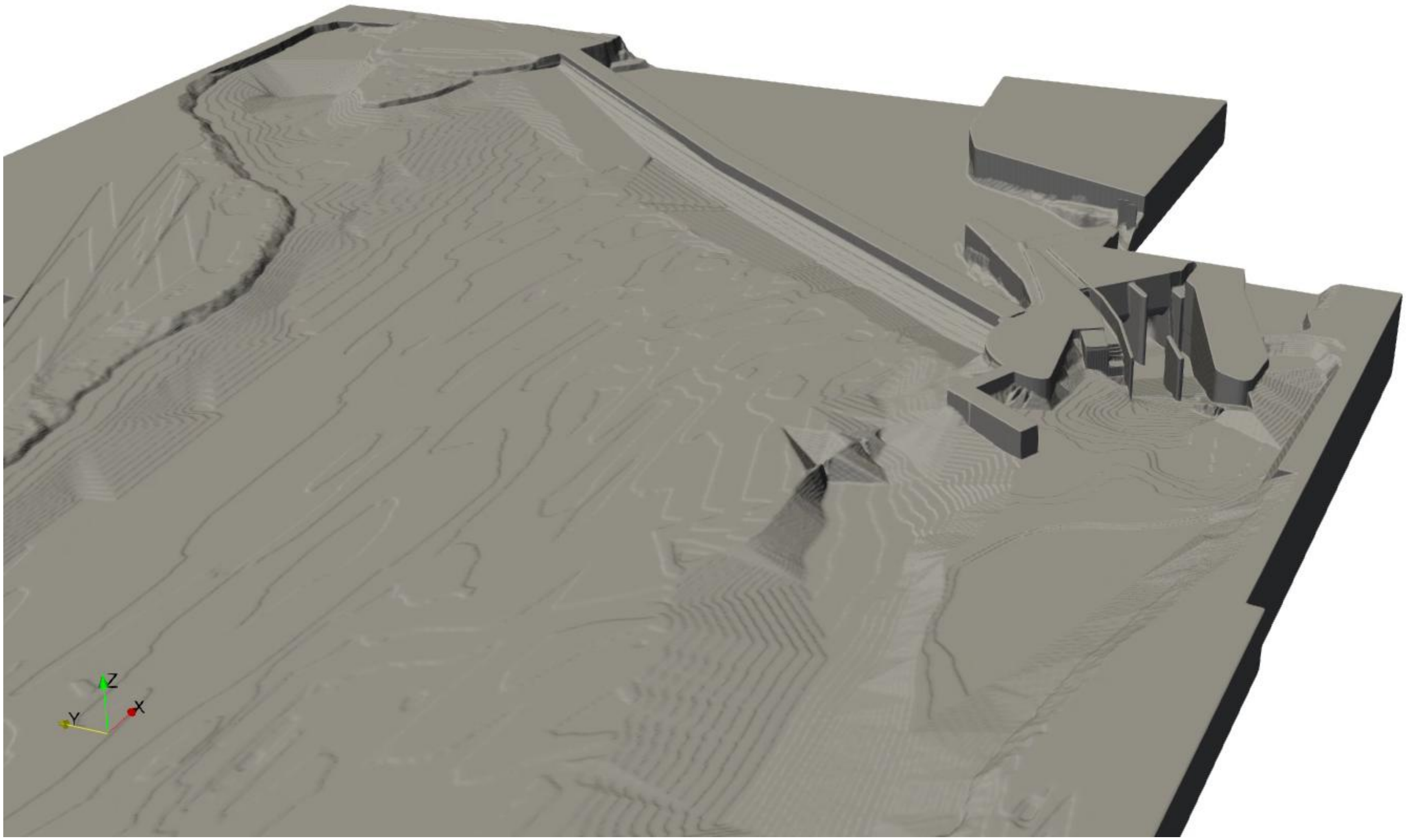

Figure 3-1: Combined Fish Passage, Powerhouse, Dam, Guidance Boom, and Reservoir Bathymetry

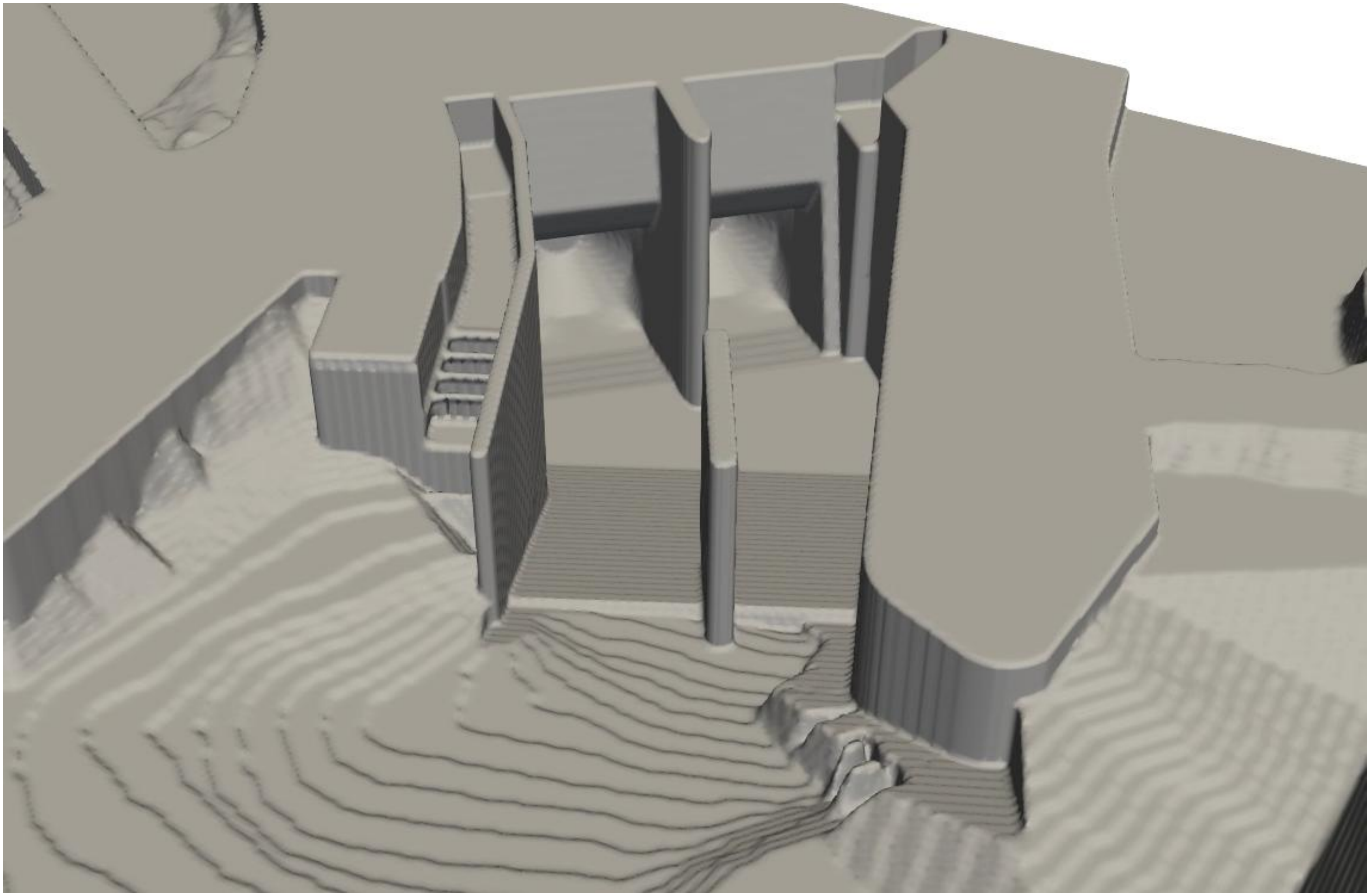

Figure 3-2: Powerhouse Features

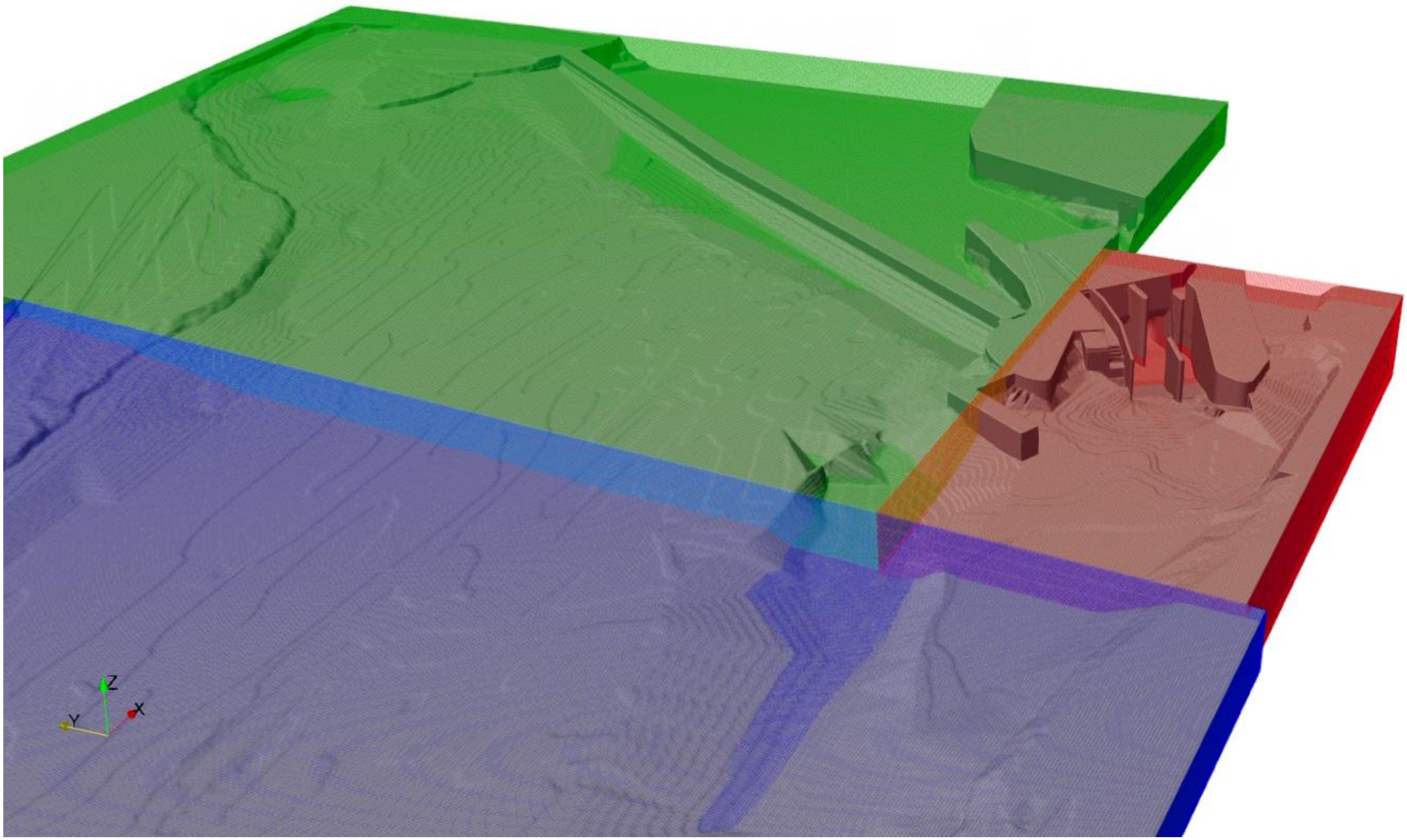

Figure 3-3: Combined Fish Passage, Powerhouse, Dam, and Reservoir Bathymetry Mesh Refinement Zones

3.1.3 Model Scenarios

Table 3-1 lists the model analysis scenarios for the upstream model. The simulations feature different inflow conditions to vary the hydraulics through the reach. Simulation (Case) 1 is unique in that it was used for model validation. Operations during field data collection were replicated in Case 1, and the resulting comparison between modeled and measured data is presented in Section 4.1.

Mass sinks were used to pull flow from the domain, simulating it leaving the forebay through either the units or the fish passage flow entrances (diffuser, fish lift). It was assumed the fish lift and diffuser flow were consistent between simulations 2 through 5. Downstream passage flows are set at 2 percent of total unit flow. The flow (in cubic feet per second [cfs]) over the crest gate for Case 5 was assigned to a mass source using a unit flowrate calculation to evenly distribute flow over the crest. The eel way and fish lift entrance gates are located downstream of the powerhouse and are not included in the upstream model. Spill is pulled from Zone 3, on the North side of the dam. Unless indicated otherwise, the target elevation is in National Geodetic Vertical Datum 1929 (NGVD29).

Table 3-1: Model Analysis Scenarios for the Upstream Model

Case	Total Unit Flow (cfs)	Spill (cfs)	Diffuser (cfs)	Fish Lift (cfs)	Downstream Passage (cfs)	Fish Lift Gate 1 (cfs)	Fish Lift Gate 2 (cfs)	Target Pond Elevation (ft)
1	1,585 (Unit 1 Only)	0	0	0	47.55	0	0	44.2
2	1,000	0	150	50	20	0	0	44.2
3	3,000	0	150	50	60	0	0	44.2
4	8,000	0	150	50	160	0	0	44.2
5	8,000	8,000	150	50	160	0	0	44.2

3.2 Powerhouse Tailrace Model

3.2.1 Geometry and Mesh

A tailrace model was developed to study the hydraulics of flow from the powerhouse and fish passage exits. The full model geometry is shown in Figure 3-4. Approximately 6 feet of each AWS gate and the bypass chute are included in the geometry. The diffuser and fish lift flow entrances are not included in the tailrace model. The eel way is located on the northern side of the powerhouse, outside of the tailrace, and is not included in the tailrace model (but is included in the Essex Fishway/Downstream Model).

The tailrace model was built using four mesh blocks. The tailrace area was modeled using orthogonal cells spaced at 1’x1’x1’. Nested within the tailrace area is the fish bypass block. Increased resolution at 0.5’x0.5’x0.5’ was used to capture the plunging flow of the bypass into the tailrace. The model extends 20 feet upstream into the draft tubes. Individual blocks with orthogonal cells spaced at 0.5’x0.5’x0.5’ were used for each draft tube. The tailrace model mesh is shown in Figure 3-5.

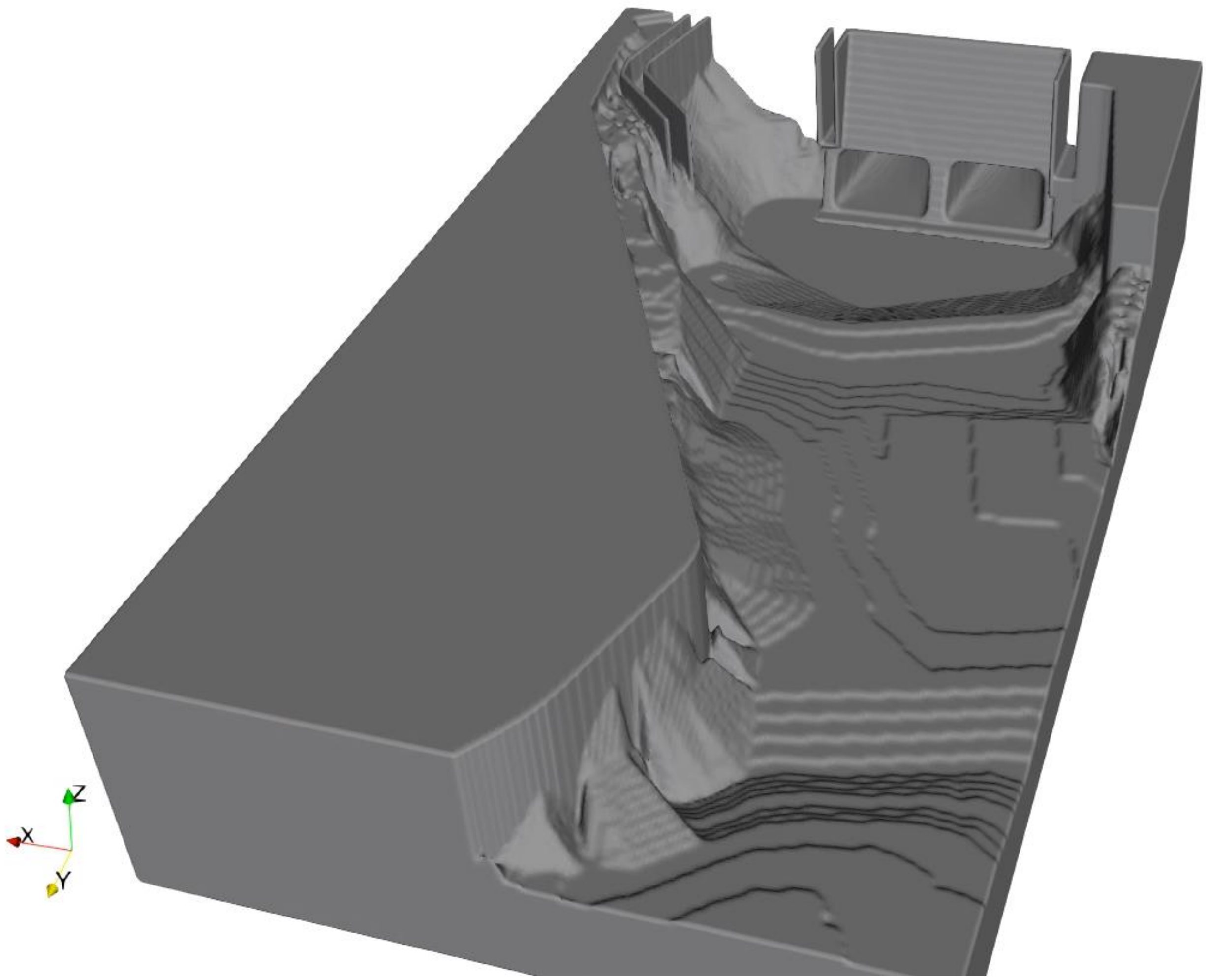

Figure 3-4: Powerhouse Tailrace Model Geometry

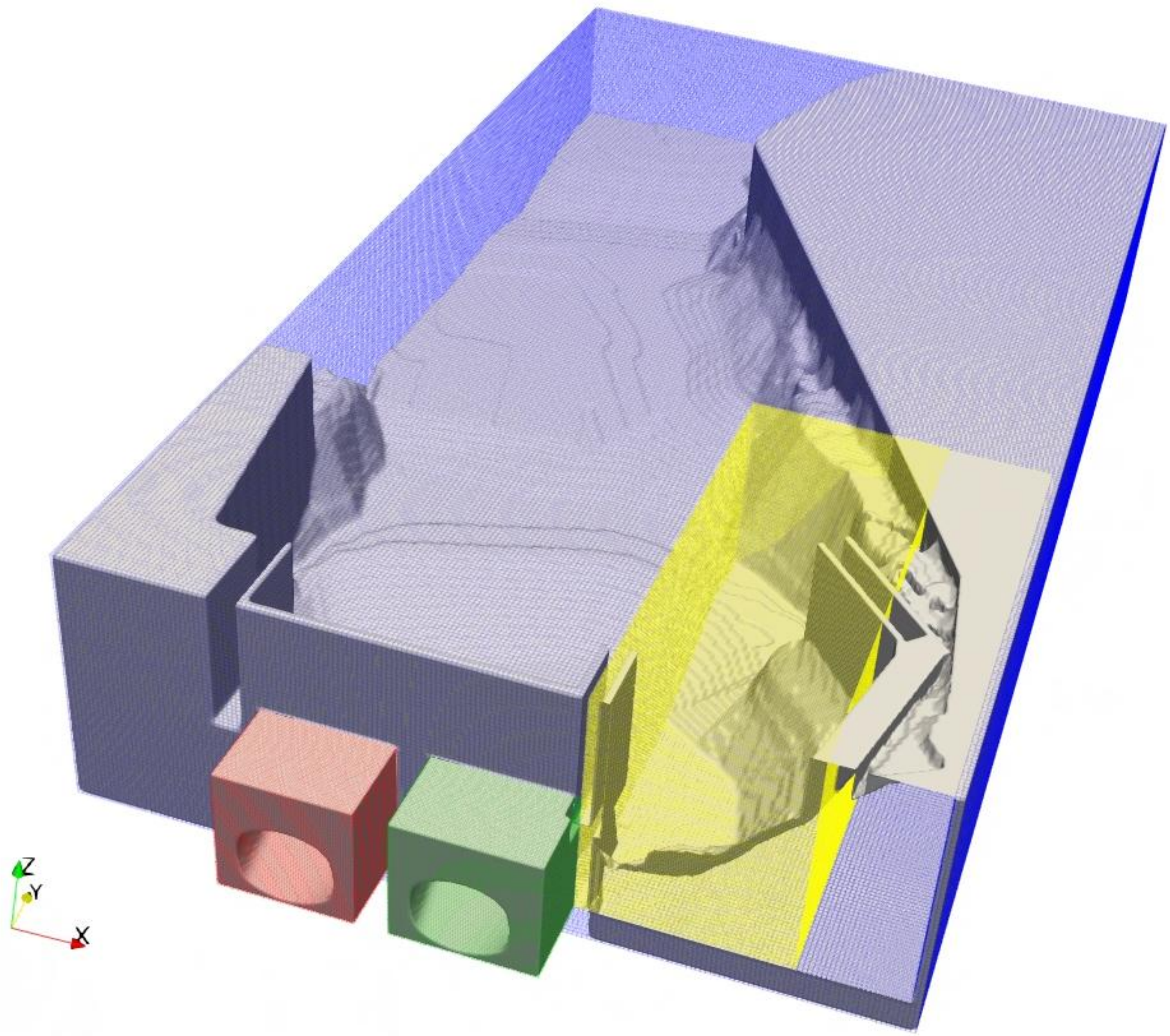

Figure 3-5: Powerhouse Tailrace Model Mesh

3.2.2 Model Scenarios

Table 3-2 lists the model scenarios for the tailrace model. The tailrace model used a constant water surface elevation (WSE) flow from the powerhouse channel as the downstream boundary condition. Mass sources were used to allow flow to be added from the draft tubes, bypass chute, and the auxiliary water supply. The effect of spill (Case 9) was modeled using the elevated tailwater level. Potential crossflow effects are modeled using the downstream model.

Table 3-2: CFD Model Scenarios for Tailrace Model

Case	Total Unit Flow (cfs)	Spill (cfs)	Downstream Passage (cfs)	Eel Way (cfs)	AWS Gate 1 (cfs)	AWS Gate 2 (cfs)	Tailwater Elevation (ft)
6	1,000	0	20	0	120	80	13.4
7	3,000	0	60	0	120	80	14.6
8	8,000	0	160	0	120	80	17.6
9	8,000	8,000	160	0	120	80	22.4

3.3 Essex Fishway/Downstream Model

3.3.1 Geometry and Mesh

A downstream model was developed to study the hydraulics of the flow leaving the tailrace and continuing downstream. The model extends approximately 1,500 feet downstream of the dam. The full model geometry is shown on Figure 3-6. The model utilized the same geometry for the tailrace as the tailrace model, and a detailed view of the powerhouse, bypass chute, and AWS boundaries are shown on Figure 3-4.

The downstream model utilized the same meshing setup for the draft tubes and tailrace as the tailrace model (see Section 3.2.1). Two additional mesh blocks were added to define the dam and the downstream reach. These blocks both utilized orthogonal grids with 2’x2’x2’ spacing. The downstream model mesh blocks are shown in

Figure 3-7.

Figure 3-6: Downstream Model Geometry

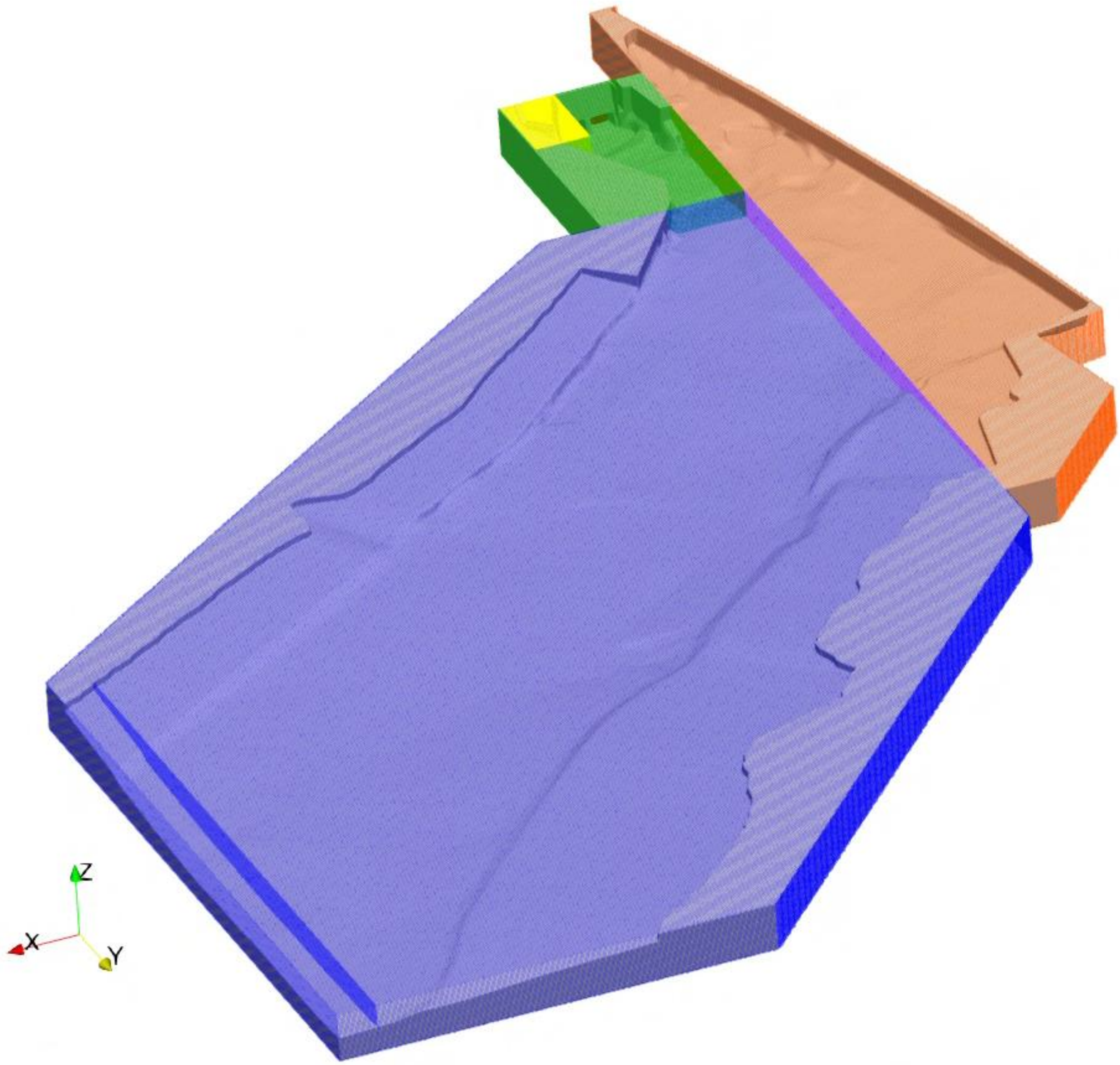

Figure 3-7: Downstream Model Mesh

3.3.2 Model Scenarios

The downstream model used a constant WSE as the downstream boundary condition. Mass sources were used to allow flow to be added from the draft tubes, spillway, bypass chute, and the AWS. As with the tailrace model, the fish lift and diffuser are not included. The eel way is located on the west side of the powerhouse, near the dam's right abutment. Flow is added to the downstream model at the eel way via a mass source. Spill is added to Zone 3, on the North side of the Dam. Table 3-3 lists the model scenarios for the downstream model.

Table 3-3: CFD Model Scenarios for the Essex Fishway/Downstream Model

Case	Total Unit Flow (cfs)	Spill (cfs)	Downstream Passage (cfs)	Eel Way (cfs)	AWS Gate 1 (cfs)	AWS Gate 2 (cfs)	Tailrace Elevation (ft)
10	1,000	0	20	0.67	120	80	13.4
11	8,000	0	160	0.67	120	80	17.6
12	8,000	4,000	160	0.67	120	80	20.0
13	8,000	16,000	160	0.67	120	80	32.0

3.4 Model Approach

3.4.1 Boundary Conditions

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

3.4.1.1 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 upstream/forebay model and the draft tubes for the tailrace and downstream models. A directional vector was applied to the inflow. The water surface was known and was specified for the inflow boundary condition.

3.4.1.2 PRESSURE INLET/OUTLET

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

3.4.1.3 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, AWS, eel way, and powerhouse inlet. A directional vector and uniform velocity distribution were applied to the inflow.

3.4.1.4 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.

3.4.1.5 ROUGHNESS HEIGHT

The roughness height for the structures was set to 0.002 foot, consistent with concrete and 0.08 foot for the topography, consistent with channel roughness values for large rivers.

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.

3.5.1.1 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.

3.5.1.2 MONITORING POINTS

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

4.0 Results

4.1 Upstream/Forebay Model

Five upstream 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 powerhouse units. Streamlines through the fish lift, diffuser and in the immediate upstream region are presented in plan, profile, and isometric view. Plan view of the full model depth averaged velocity is also presented.

4.1.1 Case 1

Case one is the validation case, where velocities were measured in the reservoir by Normandeau in September 2024 using an Acoustic Doppler Current Profiler. The Project operations occurring during the field study were replicated in the model and results are compared at the 14 measurement locations. The measured data is shown on

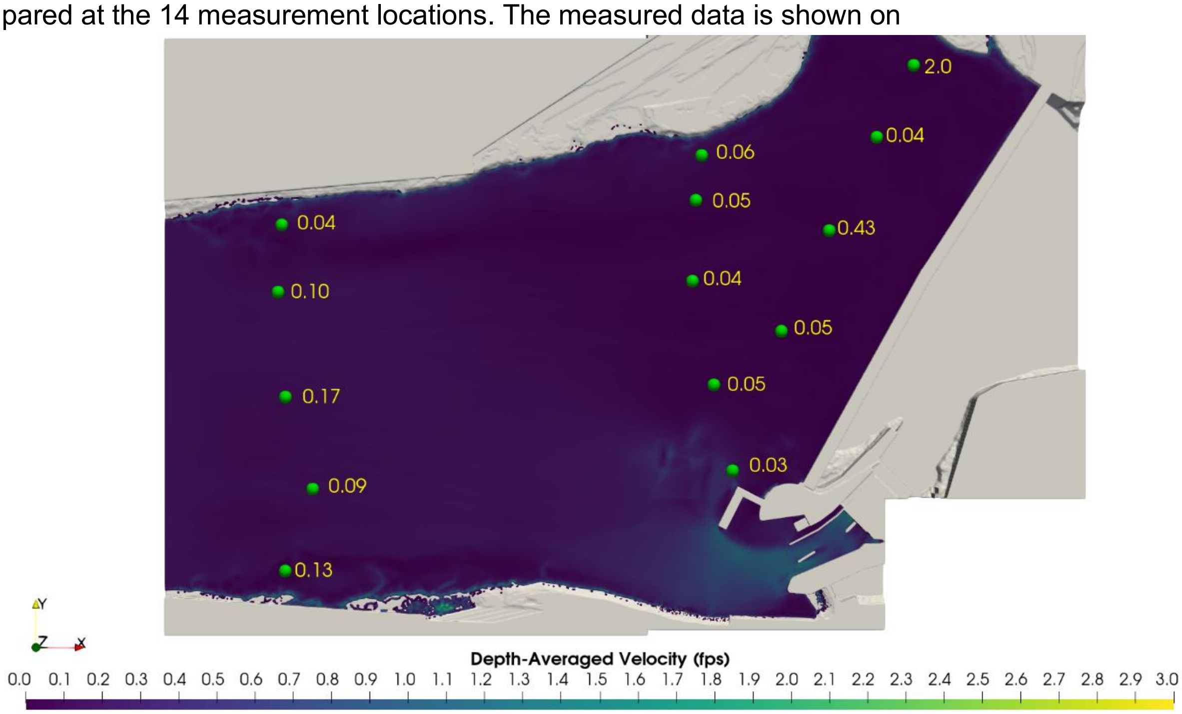

Figure 4-1 and the modeled data is shown on Figure 4-2.

Operations during the field study had 1,585 cfs passing through Unit 1 and a downstream passage flow of 47.55 cfs.

Streamlines colored by velocity are shown on Figure 4-3. Figure 4-4 shows velocity contours for slices through each unit centerline. Figure 4-5 presents plan views of velocity contours and vectors at the diffuser bench elevation in the forebay vicinity, and Figure 4-6 shows the overall model in plan view, with depth-averaged velocity contours.

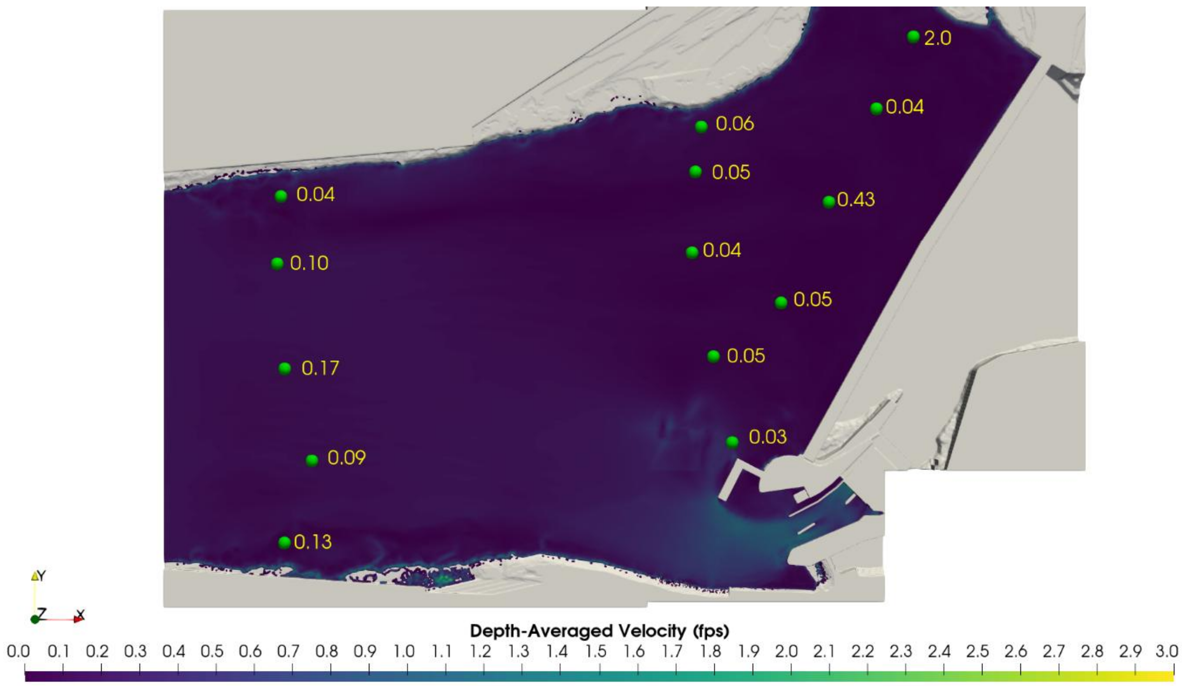

Figure 4-1: Measured Velocities overlaid on Model Contours

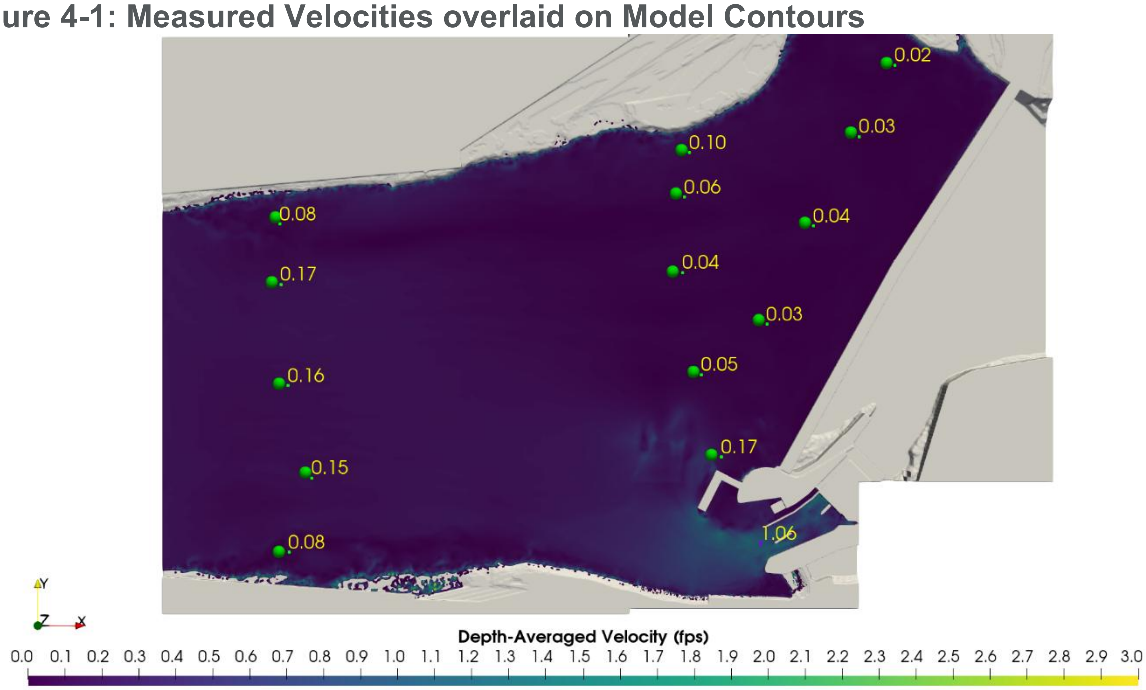

Figure 4-2: Modeled Velocities overlaid on Model Contours

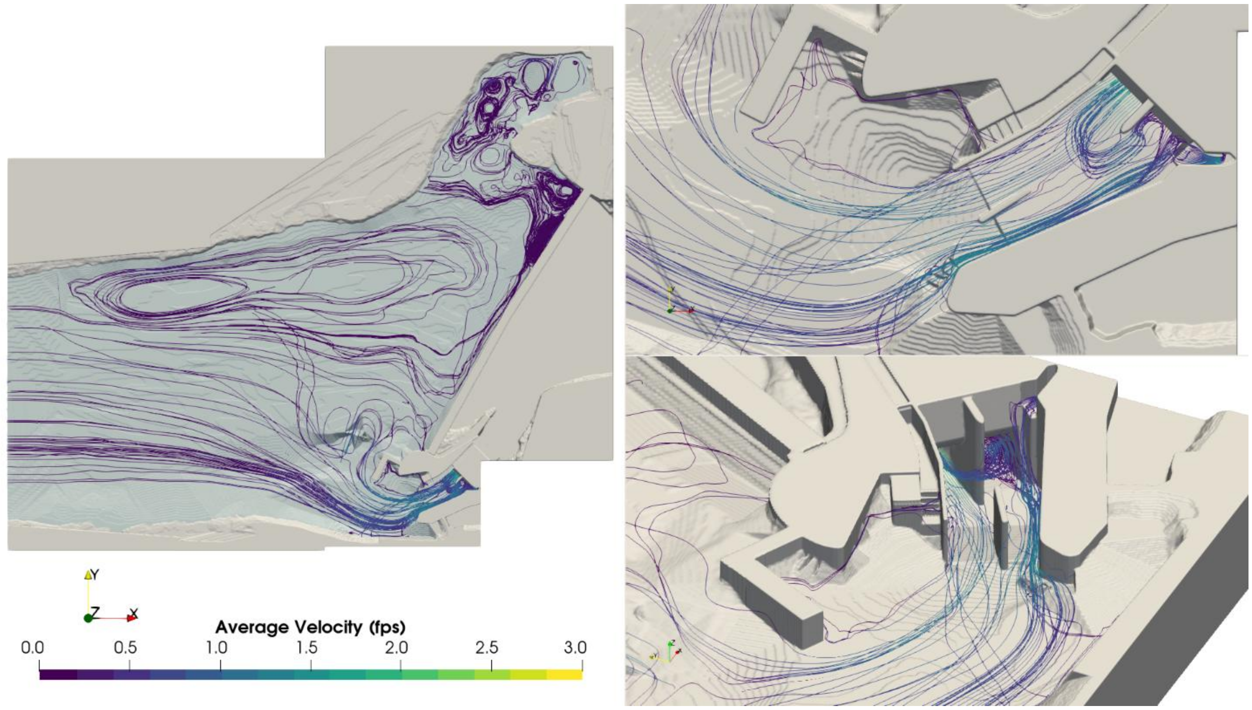

Figure 4-3: Streamlines Colored by Velocity Showing Flow Patterns for Case 1

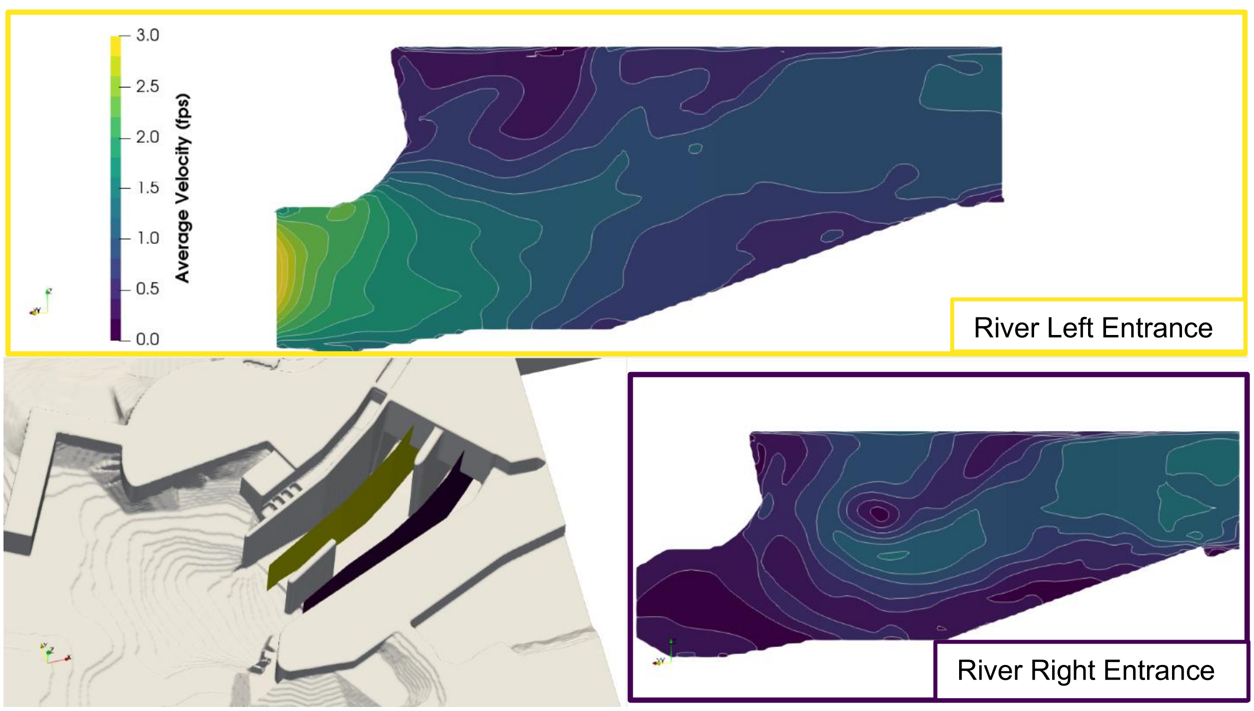

Figure 4-4: Unit Centerline Velocity Contours for Case 1

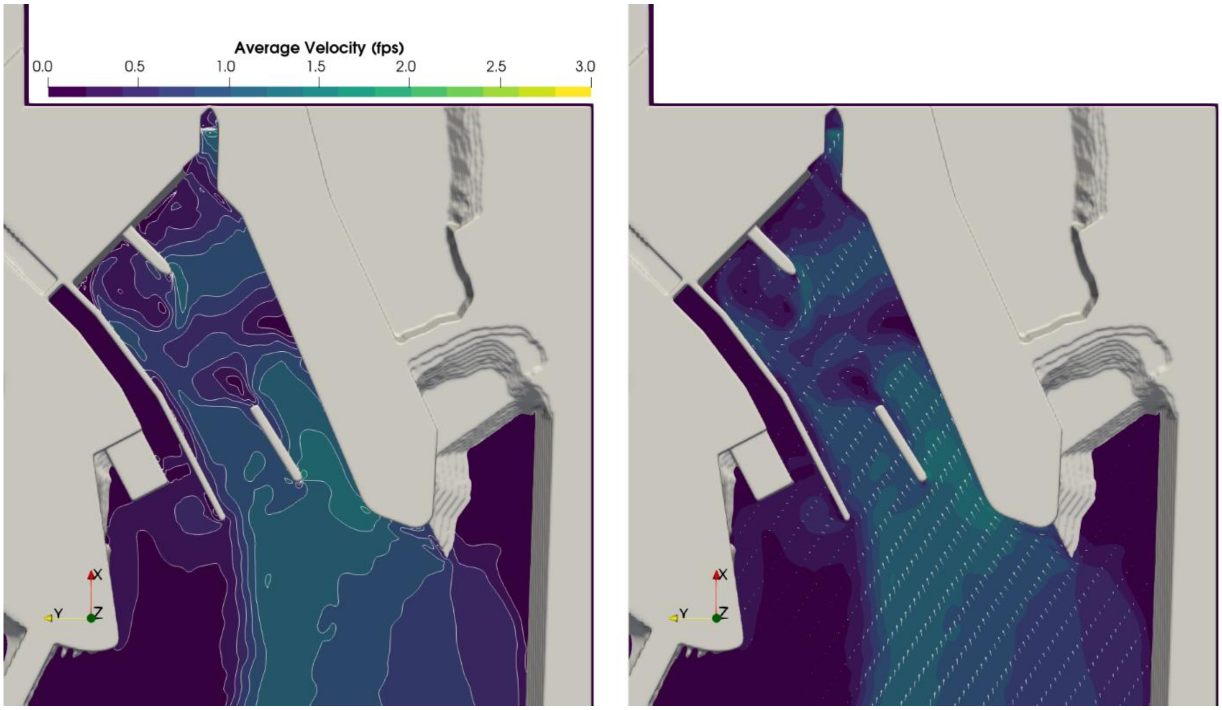

Figure 4-5: Plan Views of Velocity Contours and Vectors in the Forebay Vicinity

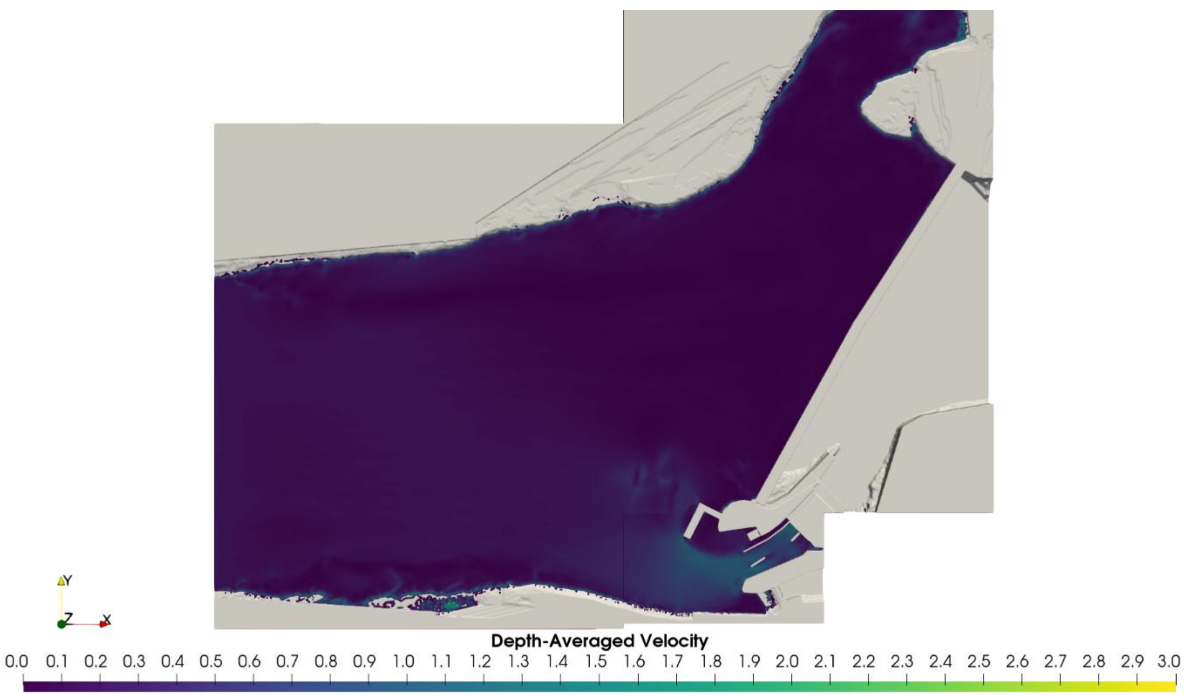

Figure 4-6: Plan View of Depth-Averaged Velocity

The measured and modeled velocities were generally in good agreement. While the magnitude (%) difference between the datasets might appear high, this is due to the very low velocities and the velocity magnitudes are in good agreement.

Attraction flows at the fish lift are well below 1 foot per second (fps), while the diffuser flows peak around 1.5 fps.

4.1.2 Case 2

Case two analyzes low flow through the units and normal diffuser and fish lift operations.

Streamlines colored by velocity are shown on Figure 4-7. Figure 4-8 shows velocity contours for slices through each unit centerline. Figure 4-9 presents plan views of velocity contours and vectors at the diffuser bench elevation in the forebay vicinity, and Figure 4-10 shows the overall model in plan view, with depth-averaged velocity contours.

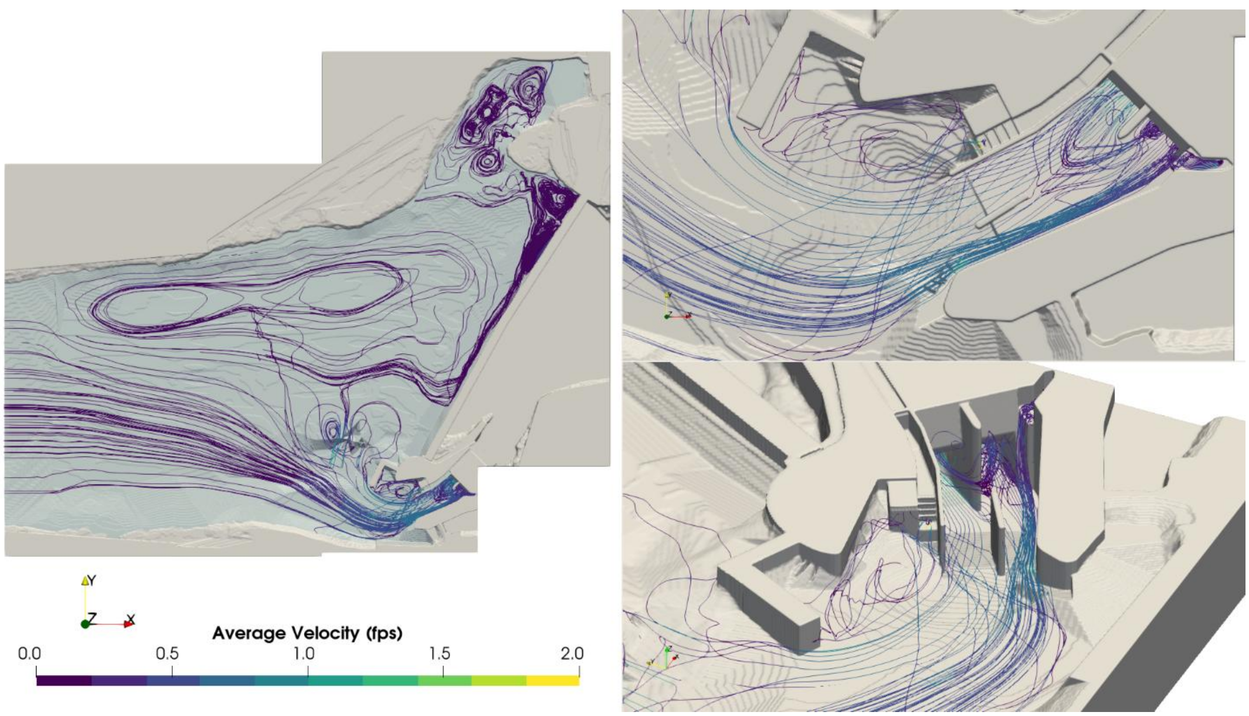

Figure 4-7: Streamlines Colored by Velocity Showing Flow Patterns for Case 2

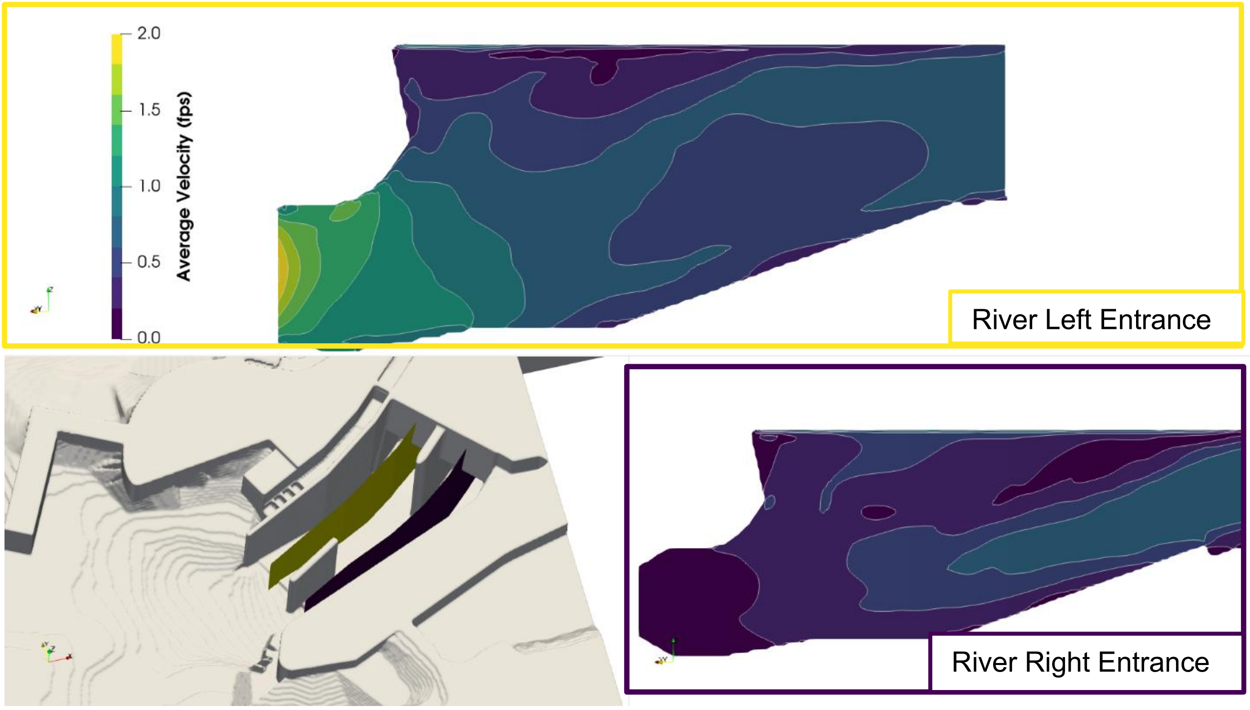

Figure 4-8: Unit Centerline Velocity Contours for Case 2

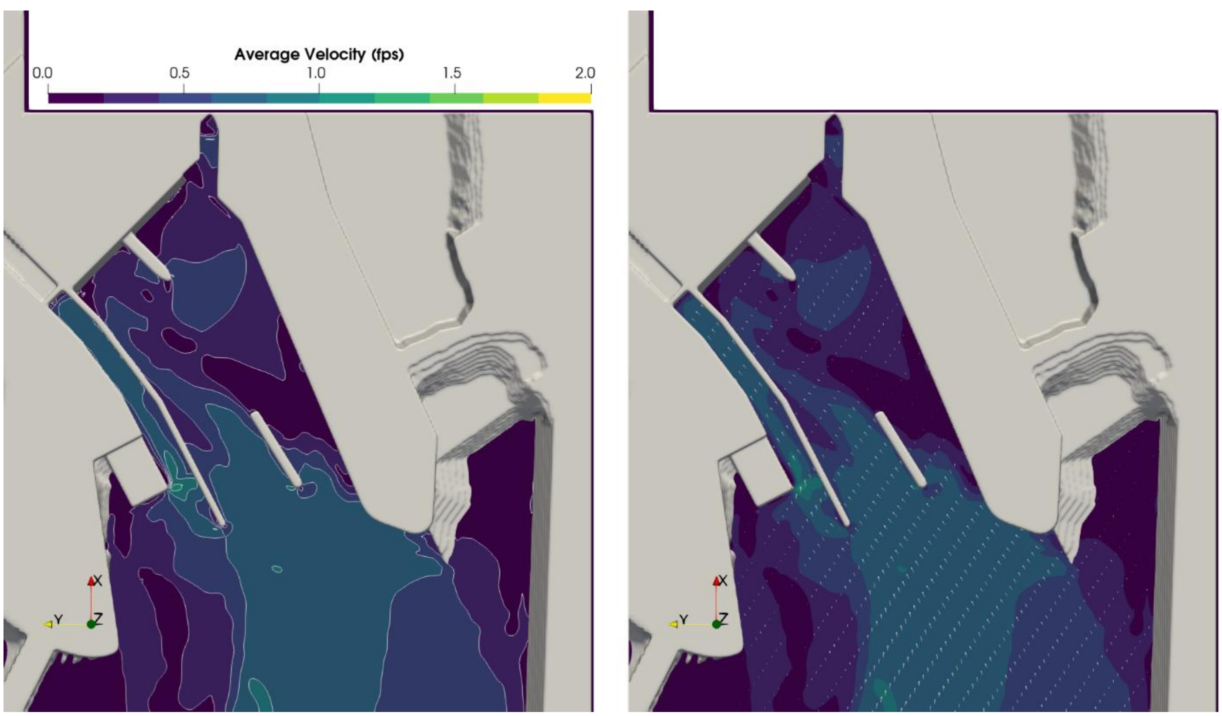

Figure 4-9: Plan Views of Velocity Contours and Vectors in the Forebay Vicinity for Case 2

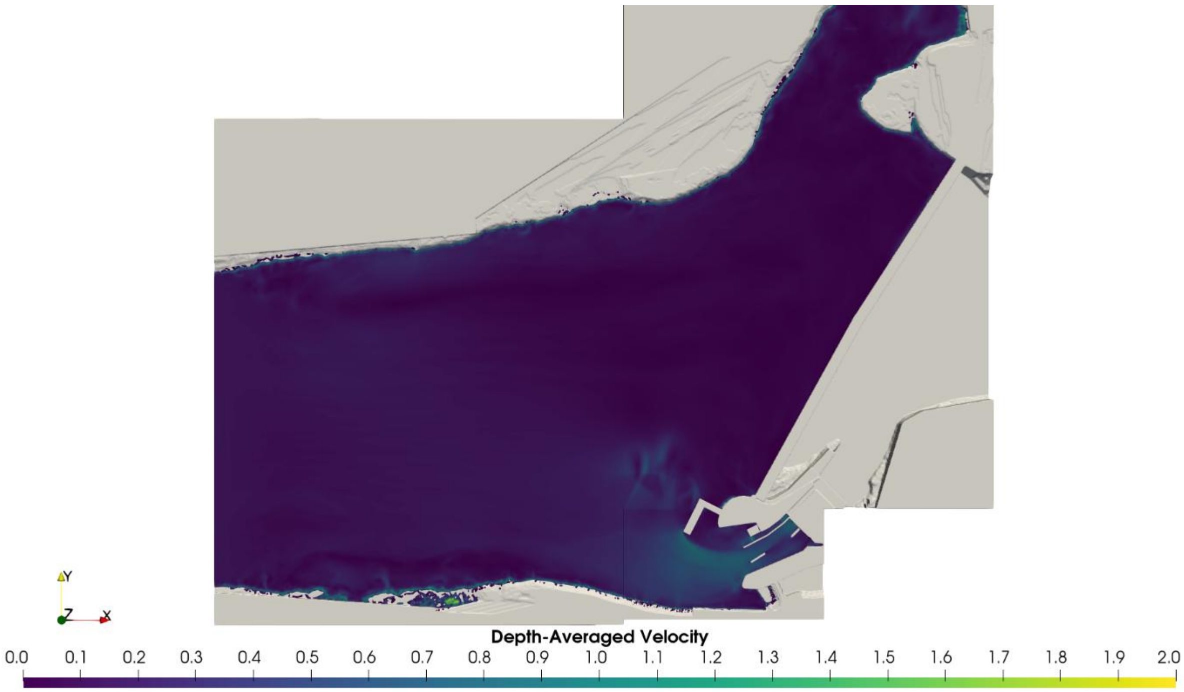

Figure 4-10: Plan View of Depth-Averaged Velocity for Case 2

Low flow rates through the Project produce slow velocities. Velocities at the diffuser and fish lift flow entrance are around 0.5-1.0 fps, roughly matching the surrounding flow into the powerhouse. A recirculation area is present near the fish lift flow entrance.

4.1.3 Case 3

Case 3 simulates a generation discharge of 3,000 cfs and normal fish passage operations.

Streamlines colored by velocity are shown on Figure 4-11. Figure 4-12 shows velocity contours for slices through each unit centerline. Figure 4-13 presents plan views of velocity contours and vectors at the diffuser bench elevation in the forebay vicinity, and Figure 4-14 shows the overall model in plan view, with depth-averaged velocity contours.

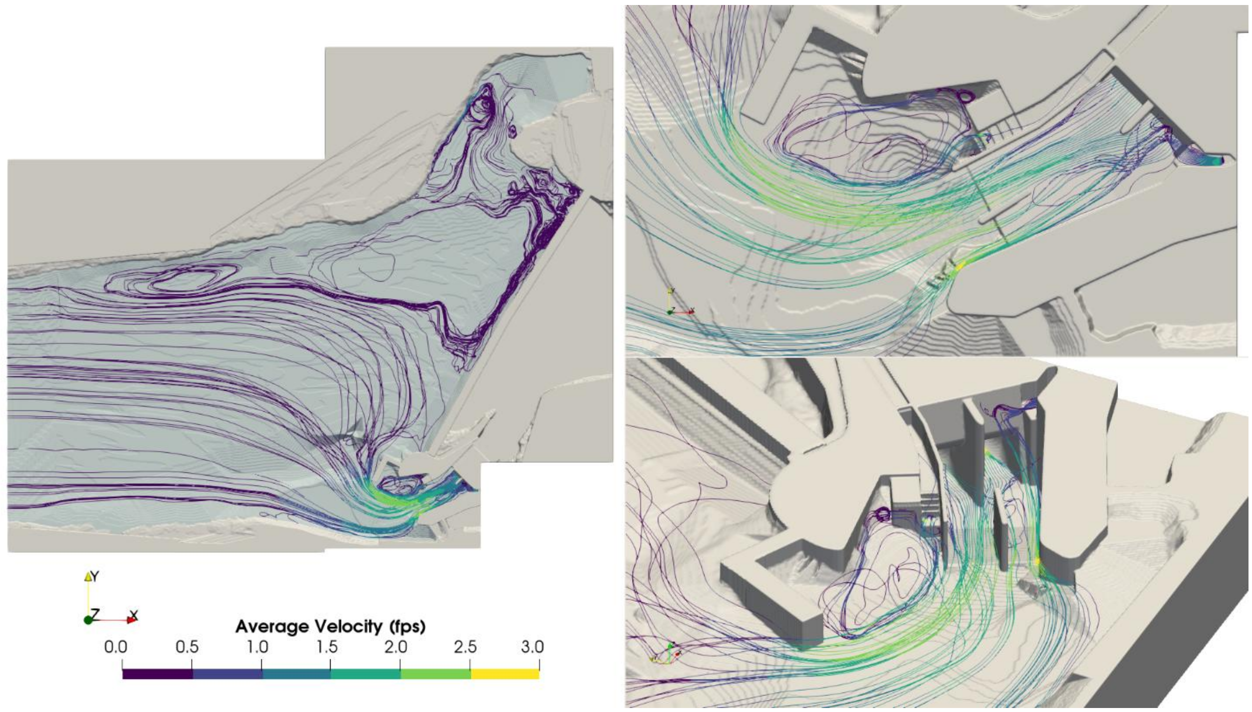

Figure 4-11: Streamlines Colored by Velocity Showing Flow Patterns for Case 3

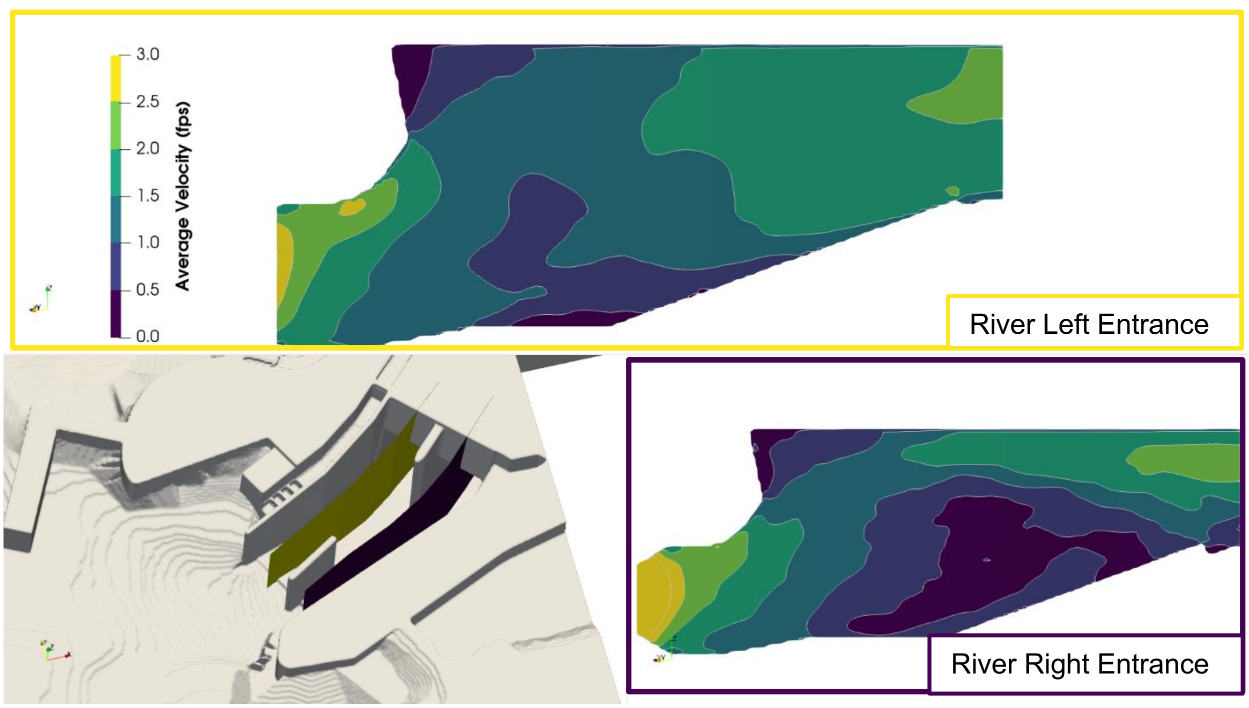

Figure 4-12: Unit Centerline Velocity Contours for Case 3

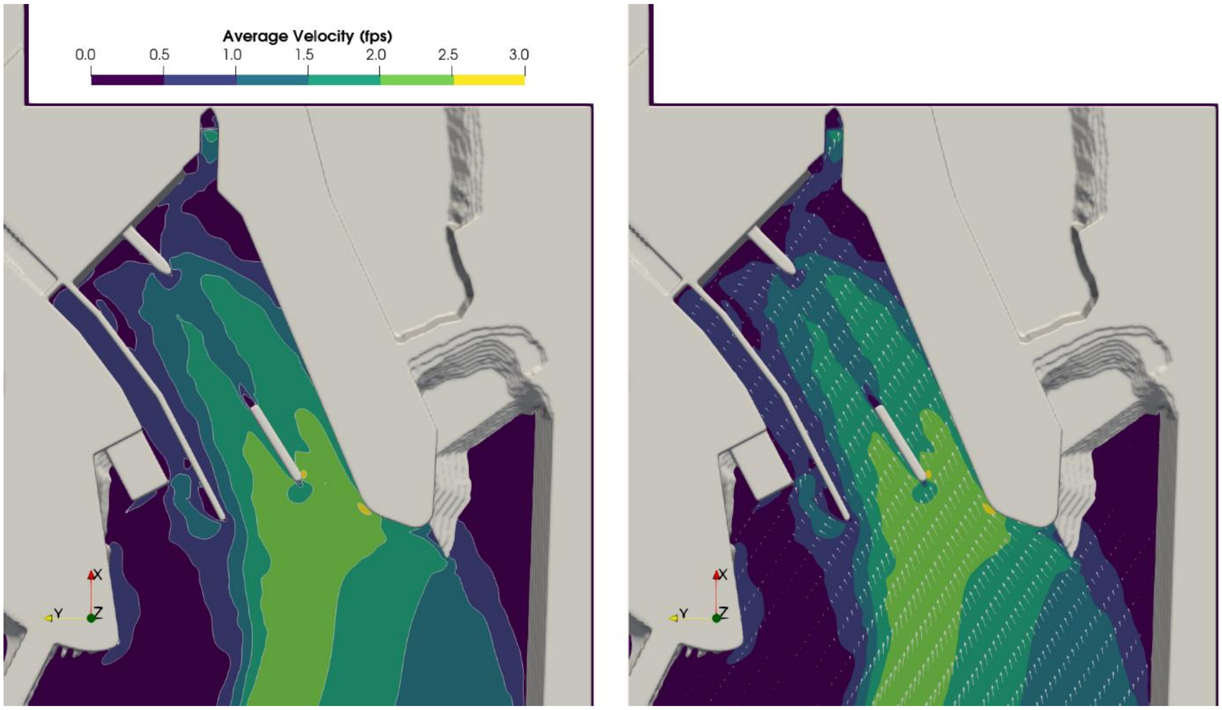

Figure 4-13: Plan Views of Velocity Contours and Vectors in the Forebay Vicinity for Case 3

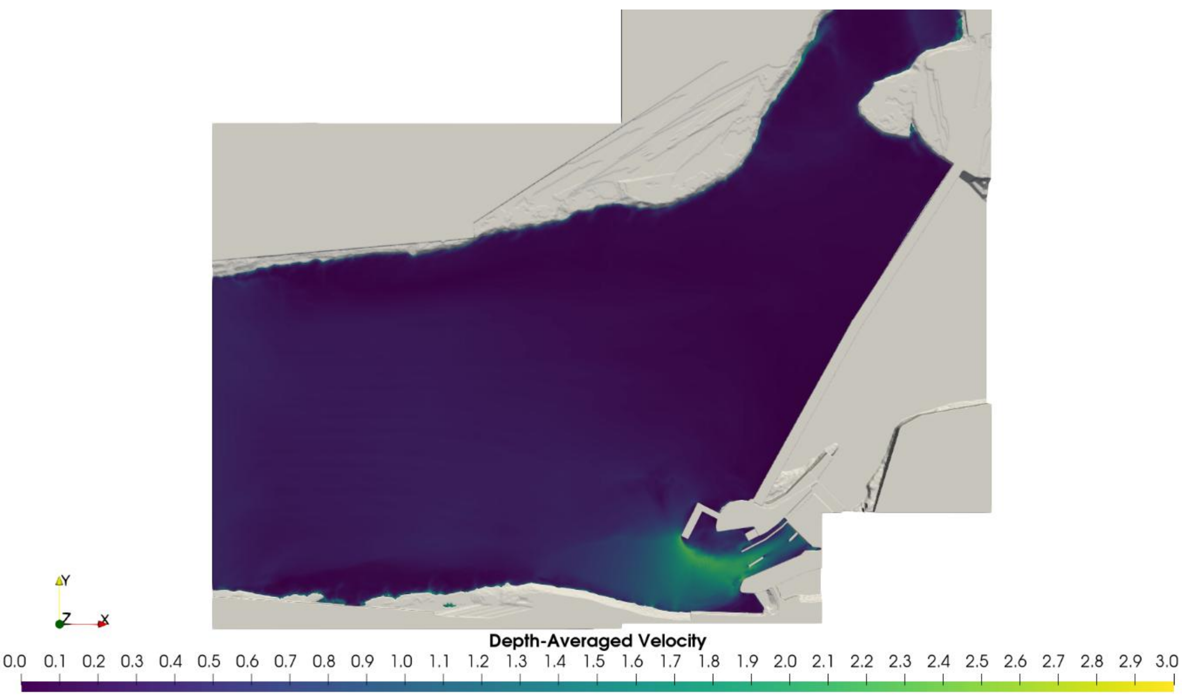

Figure 4-14: Plan View of Depth-Averaged Velocity for Case 3

Increased powerhouse flows create elevated velocities in the forebay with velocities peaking around 1 fps at the fish lift flow entrance. There is an attraction flow present at the diffuser entrance, with velocities also exceeding 2 fps. The elevated velocity at the diffuser entrance is limited to the top of the water column. The configuration of the forebay results in higher velocities along the diffuser side of the forebay. At depth, flows into the units exceed 2 fps.

Recirculation areas are present both near the fish lift entrance and in front of unit 1.

4.1.4 Case 4

Case 4 simulates the maximum generation discharge of 8,000 cfs and normal fish passage operations.

Streamlines colored by velocity are shown on Figure 4-15. Figure 4-16 shows velocity contours for slices through each unit centerline. Figure 4-17 presents plan views of velocity contours and vectors at the diffuser bench elevation in the forebay vicinity, and Figure 4-18 shows the overall model in plan view, with depth-averaged velocity contours.

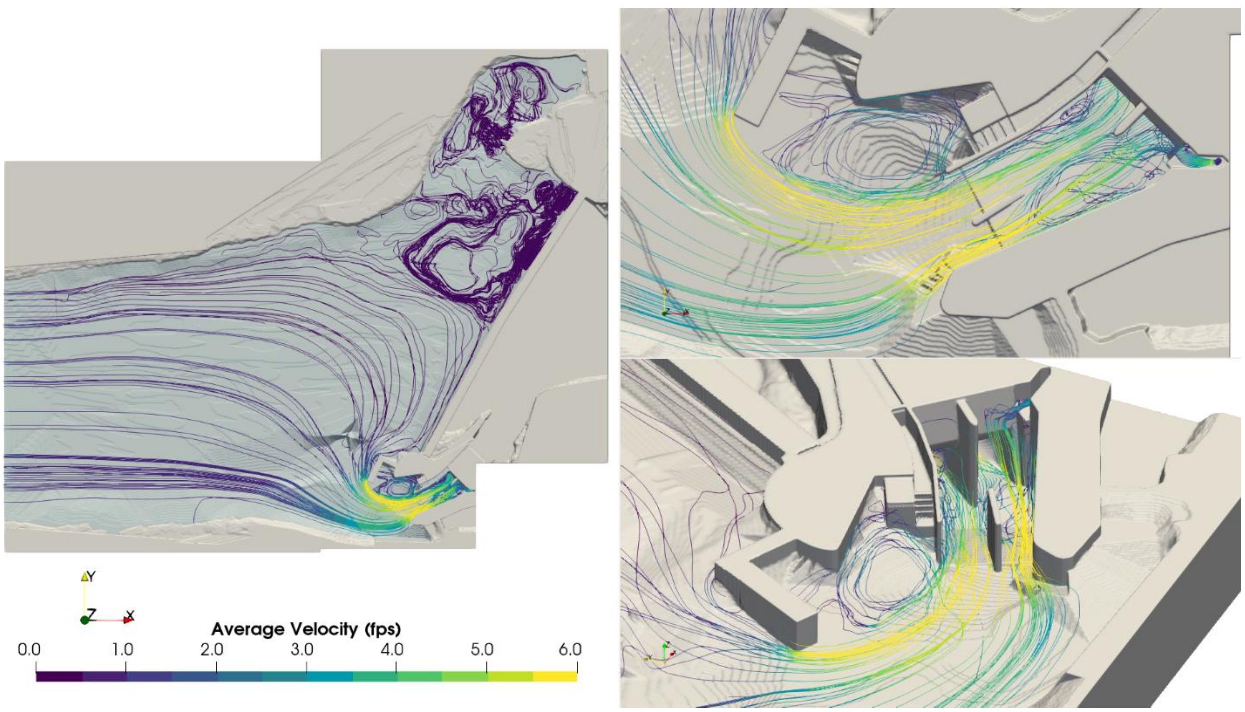

Figure 4-15: Streamlines Colored by Velocity Showing Flow Patterns for Case 4

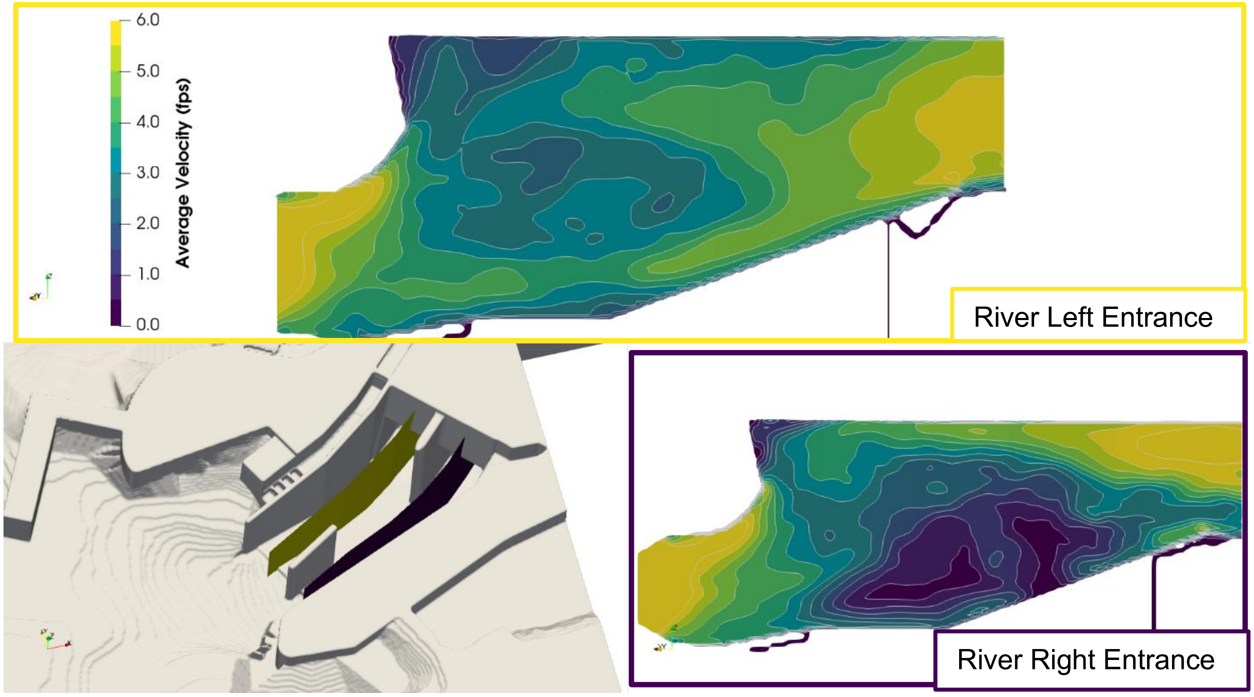

Figure 4-16: Unit Centerline Velocity Contours for Case 4

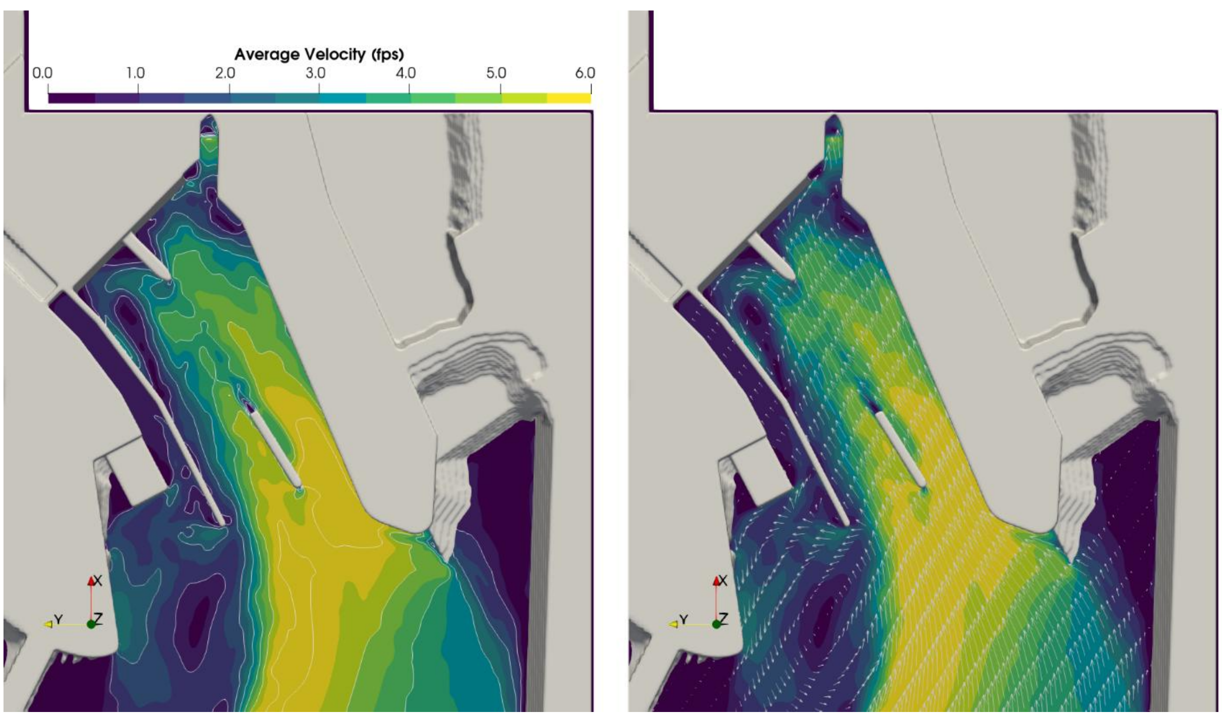

Figure 4-17: Plan Views of Velocity Contours and Vectors in the Forebay Vicinity for Case 4

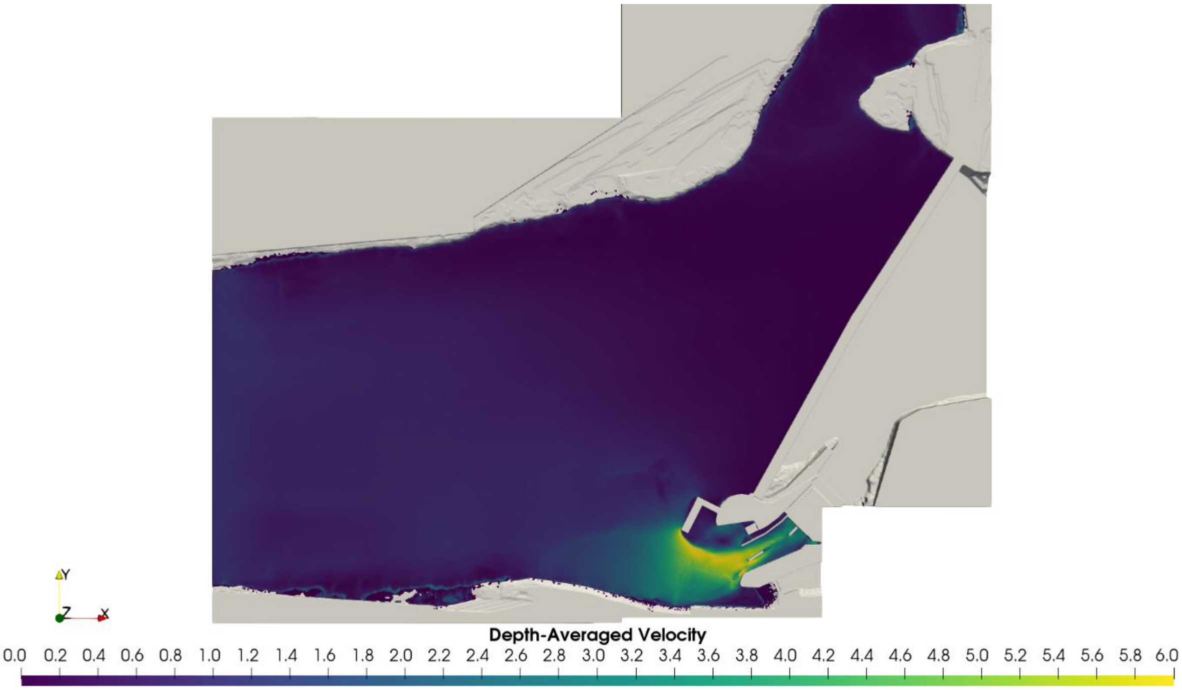

Figure 4-18: Plan View of Depth-Averaged Velocity for Case 4

The maximum unit flowrates create velocities in the forebay between 5 and 6 fps. The fish lift entrance velocities remain around 1 fps, and the diffuser creates an elevated velocity area at the surface, peaking around 5 fps. Surface velocities at the unit entrance are generally less than 1.5 fps, while at depth, velocities at the units exceed 6 fps.

A recirculation area is present near the fish lift entrance.

4.1.5 Case 5

Case 5 simulates the maximum generation discharge of 8,000 cfs and normal fish passage operations. Case 5 also simulates a spillway discharge of 8,000 cfs through the center of the dam.

Streamlines colored by velocity are shown on Figure 4-19. Figure 4-20 shows velocity contours for slices through each unit centerline. Figure 4-21 presents plan views of velocity contours and vectors at the diffuser bench elevation in the forebay vicinity, and Figure 4-22 shows the overall model in plan view, with depth-averaged velocity contours.

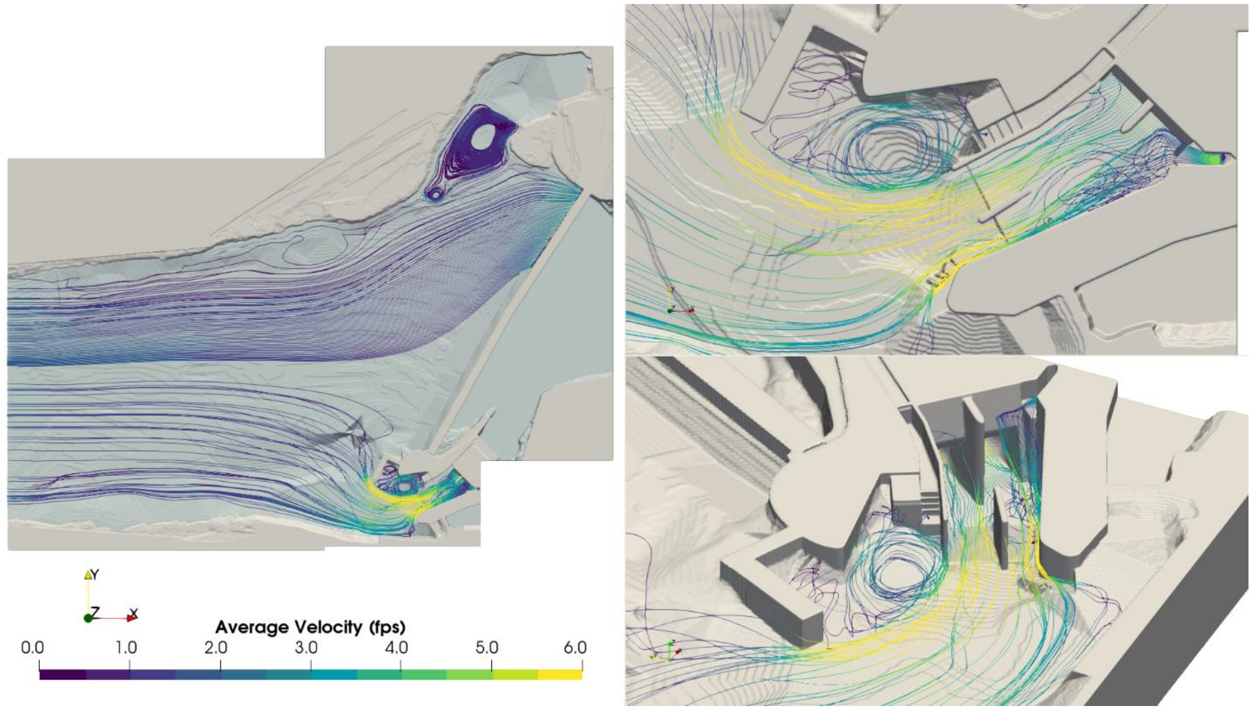

Figure 4-19: Streamlines Colored by Velocity Showing Flow Patterns for Case 5

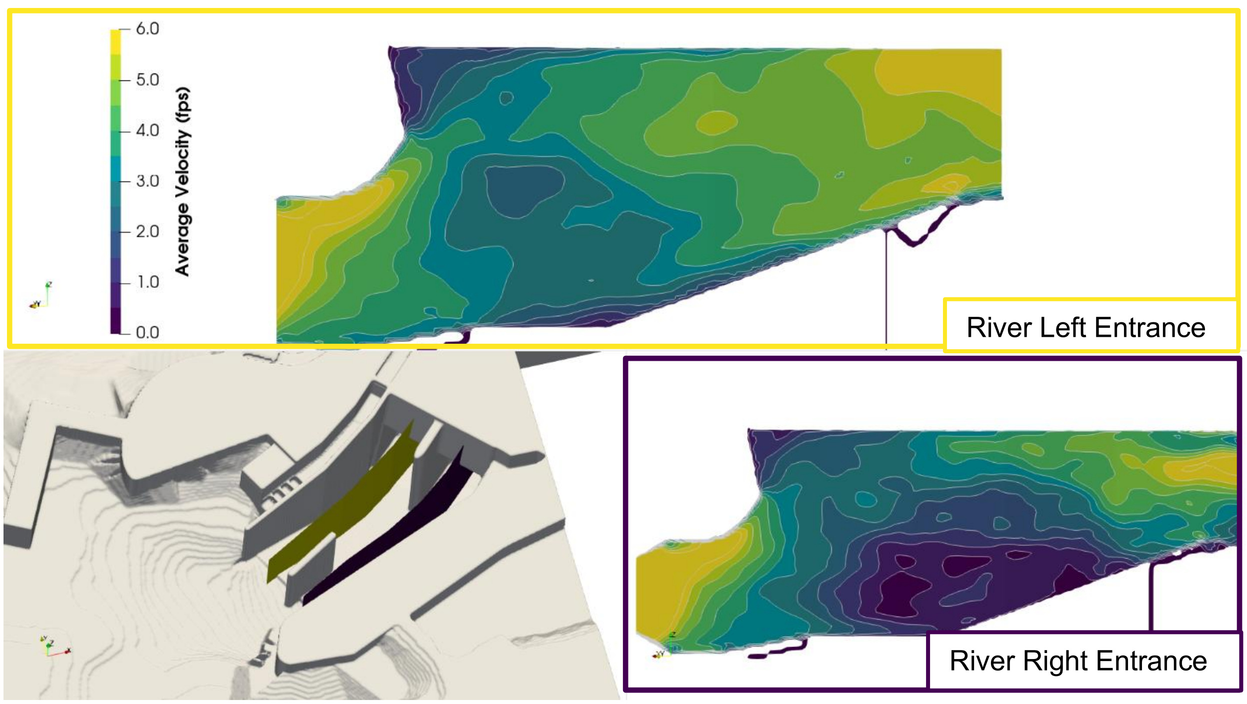

Figure 4-20: Unit Centerline Velocity Contours for Case 5

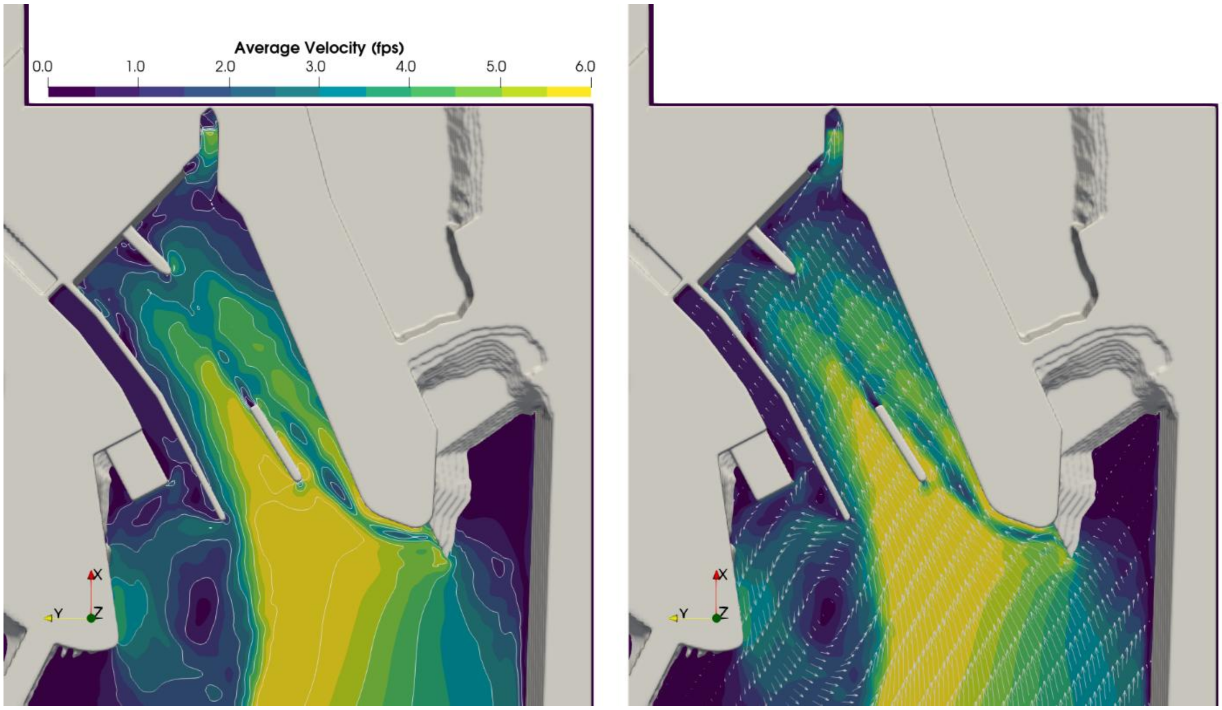

Figure 4-21: Plan Views of Velocity Contours and Vectors in the Forebay Vicinity for Case 5

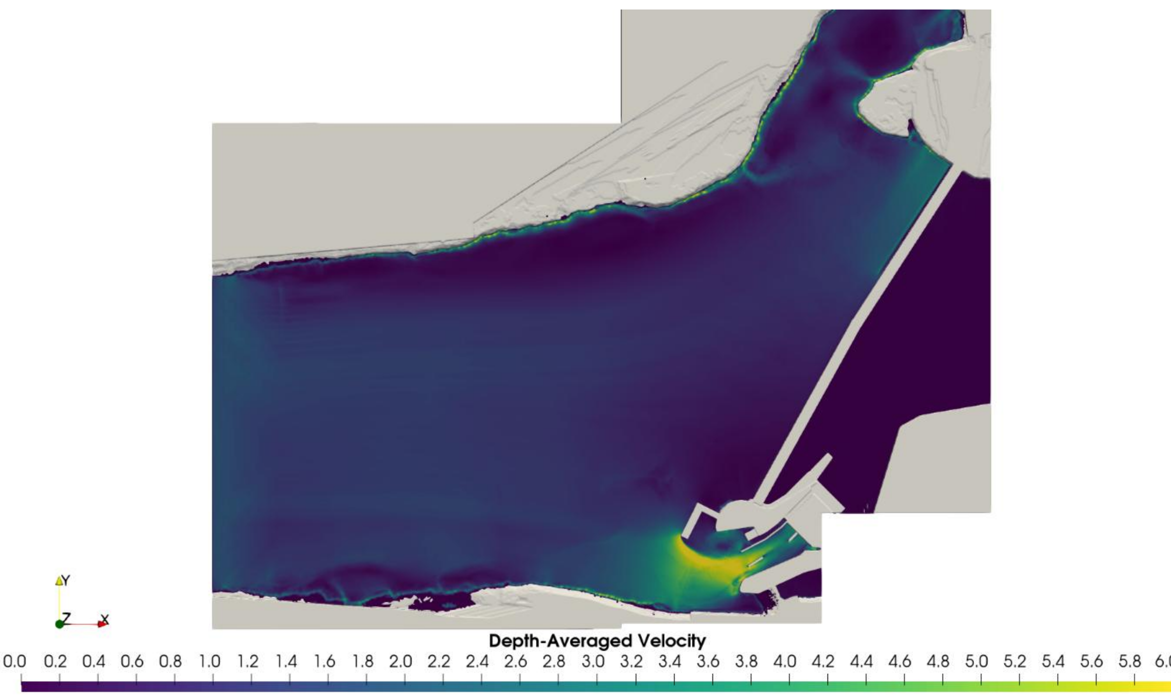

Figure 4-22: Plan View of Depth-Averaged Velocity for Case 5

Results from Case 5 are similar to Case 4. The maximum unit flow rates create velocities in the forebay between 5 and 6 fps. The fish lift entrance velocities remain around 1 fps, and the diffuser creates an elevated velocity area at the surface, peaking around 5 fps. Surface velocities at the unit entrance are generally less than 1.5 fps, while at depth, velocities at the units exceed 6 fps.

A recirculation area is present near the fish lift entrance. The spill creates a uniform flow pattern through the center of the reservoir.

4.2 Powerhouse Tailrace Model

Four tailrace 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 powerhouse units and the bypass chute. Streamlines through the fish lift, diffuser, bypass chute, and unit centerlines are presented in plan and isometric view. Plan view of the full model depth averaged velocity is also presented.

4.2.1 Case 6

Case 6 simulates a low generation discharge of 1,000 cfs and normal fish passage operations.

Streamlines colored by velocity are shown on Figure 4-23. Figure 4-24 shows velocity contours for slices through each AWS centerline and the centerline of the bypass chute. Figure 4-25 presents plan views of velocity contours and vectors in the tailrace, and Figure 4-26 shows the overall model in plan view with depth-averaged velocity contours.

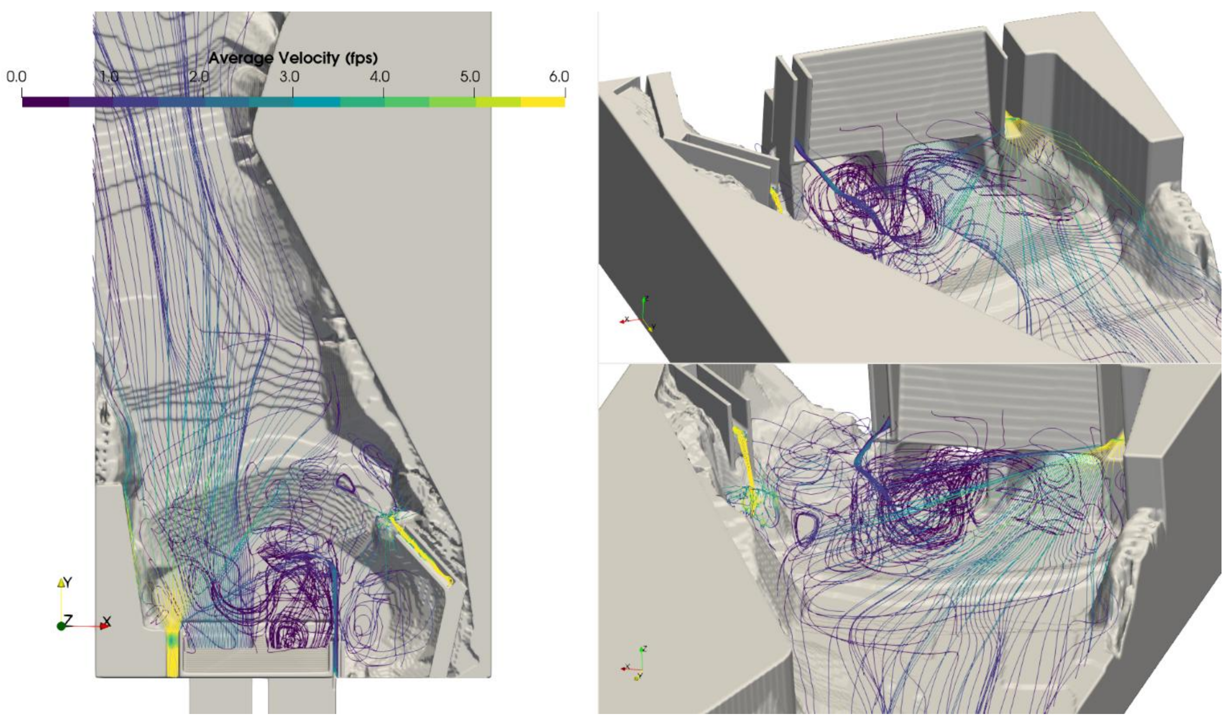

Figure 4-23: Streamlines Colored by Velocity Showing Flow Patterns for Case 6

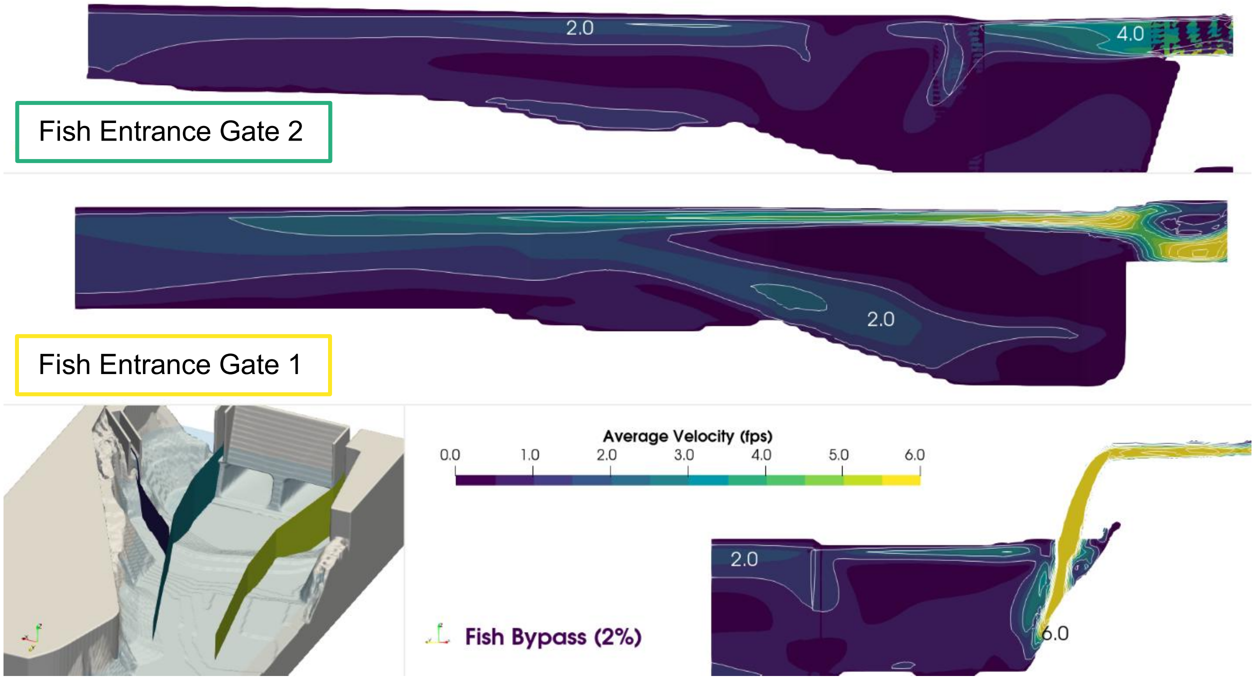

Figure 4-24: AWS & Bypass Chute Centerline Velocity Contours for Case 6

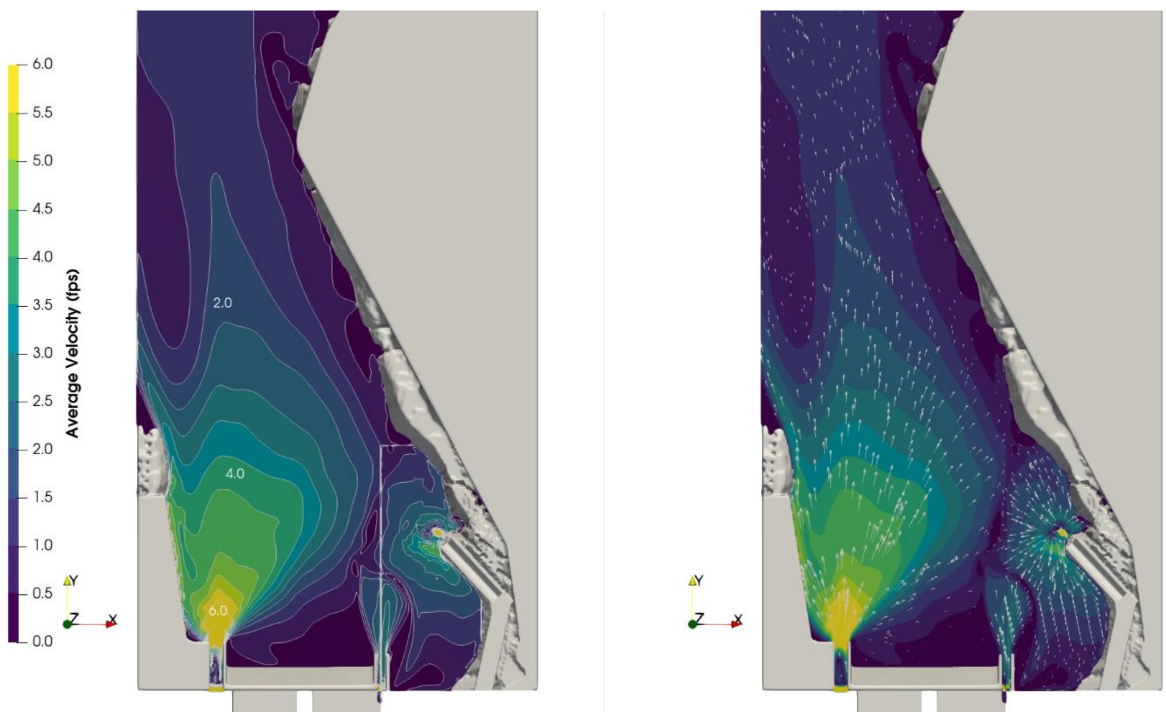

Figure 4-25: Plan Views of Velocity Contours & Vectors in the Tailrace for Case 6

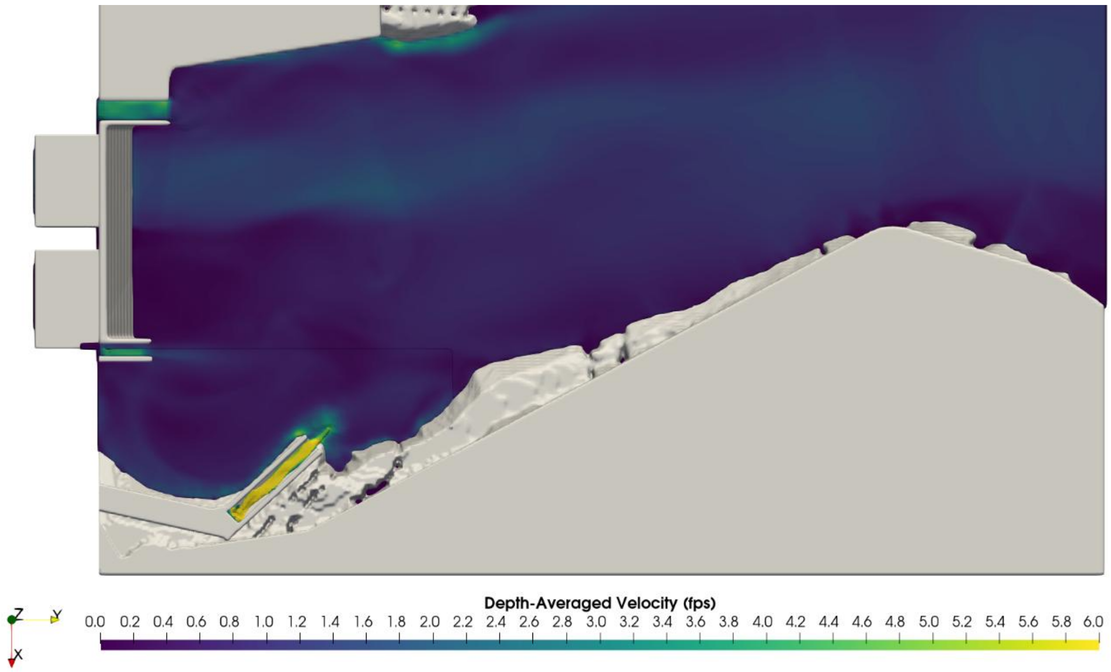

Figure 4-26: Plan View of Depth-Averaged Velocity for Case 6

Velocities exiting the draft tubes are diffused by the tailwater for Case 6. This results in relatively high velocities at the AWS 1 and 2 gates as well as the bypass chute. A large recirculation area forms between the units and the bypass chute. Flow from the bypass chute plunges to the river bed.

Further downstream, there are distinct flowpaths leading to the AWS gates.

4.2.2 Case 7

Case 7 simulates a generation discharge of 3,000 cfs and normal fish passage operations.

Streamlines colored by velocity are shown on Figure 4-27. Figure 4-28 shows velocity contours for slices through each AWS centerline and the centerline of the bypass chute. Figure 4-29 presents plan views of velocity contours and vectors in the tailrace, and Figure 4-30 shows the overall model in plan view with depth-averaged velocity contours.

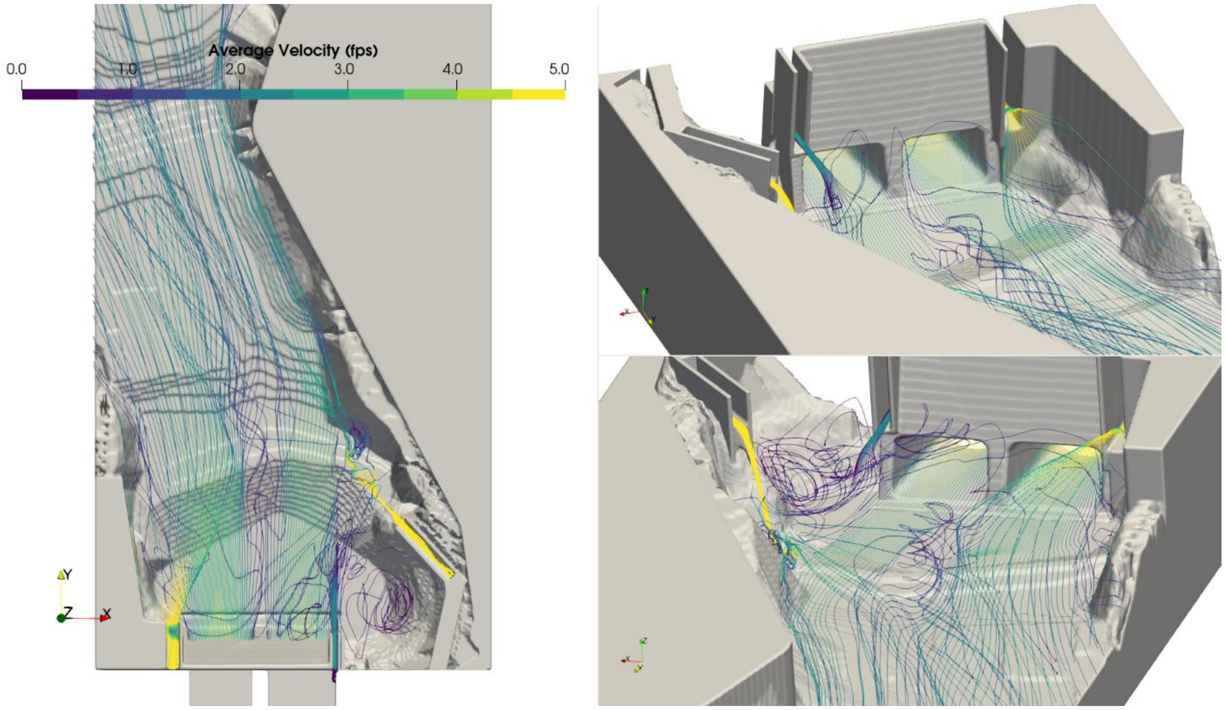

Figure 4-27: Streamlines Colored by Velocity Showing Flow Patterns for Case 7

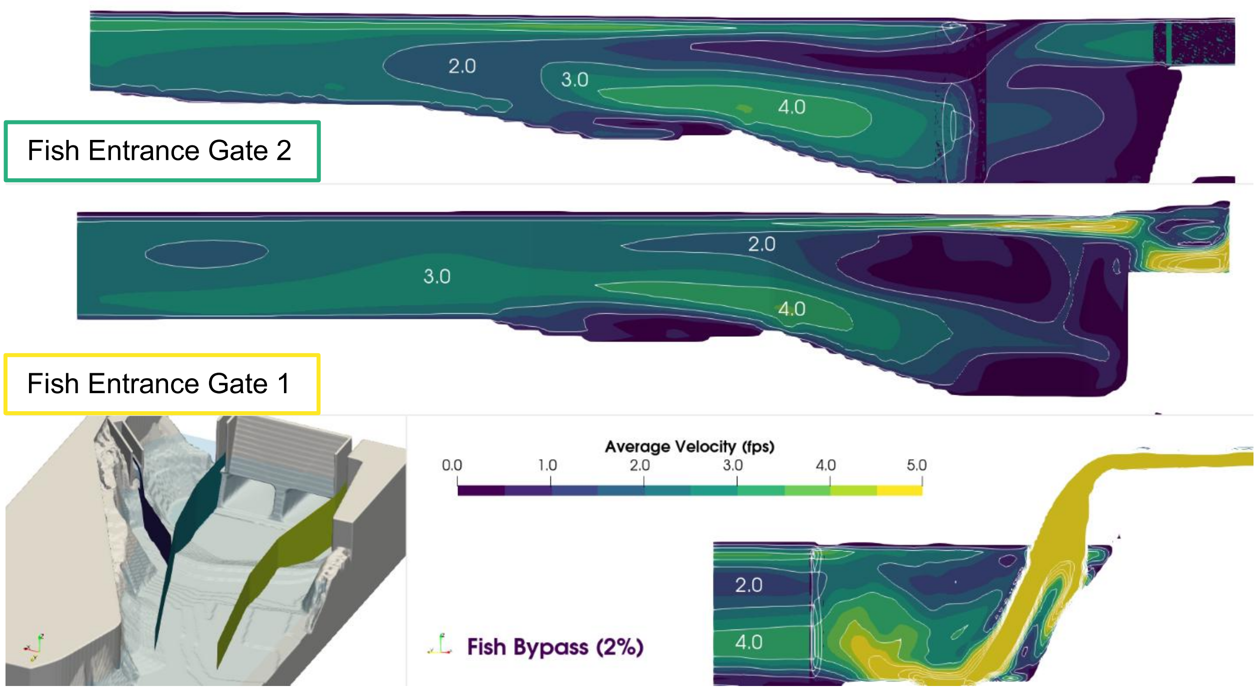

Figure 4-28: AWS & Bypass Chute Centerline Velocity Contours for Case 7

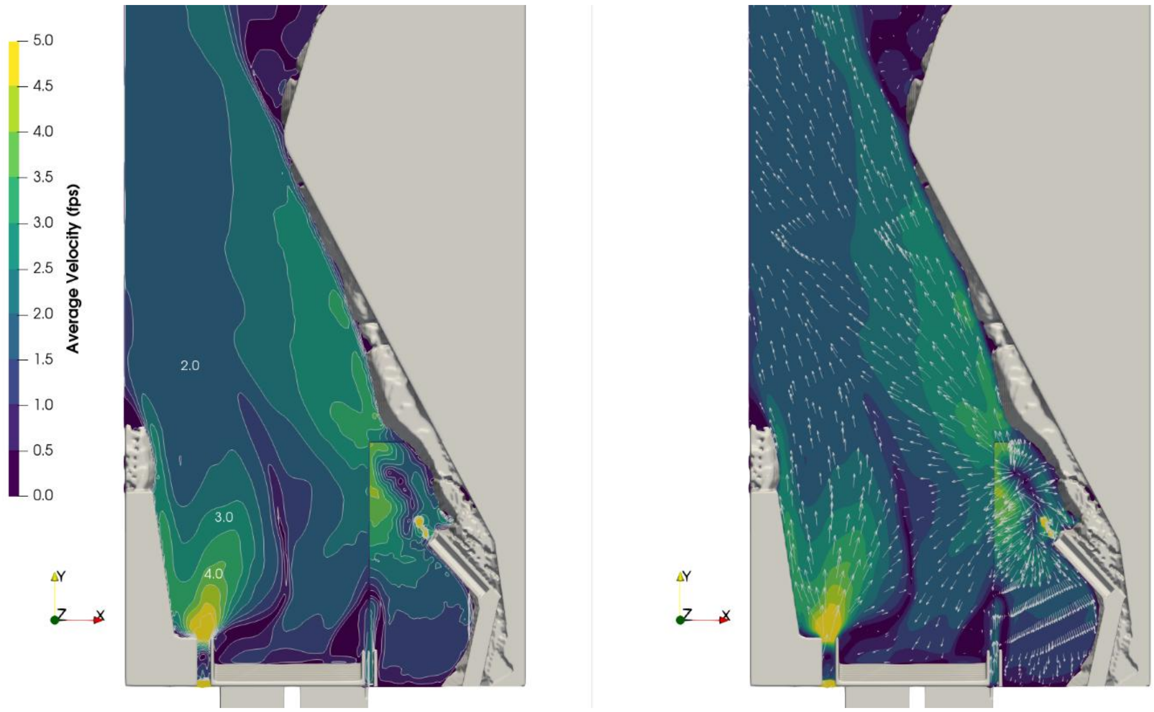

Figure 4-29: Plan Views of Velocity Contours & Vectors in the Tailrace for Case 7

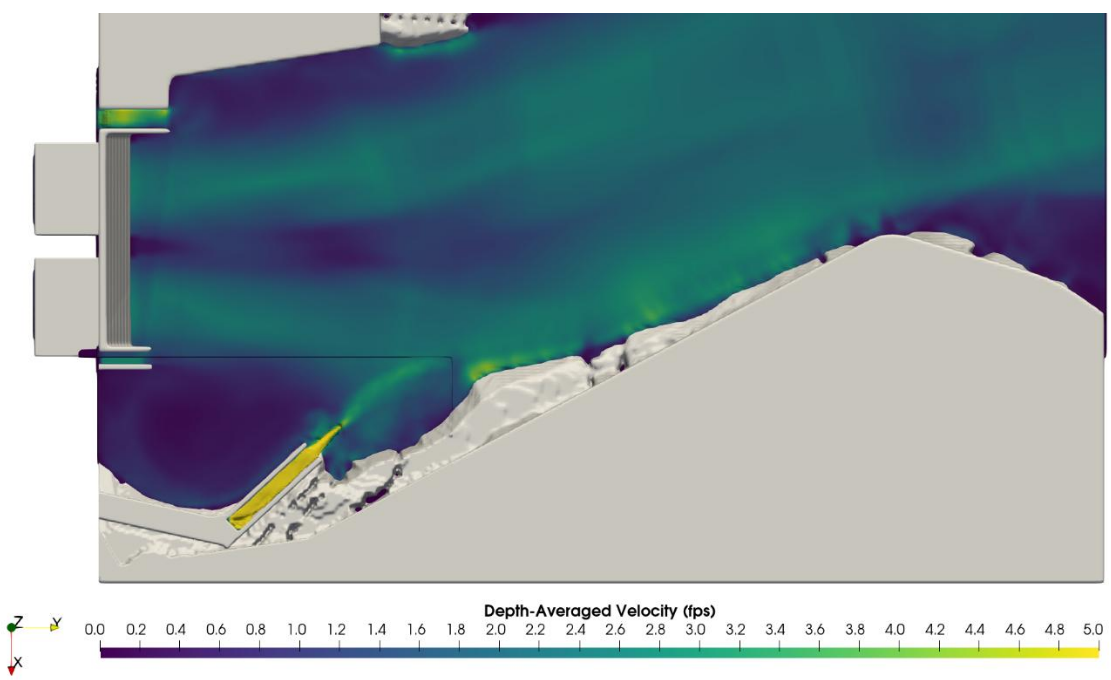

Figure 4-30: Plan View of Depth-Averaged Velocity for Case 7

The generation flow creates a relatively uniform flow pattern at the downstream end of the tailrace, ranging from 2-3 fps. Near the surface, as shown in Figure 4-29, there are distinct flowpaths from the AWS gates and the bypass chute. As flow from the units moves towards the surface and combines with the AWS flow, these flowpaths are washed out. There is a large recirculation area near the surface at the draft tube exit, and another between the units and the bypass chute. Flow from the bypass chute plunges to the riverbed.

4.2.3 Case 8

Case 8 simulates the maximum generation discharge of 8,000 cfs and normal fish passage operations.

Streamlines colored by velocity are shown on Figure 4-31. Figure 4-32 shows velocity contours for slices through each AWS centerline and the centerline of the bypass chute. Figure 4-33 presents plan views of velocity contours and vectors in the tailrace, and Figure 4-34 shows the overall model in plan view with depth-averaged velocity contours.

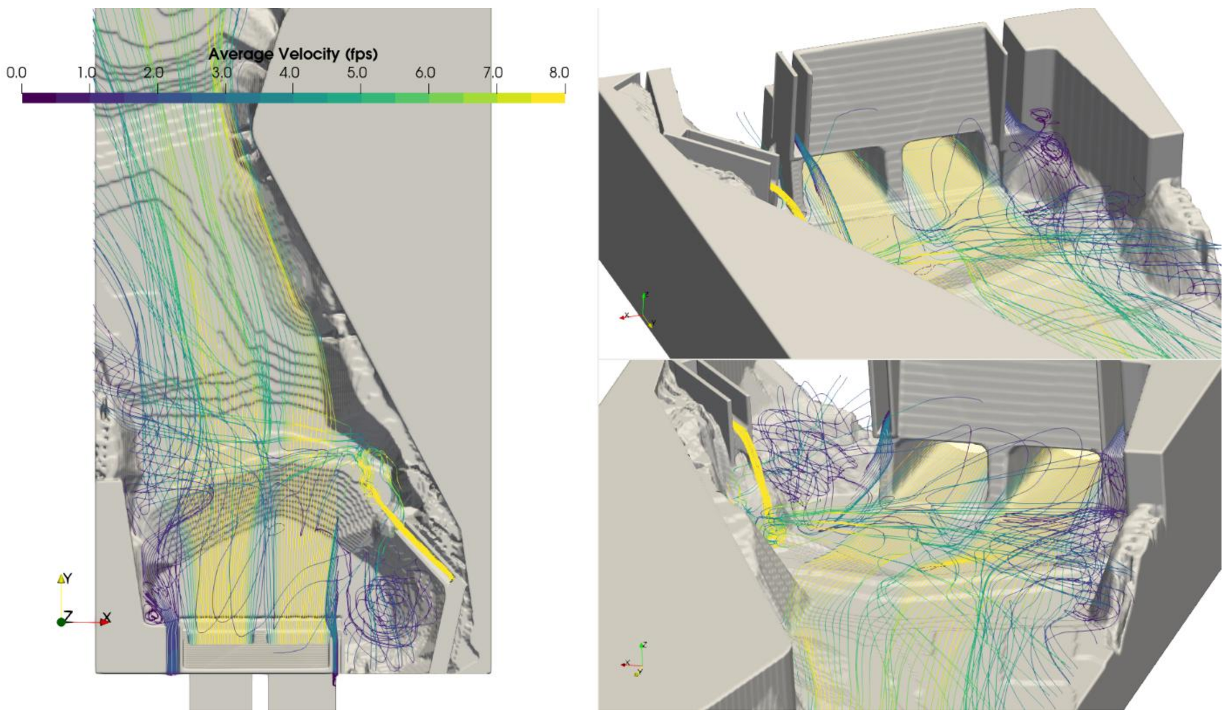

Figure 4-31: Streamlines Colored by Velocity Showing Flow Patterns for Case 8

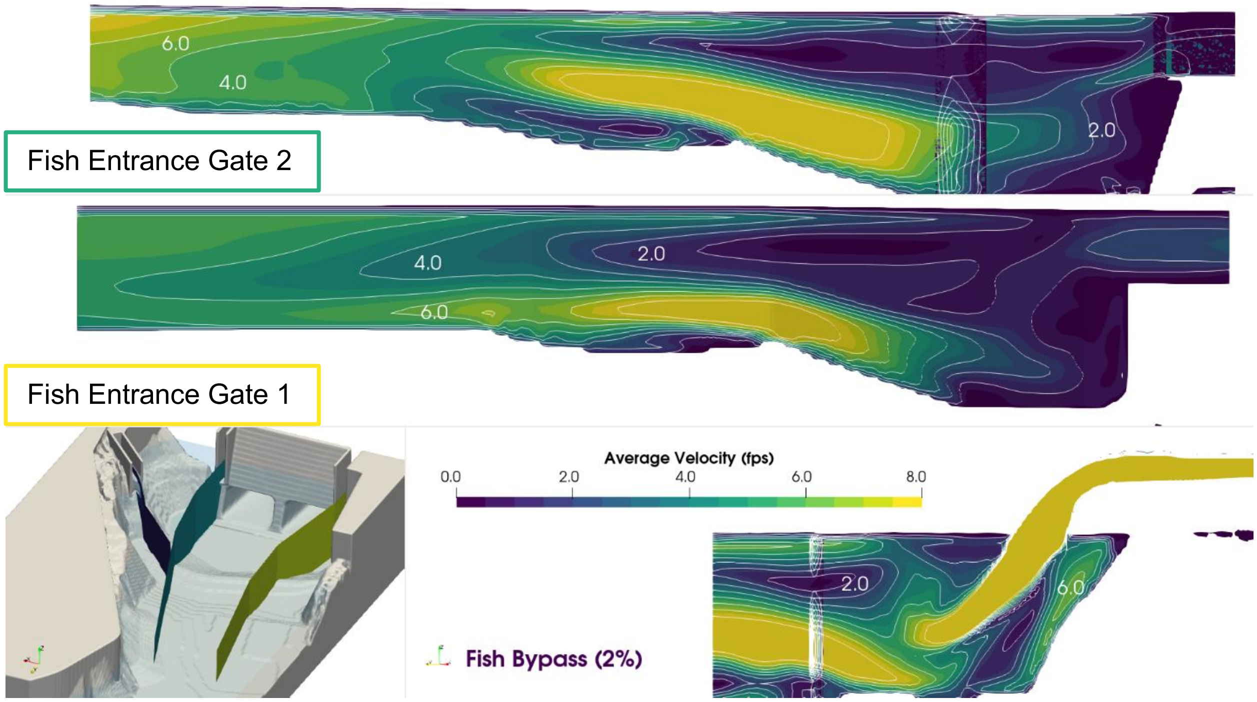

Figure 4-32: AWS & Bypass Chute Centerline Velocity Contours for Case 8

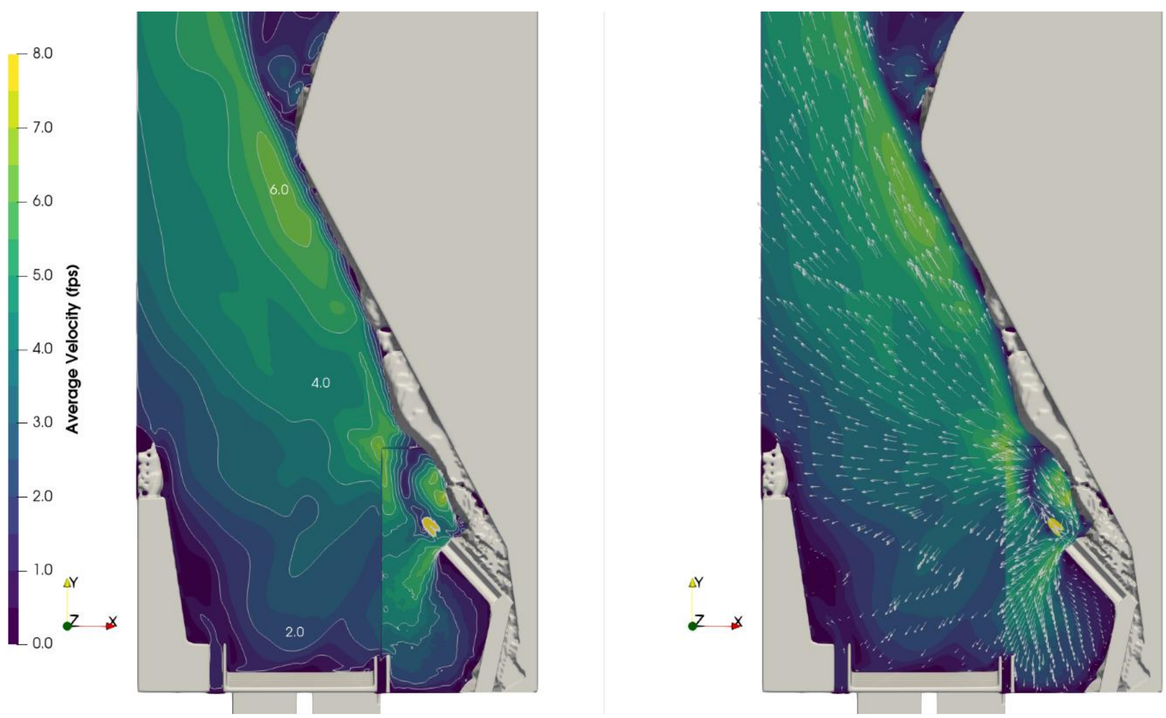

Figure 4-33: Plan Views of Velocity Contours & Vectors in the Tailrace for Case 8

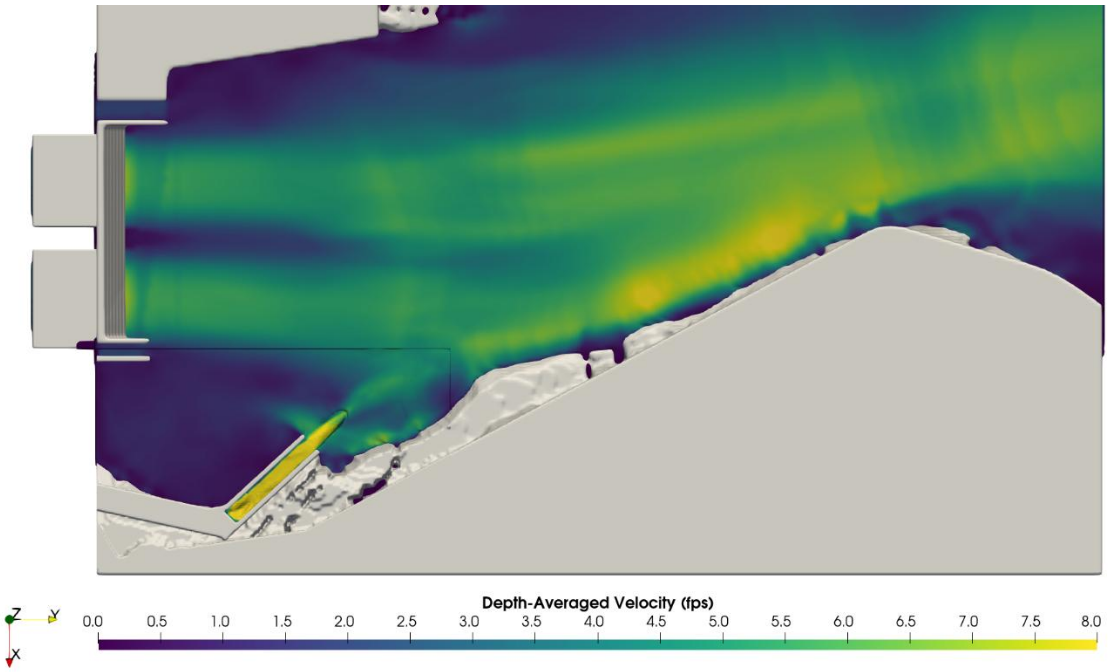

Figure 4-34: Plan View of Depth-Averaged Velocity for Case 8

The maximum generation flow simulated for Case 8 creates a relatively uniform flow pattern at the downstream end of the tailrace. Velocities exceed 7 fps, and the flowpath generally hugs the right bank of the tailrace. Flows from the fish entrance gates are entrained into the large recirculation area that forms above the draft tubes. The distinct local flowpaths and elevated velocities to the fish entrance gates and bypass chute seen in Case 7 are not present in Case 8. The recirculation area between the units and the bypass chute remains, as does the plunging flow from the bypass chute that carries flow to the riverbed.

4.2.4 Case 9

Case 9 simulates the maximum generation discharge of 8,000 cfs, normal fish passage operations, and spill from the center of the dam of 8,000 cfs.

Streamlines colored by velocity are shown on Figure 4-35. Figure 4-36 shows velocity contours for slices through each fish lift entrance gate flow centerline and the centerline of the bypass chute. Figure 4-37 presents plan views of velocity contours and vectors in the tailrace, and Figure 4-38 shows the overall model in plan view with depth-averaged velocity contours.

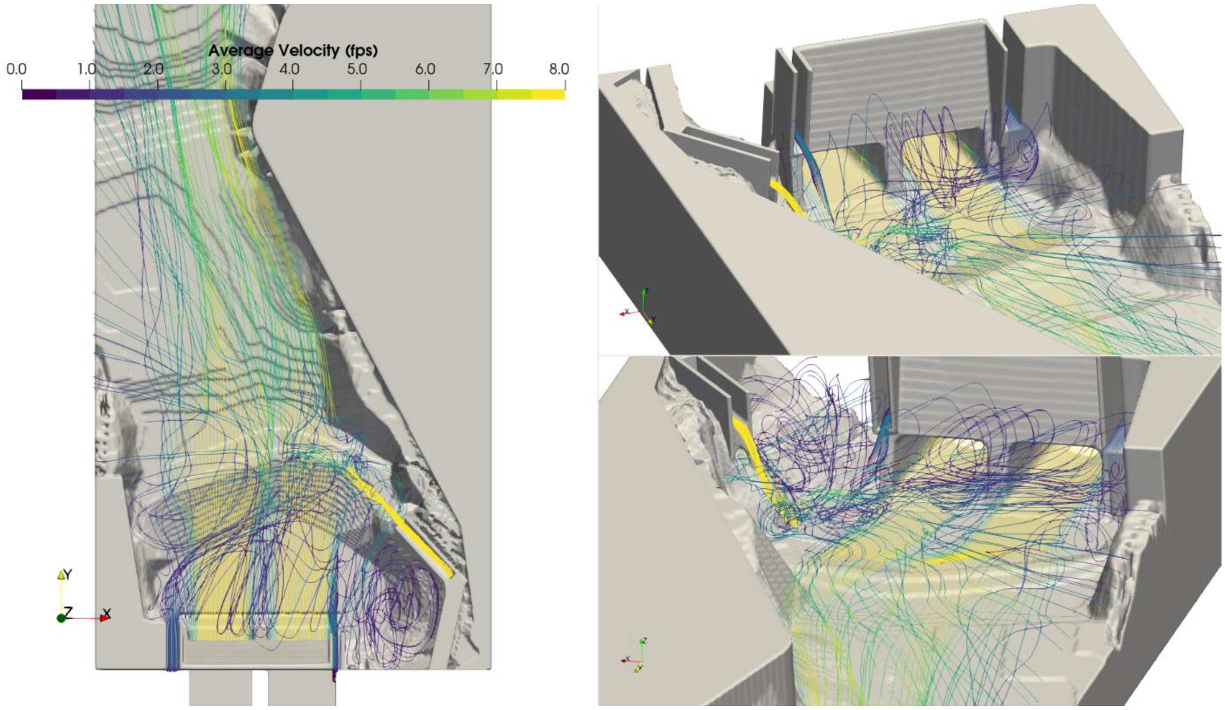

Figure 4-35: Streamlines Colored by Velocity Showing Flow Patterns for Case 9

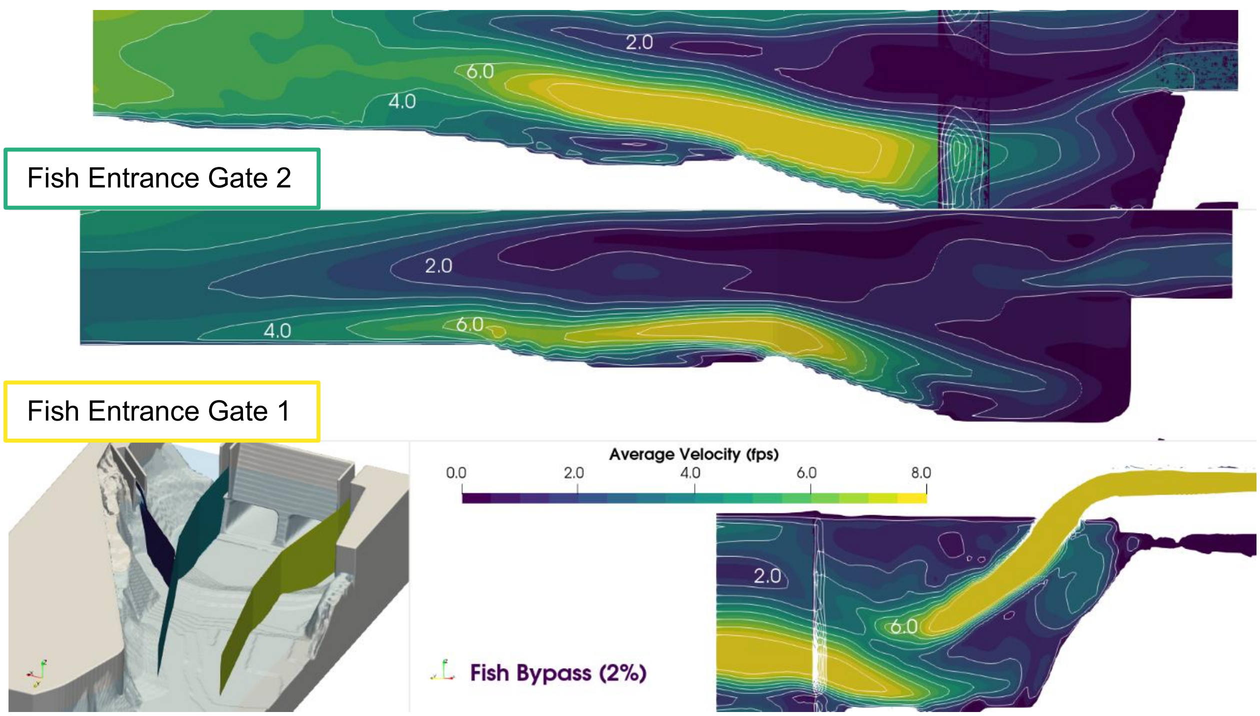

Figure 4-36: AWS & Bypass Chute Centerline Velocity Contours for Case 9

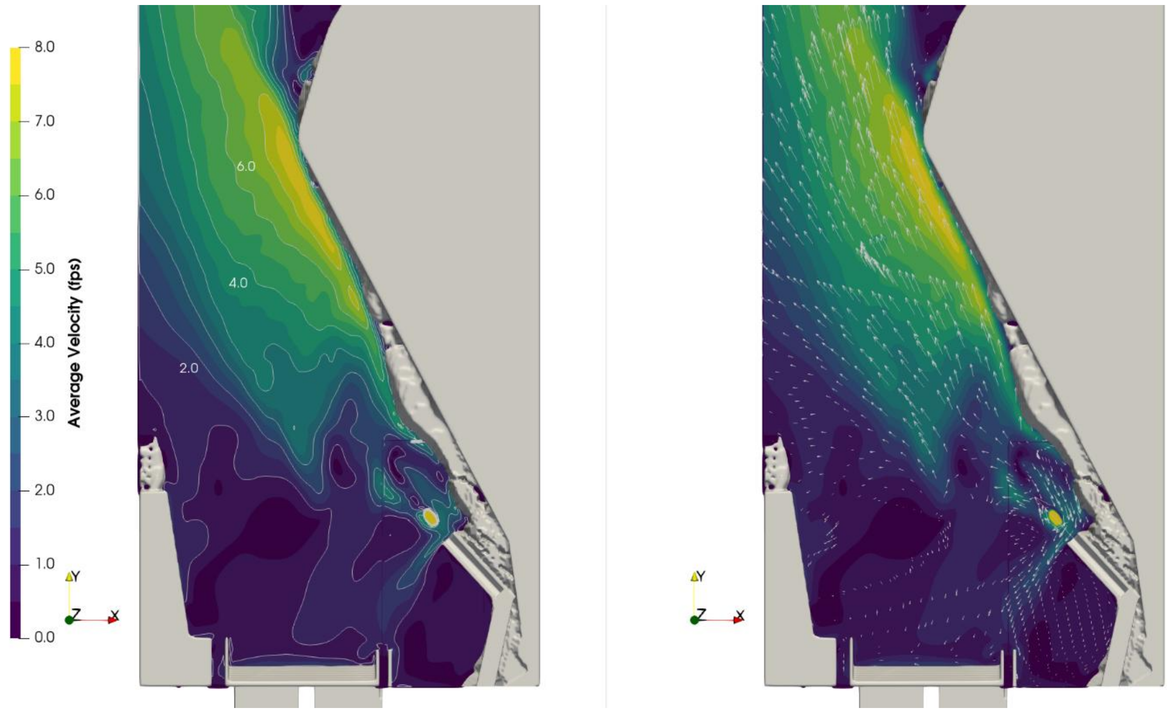

Figure 4-37: Plan Views of Velocity Contours & Vectors in the Tailrace for Case 9

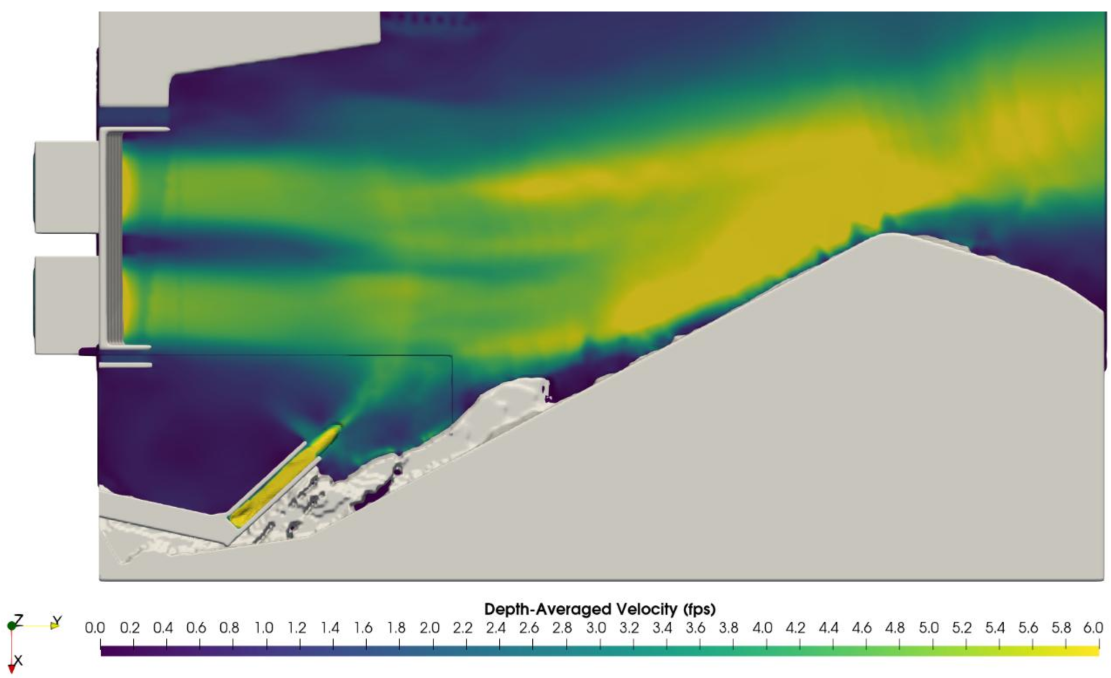

Figure 4-38: Plan View of Depth-Averaged Velocity for Case 9

Case 9 results are very similar to Case 8. The increased tailwater elevation as a result of the spill serves to slightly reduce velocities globally. The maximum generation flow simulated for Case 9 creates a relatively uniform flow pattern at the downstream end of the tailrace. Velocities exceed 7 fps, and the flowpath generally hugs the right bank of the tailrace. Flows from the fish lift entrance gates are entrained into the large recirculation area that forms above the draft tubes. The distinct local flowpaths to the fish lift entrance gates and bypass chute seen in Case 8 are not present in Case 9. The recirculation area between the units and the bypass chute remains, as does the plunging flow from the bypass chute that carries flow to the riverbed.

4.3 3. Essex Fishway Model/Downstream Model

Four downstream 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 powerhouse units. Streamlines through the fish lift, diffuser, bypass chute, and unit centerlines and in the downstream reach are presented in plan, profile, and isometric view. Plan view of the full model depth-averaged velocity is also presented. Note all views for the downstream model are rotated approximately 90 degrees from north.

4.3.1 Case 10

Case 10 analyzes low flow through the units (1,000 cfs), and normal diffuser, eel way, and fish lift operations.

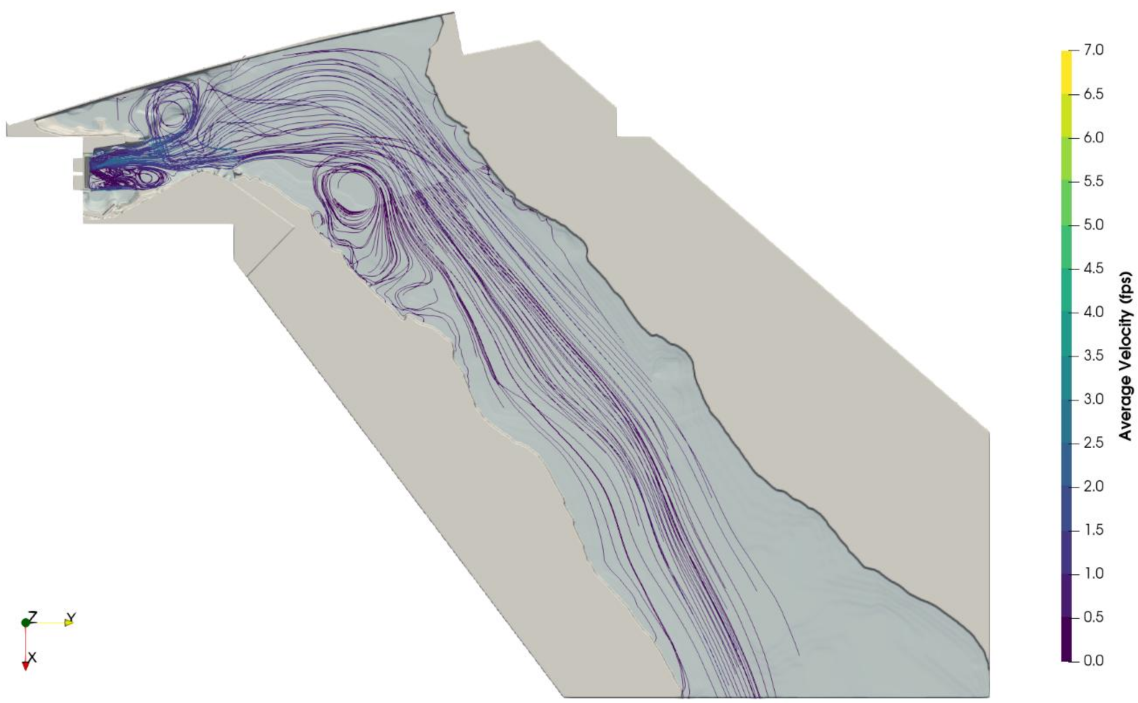

Figure 4-39: Streamlines Colored by Velocity Showing Flow Patterns for Case 10

Figure 4-40: Plan Views of Velocity Contours (0.5 fps) for Case 10

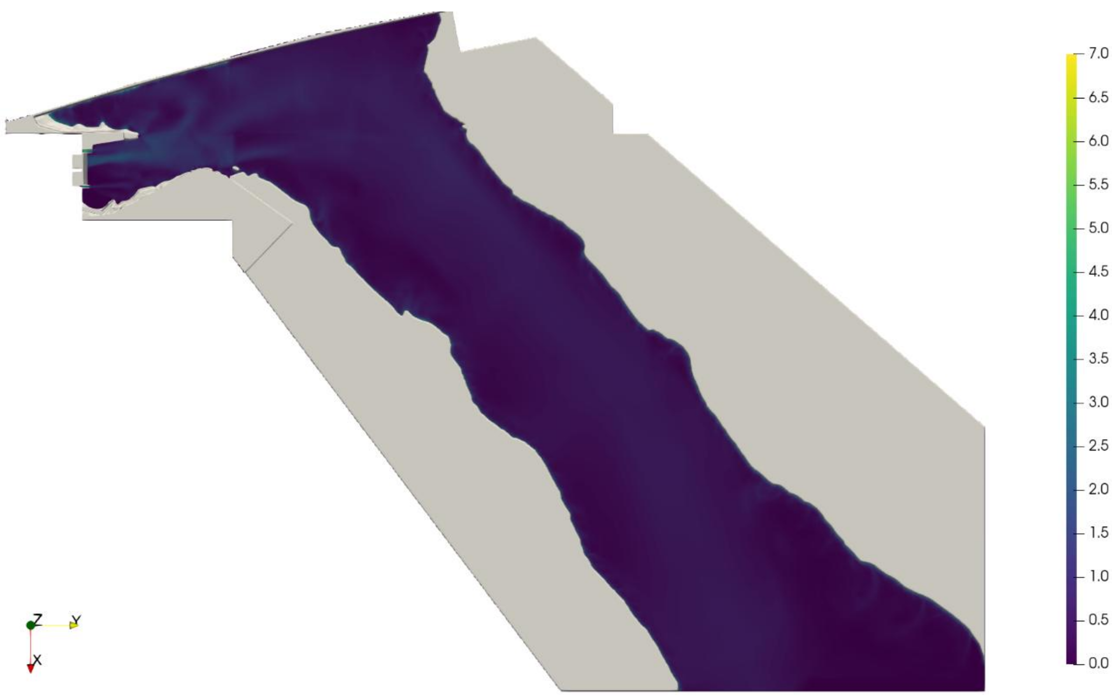

Figure 4-41: Plan View of Depth-Averaged Velocity for Case 10

Low flows from the powerhouse result in velocities generally less than 1fps throughout the model domain. Downstream flowpaths are relatively uniform, with small recirculation areas near the toe of the dam on the north side, and about halfway down on both banks. No defined flowpaths leading to the eel way are present.

4.3.2 Case 11

Case 11 analyzes maximum flow through the units (8,000 cfs) and normal diffuser, eel way, and fish lift operations.

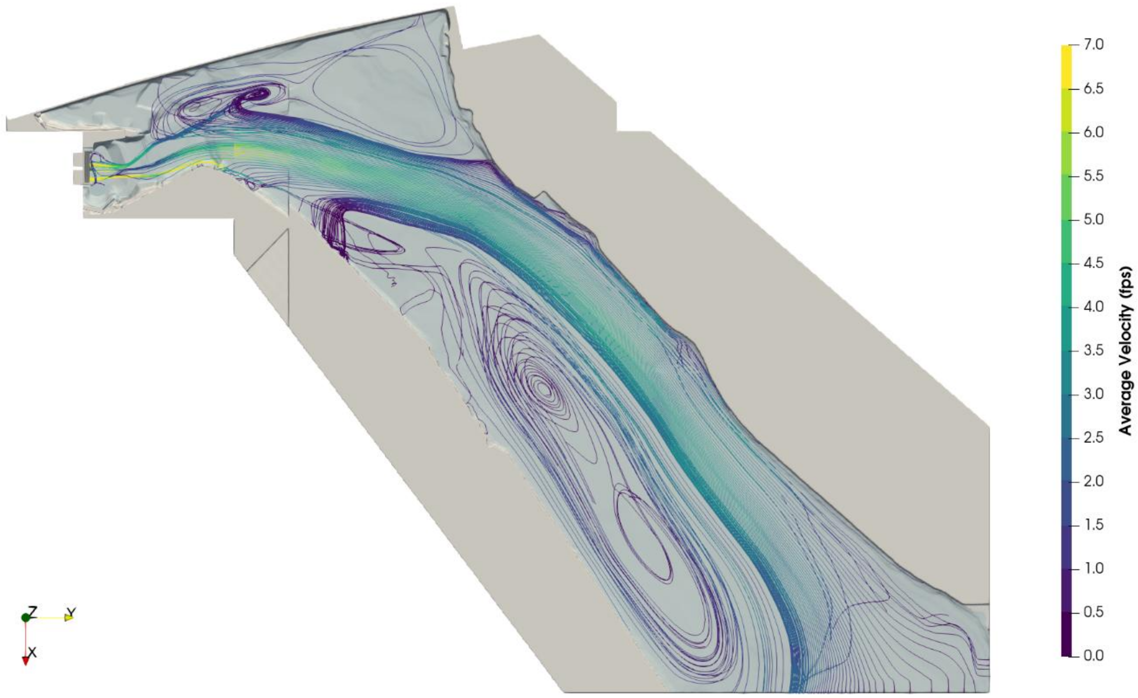

Figure 4-42: Streamlines Colored by Velocity Showing Flow Patterns for Case 11

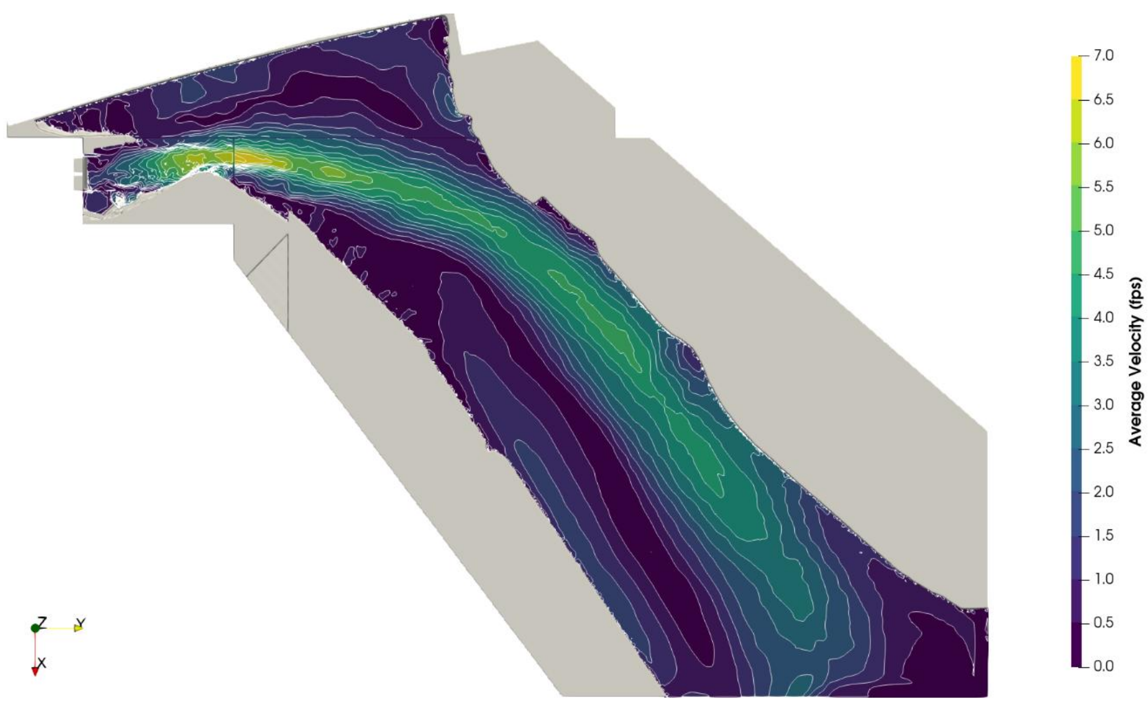

Figure 4-43: Plan Views of Velocity Contours (0.5 fps) for Case 11

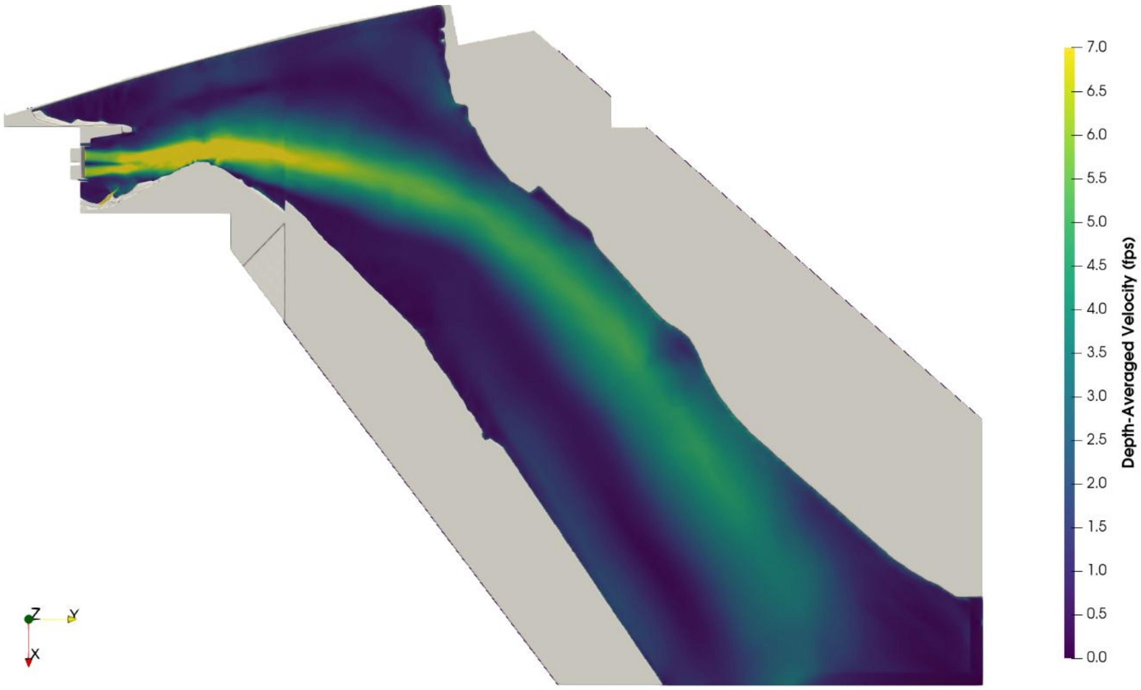

Figure 4-44: Plan View of Depth-Averaged Velocity for Case 11

The maximum unit flow creates a well-defined flowpath extending to the downstream boundary of the model, approximately 1,500 feet below the dam. Flow from the eel way (0.67 cfs) is very small. No defined flowpaths leading to the eel way are present. Streamlines show large recirculation areas along the right descending bank and at the toe of the dam.

4.3.3 Case 12

Case 12 analyzes maximum flow through the units (8,000 cfs); normal diffuser, eel way, and fish lift operations; and a spill of 4,000 cfs through the center of the dam.

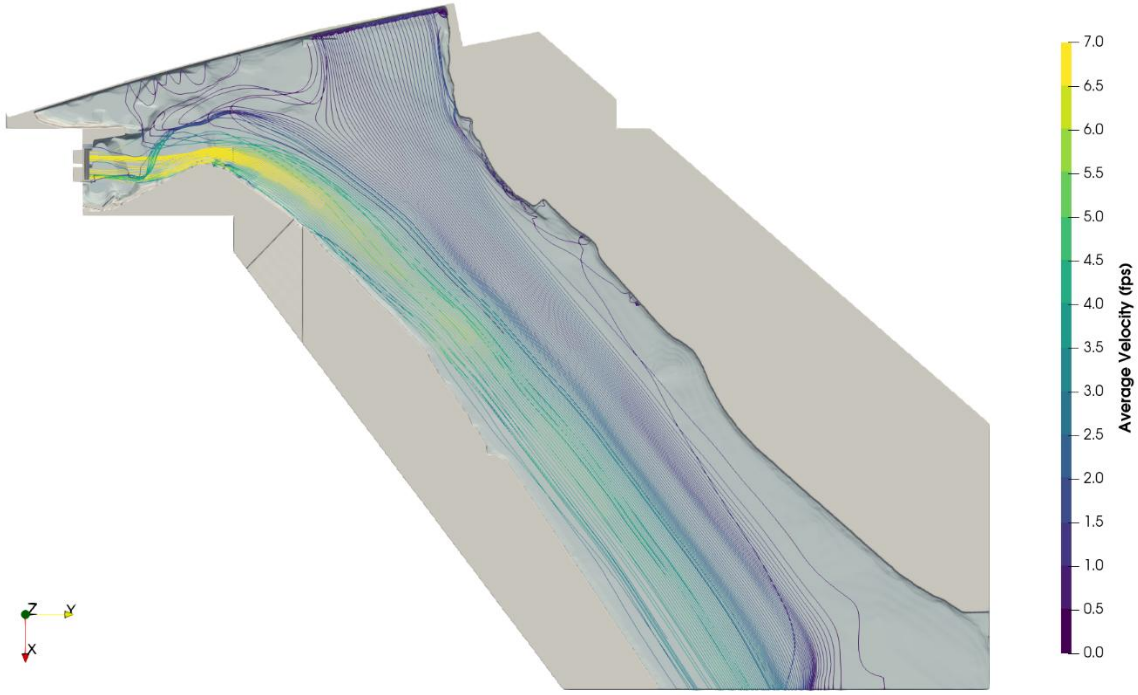

Figure 4-45: Streamlines Colored by Velocity Showing Flow Patterns for Case 12

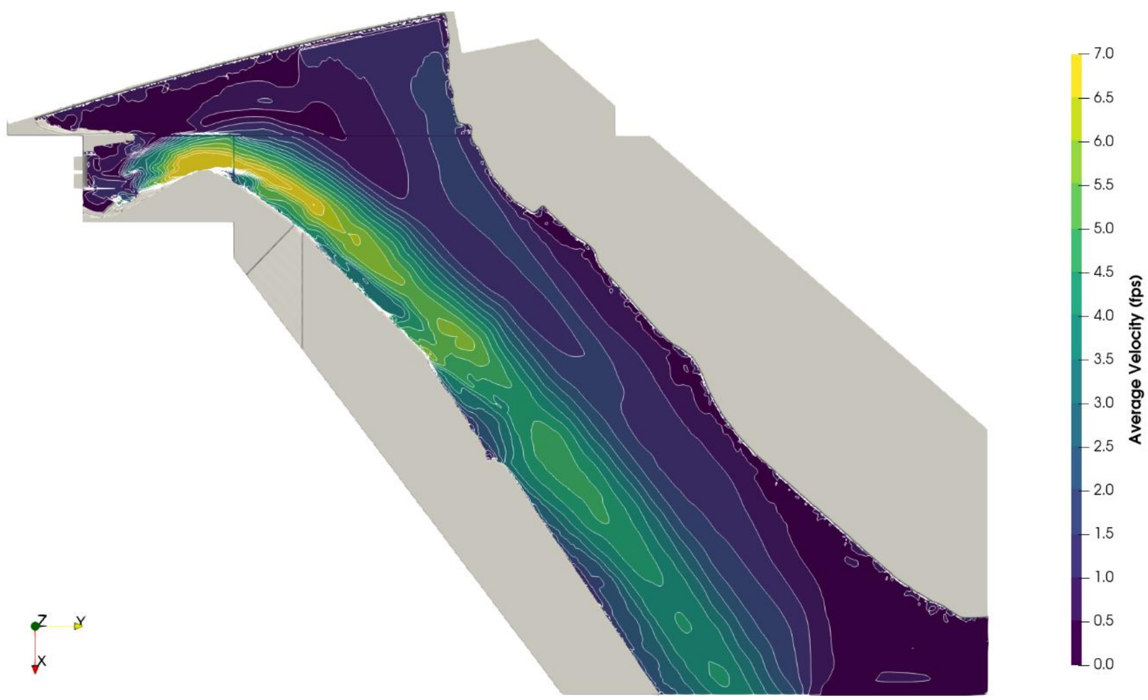

Figure 4-46: Plan Views of Velocity Contours (0.5 fps) for Case 12

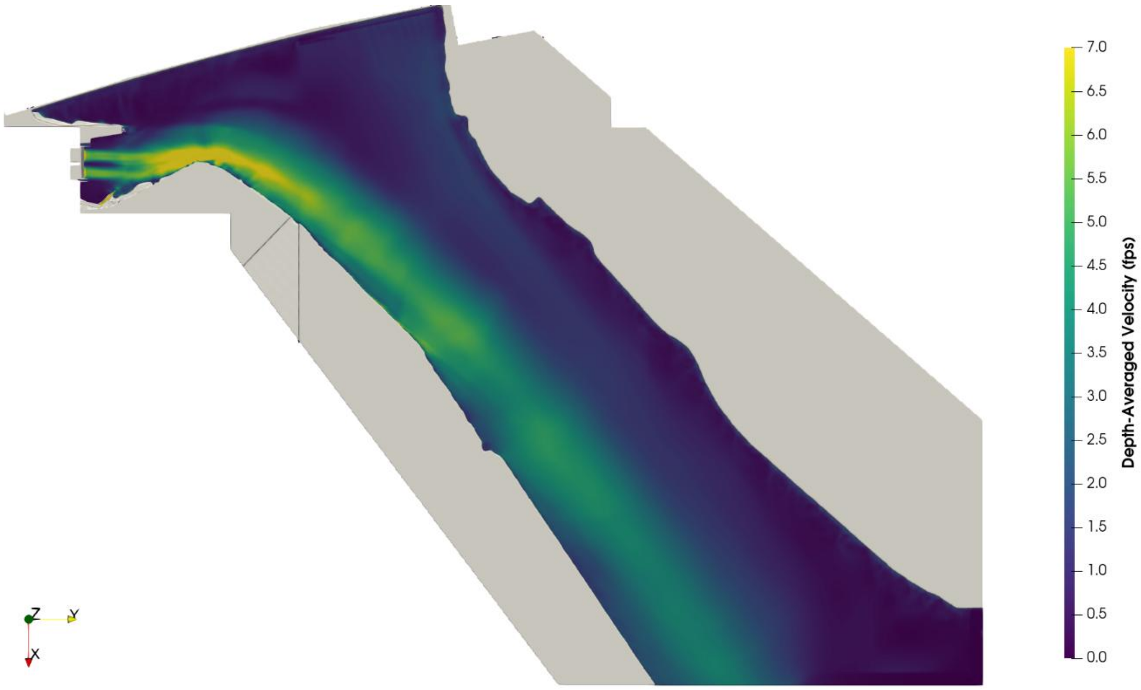

Figure 4-47: Plan View of Depth-Averaged Velocity for Case 12

The well-defined flowpath created by the maximum unit flow is still present in Case 12. The spill and resulting elevated tailwater elevation slightly dampens its effect. Flow from the spillway reaches approximately 2.5 fps before combining with the outflow from the powerhouse and fish passage facilities. Powerhouse and fish passage flows are the same for Case 11 and Case 12. The effect of the spill introduced in Case 12 is to slightly concentrate the flowpath from the powerhouse, pushing it away from the dam. The raised tailwater caused by the spill reduces the velocity of the concentrated flow leaving the powerhouse as it moves downstream.

Flow from the eel way (0.67 cfs) is very small. No defined flowpaths leading to the eel way are present. Streamlines show large recirculation areas along the right descending bank and in the northern corner near the toe of the dam.

4.3.4 Case 13

Case 13 analyzes maximum flow through the units (8,000 cfs); normal diffuser, eel way, and fish lift operations; and a spill of 16,000 cfs through the center of the dam.

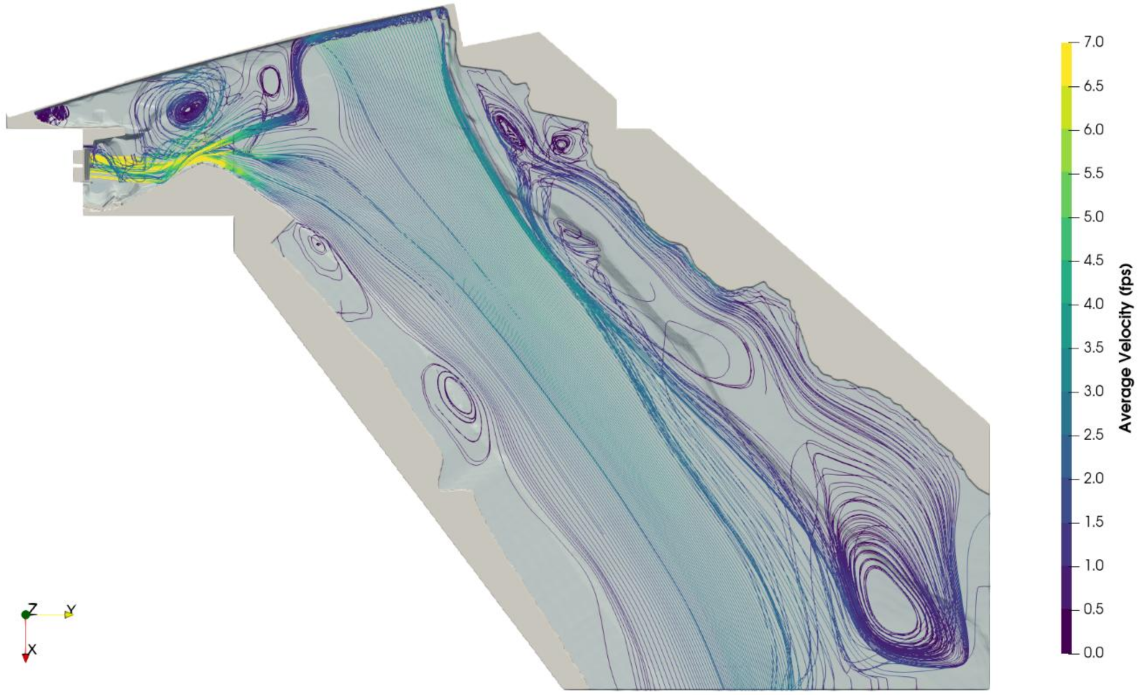

Figure 4-48: Streamlines Colored by Velocity Showing Flow Patterns for Case 13

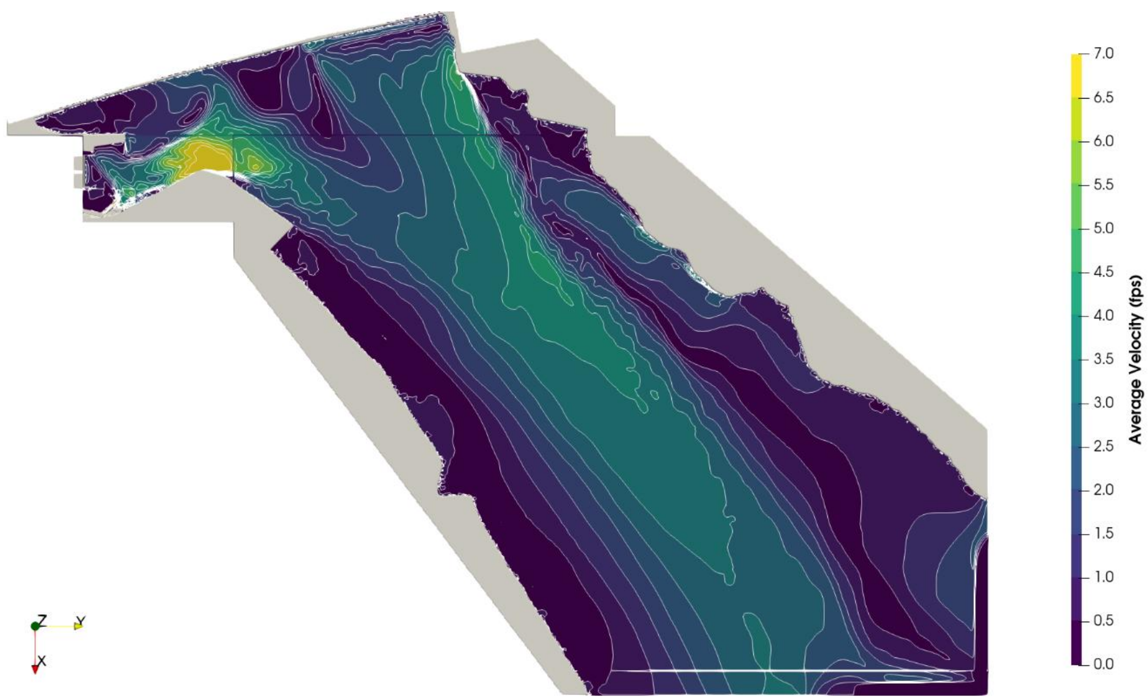

Figure 4-49: Plan Views of Velocity Contours (0.5 fps) for Case 13

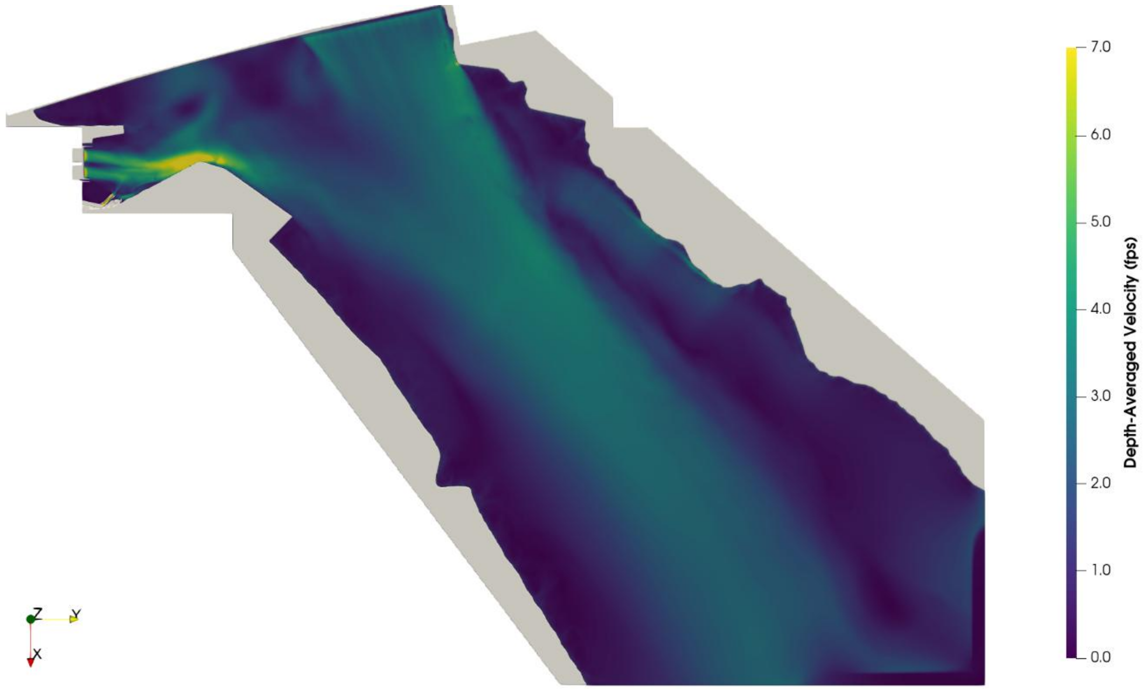

Figure 4-50: Plan View of Depth-Averaged Velocity for Case 13

As with the previous cases, the well-defined flowpath created by the maximum unit flow is still present in Case 13, however, the spill and resulting elevated tailwater elevation dampen its effect. Additionally, the spill and powerhouse/fish passage facility flows combine just downstream of the tailrace, forming a single large area of elevated velocity. Flow from the eel way (0.67 cfs) is very small. No defined flowpaths leading to the eel way are present. Streamlines show large recirculation areas along the right descending bank and along the left descending bank near the dam.

5.0 Conclusions

Three CFD models were developed to help assess the hydraulic conditions near the Essex Dam fish passage facilities, forebay, and tailrace. Multiple simulations were run for each model and the results were graphically presented.

The upstream model shows high velocities in the forebay, while velocities at the fish lift and fish lift exit are relatively consistent across the 4 simulations that modeled fish passage facilities. Generally, flows are pushed to the right side of the forebay, regardless of flowrate through the units, due to the approach conditions to the powerhouse. In all simulations, there was a large recirculation area present near the upstream end of the fish lift, also called the fish exit. Adding spill to the model creates a more uniform flow pattern through the center of the reservoir, increasing velocities throughout, slightly.

The tailwater model shows at low flowrates there are distinct flowpaths to the fish lift entrance gates and the bypass chute. As generation flows increase, these flowpaths are entrained in the large recirculation area above the draft tube exits, and the distinct flowpaths to the fish lift entrance gates are reduced. A large recirculation area is present for all cases between the units and the bypass chute. Spill over the dam creates higher tailwater and serves to slightly reduce velocities in the tailrace when comparing cases with similar powerhouse and fish passage operations.

The downstream model shows at low flowrates there are relatively uniform flowpaths to the tailrace area, and overall low velocities. As generation flows increase, the flowpath to the tailrace and fish passage facilities become well-defined, with velocities around 5 fps. Adding the spill to the maximum unit discharge elevates the tailwater, the flow paths in the downstream reach are maintained with slightly reduced velocities. As spill increases to 16,000 cfs, the flowpath definition is further reduced and the spill combines with the unit discharge to form a large area of high velocity just downstream of the tailrace.

A large recirculation area is present for all cases at the toe of the dam. For the maximum discharge cases, a recirculation area is present on the right descending bank. This creates an arced shape in the main downstream flowpath.

6.0 Variances from the Approved Study Plan

The 3D CFD Study is underway but to date there have been no variances from the approved study plan. As noted in the Commission's Study Plan Determination, the 3D CFD Study will include a 2D model of the downstream reach (i.e., the Merrimack River from the downstream face of the Project's dam to the downstream side of the Union Street Bridge). Included within this report is a 3D model of the reach below the dam approximately 1,500 feet. Essex is processing the bathymetry data from the side-scan sonar data collected in the fall of 2025 for the Sturgeon Distribution and Project Interaction Study. With the results of the 3D downstream model available with this report (Section 4.3.3), Essex anticipates further discussion with relevant fisheries agencies on the model conditions and scenarios to be simulated for the 2D portion of the study.

7.0 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 Mathematicians (pp. 1395-1399). Berkley: Princeton University.