

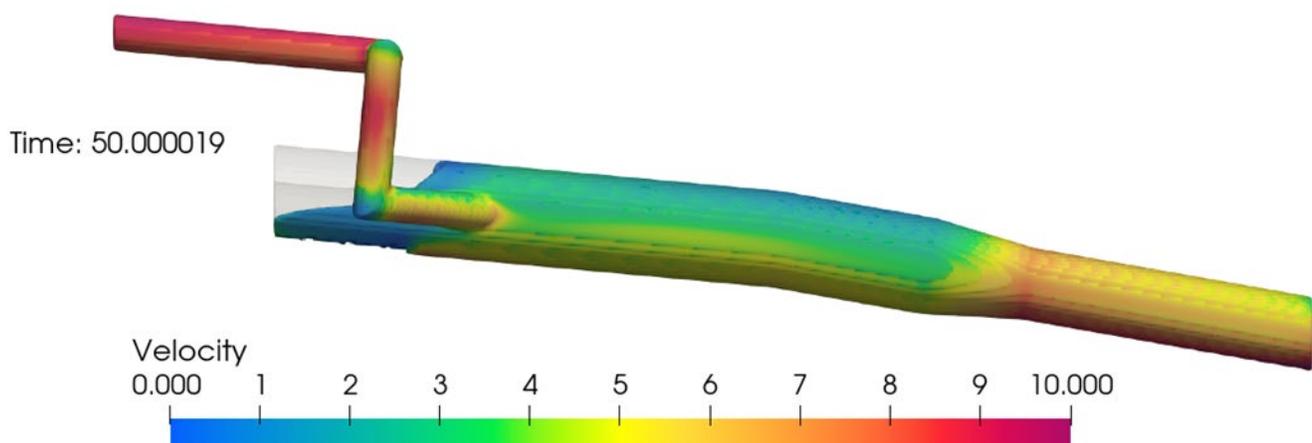


— BUREAU OF —
RECLAMATION

Hydraulic Laboratory Report HL-2025-02

Hydraulic Improvements for the Antioch Fish Release Site – Computational Fluid Dynamics Model Study

Tracy Fish Collection Facility, California
California-Great Basin Region



Mission Statements

The U.S. Department of the Interior protects and manages the Nation's natural resources and cultural heritage; provides scientific and other information about those resources; honors its trust responsibilities or special commitments to American Indians, Alaska Natives, Native Hawaiians, and affiliated Island Communities.

The mission of the Bureau of Reclamation is to manage, develop, and protect water and related resources in an environmentally and economically sound manner in the interest of the American public.

Disclaimer

The information provided in this report is believed to be appropriate and accurate for the specific purposes described, but users bear all responsibility for exercising sound engineering judgment in its application, especially to situations different from those studied. References for commercial products do not imply endorsement by the Bureau of Reclamation and may not be used for advertising or promotional purposes.

Hydraulic Laboratory Reports

The Hydraulic Laboratory Report series is produced by the Bureau of Reclamation's Hydraulic Investigations and Laboratory Services Group (Mail Code 86-68560), P.O. Box 25007, Denver, Colorado 80225-0007. This report was made available online at <https://www.usbr.gov/tsc/>.

Cover Image – A computational fluid dynamics (CFD) model of the inlet conditions of the Antioch Fish Release Site during a truck release flow (Bureau of Reclamation).

Hydraulic Laboratory Report HL-2025-02

Hydraulic Improvements for the Antioch Fish Release Site – Computational Fluid Dynamics Model Study

**Tracy Fish Collection Facility, California
California-Great Basin Region**

Prepared by:

**Bureau of Reclamation
Technical Service Center
Denver, Colorado**

Hydraulic Laboratory Report HL-2025-02

Hydraulic Improvements for the Antioch Fish Release Site – Computational Fluid Dynamics Model Study

Tracy Fish Collection Facility, California California-Great Basin Region

Prepared by: Jacob Carter-Gibb, P.E.
Hydraulic Engineer, Hydraulic Investigations and Laboratory Services Group, 86-68560

Checked by: Kurt Smithgall, P.E.
Hydraulic Engineer, Hydraulic Investigations and Laboratory Services Group, 86-68560

Peer Review Certification

This report has been reviewed and is believed to be in accordance with the service agreement and standards of the profession.

Technical Approval: Connie Svoboda, P.E.
Manager, Hydraulic Investigations and Laboratory Services Group, 86-68560

Peer reviewed by: Josh Mortensen, P.E.
Hydraulic Engineer, Hydraulic Investigations and Laboratory Services Group, 86-68560

Acronyms and Abbreviations

3D	three-dimensional
CFD	computational fluid dynamics
ft	foot/feet
ft ³ /s	cubic feet per second
RANS	Reynolds-Averaged Navier-Stokes
Reclamation	Bureau of Reclamation
VOF	Volume of Fluid Method

Symbols

°	degrees
=	equals

Contents

	Page
Abstract.....	AB-1
Executive Summary	ES-1
Background.....	1
Site Visit.....	4
Analytical Assessment.....	5
Computational Fluid Dynamics (CFD) Model	7
Model Extents	7
Boundary Conditions	8
Results.....	9
Baseline Simulations.....	9
Operational Simulations.....	11
Modification Simulations.....	12
Additional CFD Simulations of Final Proposed Geometry	13
Purpose.....	13
Numerical Model – Open FOAM.....	14
Results.....	15
Summary	16
Conclusions and Recommendations	16
References.....	19

Tables

1.—Summary of difference between physical model testing and constructed structure.....	3
2.—Simulation summary	9

Figures

1.—Top, Antioch Fish Release Site relative to the Tracy Fish Collection Facility. Bottom, the release site relative to the San Joaquin River and State Highway 160.....	2
2.—The auxiliary flow manifold and upstream portion of the release pipe are shown in the as-built drawings.	3
3.—Example of a small leakage flow during release operations with the flex trunk inserted into the pipe.	4
4.—Antioch Release Pipe operating conditions compared to blowback guidance.	6
5.—Model domain from Flow-3D [®]	7
6.—Red arrows indicate model boundaries where the flow could enter the model domain. Blue arrows indicate boundaries where flow could exit the domain.	8
7.—Truck discharge over time (truck hydrograph).	9
8.—Baseline 1 simulation.....	10
9.—Baseline 2 simulation.....	10
10.—Baseline 2 simulation CFD flow field.	11

11.—Operational simulations with the baseline shown in dark blue, the manifold flow reduced to 1 ft ³ /s shown in yellow and the manifold flowrate reduced to 0.5 ft ³ /s shown in green.....	12
12.—Modification simulations showed the greatest change in the spillage flow.....	13
13.—Summary of tested geometries.....	14
14.—Computational domain for the OpenFOAM simulations.....	14
15.—The inner hydraulics of the pipe with the peak wave height shown by a red line that matches in invert elevation of the final proposed geometry.....	15
16.—Air demand of the inflow pipe and flow rate of the truck plus the auxiliary flow manifold.....	16

Abstract

The Antioch Fish Release Site is a federally owned site that is used by both the California Department of Water Resources and the Bureau of Reclamation. The site is designed to release fish back to the San Joaquin and Sacramento Rivers via haul truck from the Old River upstream of the California Aqueduct and the Delta-Mendota Canal intakes. Construction on the Antioch Site was finished in 2021 with some differences in slope and invert elevation compared to the initial design. This led to spillage from the pipe during normal releases. To address this, a computational fluid dynamics (CFD) model was created to evaluate different operational and physical modifications to the release pipe to minimize spillage. Initial testing included extending the upstream intake 3 feet at a 5-degree incline or extending the intake 2 feet at a 10-degree incline with different combinations of reduction in the auxiliary flow rate and flex pipe insertion. The 2-foot intake extension at 10 degrees with a reduced auxiliary flow rate of 1 cubic foot per second showed the greatest improvement of any changes tested. Follow up testing in a full two-phase simulation showed that a 3-foot extension at a 15-degree incline produced no spillage. This results in the inlet of the pipe being raised above the level of the wave that is formed in the area where the manifold pipes converge.

Keywords

Fish Release, computational fluid dynamics, CFD, Tracy Fish Collection Facility

Executive Summary

Following the construction of the Antioch Fish Release Site in 2021, field observations identified overflow at the pipe inlet during fish release operations. The as-built structure deviated from what was tested in a physical model, with a lower pipe inlet elevation and shallower slope. This resulted in hydraulic conditions that allowed water to exit the pipe inlet during combined auxiliary and truck flows. Computational fluid dynamics (CFD) modeling and field testing confirmed that the primary driver of this overflow was the inlet's submergence relative to the auxiliary flow manifold, with additional contributions from air entrainment and flex trunk interference. Simulations demonstrated that operational modifications, such as reducing auxiliary flow rates, and structural changes, including extending the inlet pipe, reduced overflow. A 3-foot extension at a 15-degree slope raised the pipe invert above the turbulence zone and eliminated spillage under simulated conditions. Implementation of this modification, along with operational guidance to limit auxiliary flow and flex trunk insertion depth, is recommended to address overflow concerns and improve release performance.

Background

The Antioch Fish Release Site is located near Antioch Bridge off Highway 160 in Antioch, CA. The federally owned site is used by both the John E. Skinner Delta Fish Protective Facility (operated by the California Department of Water Resources) and the Tracy Fish Collection Facility (operated by the Bureau of Reclamation [Reclamation]). The goal of this site is to meet the current Biological Opinion needs for fish releases back to the San Joaquin and Sacramento Rivers. Fish are collected from Old River upstream of the California Aqueduct and the Delta-Mendota Canal intake channel upstream of the C.W. Bill Jones pumping plant.

The fish are put into a haul truck and driven to various release sites along the San Joaquin River Delta beyond the influence of the pumping systems. Once a haul truck arrives at a release site, such as the Antioch Site, it backs up to the inlet of the pipe and connects to the pipe via a slip fitting called a flex trunk. The release pipe is then cleared using a flushing flow of 3 cubic feet per second (ft^3/s) to remove any debris that could injure the fish during the release. This flows through an auxiliary flow manifold that ties into the pipe a couple of feet down from the inlet of the pipe. The truck operator then opens a valve at the back of the truck to allow the fish and storage water to drain through the pipe, with the truck flow rate reaching a peak of $5 \text{ ft}^3/\text{s}$. During the truck release a lower auxiliary flow is also pumped through the pipe to ensure that none of the fish will get impinged on the wall of the pipe as the flow is ramping up and down. After the truck is empty, a second flushing flow of $3 \text{ ft}^3/\text{s}$ runs through the pipe.

Hydraulic Laboratory Report No. HL-2025-02
Hydraulic Improvements for the Antioch Fish Release Site
Computational Fluid Dynamics Model Study



Figure 1.—Top, Antioch Fish Release Site relative to the Tracy Fish Collection Facility. Bottom, the release site relative to the San Joaquin River and State Highway 160.

In 2020, a 1:4 Froude-scale physical hydraulic model was constructed to model an updated design proposed for the release pipes. The primary goal of the model was to test debris passage and analyze the pre- and post-release flushes for efficacy. Debris tested included submerged aquatic vegetation as well as buoyant debris in very large quantities. It was found that the pre-release flushing flow was more than sufficient to pass any debris that would be found in the pipe, and this design would allow for safe fish passage from the haul truck into the river (Carter-Gibb 2020).

Construction on the Antioch Site was finished in 2021. The final as-built site was generally constructed per the design testing in the physical model but there were some key differences near the upstream end of the pipe. The upstream inlet was constructed 1 foot lower than the design elevation recommended from the physical model study (figure 2). This resulted in a very slight difference in slope between the physical model and the prototype structure, 2.13 degrees (°) versus 1.67° in the lower section and 9.6° versus 10° in the upper section. Additionally, the physical model had a longer section of pipe upstream of the confluence with the auxiliary manifold jets to better accommodate loading debris into the pipe for testing in the physical model study. This was for the model only and was not recommended for construction. The differences in slope and inlet elevation that led to the conditions for unsafe fish release are the primary focus of this report.

Table 1.—Summary of difference between physical model testing and constructed structure

	Model	As-Built
Upper slope	9.6°	10°
Lower slope	2.13°	1.667°
Invert elevation	11.9 feet (ft)	11.1 ft

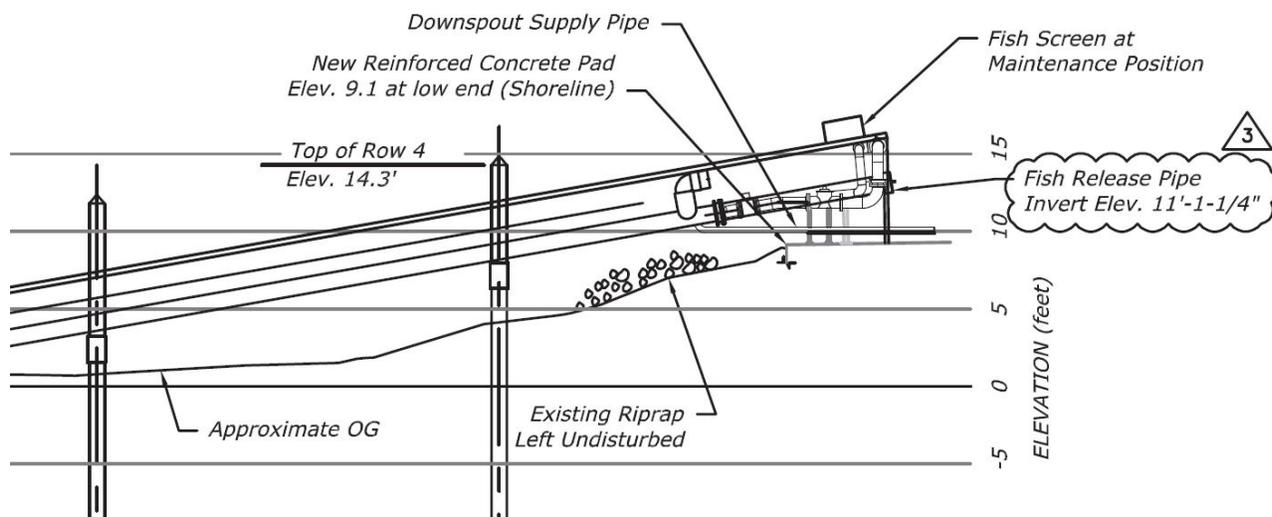


Figure 2.—The auxiliary flow manifold and upstream portion of the release pipe are shown in the as-built drawings. The pipe inlet was designed to be at elevation 11.9 ft but was constructed at elevation 11.1 ft. The rest of the manifold is the same as the original design.

Site Visit

As part of the investigation, a site visit was conducted at the end of May 2023. During this site visit several photos and videos were taken of the site during a full release cycle. Flow rates were measured through the auxiliary flow manifold using the installed flow meter but there was no flow measurement on any of the leakage. A Federal haul truck was on-site with a full tank of water without any fish or debris to provide an accurate release flow. The Federal trucks and the state trucks have slightly different geometries that result in the flex trunk being extended further into the pipe for the Federal truck than the state truck.

The inlet of the pipe was observed to leak substantially with just 3 ft³/s of flushing flow and the flex trunk inserted into the pipe (figure 3). When the truck was moved forward to where the state truck would be, the leakage was reduced but not fully stopped. When measured at the base federal truck position, the flex trunk was found to be interfering with the confluence of the auxiliary flow manifold and adding to the resistance to flow moving downstream.

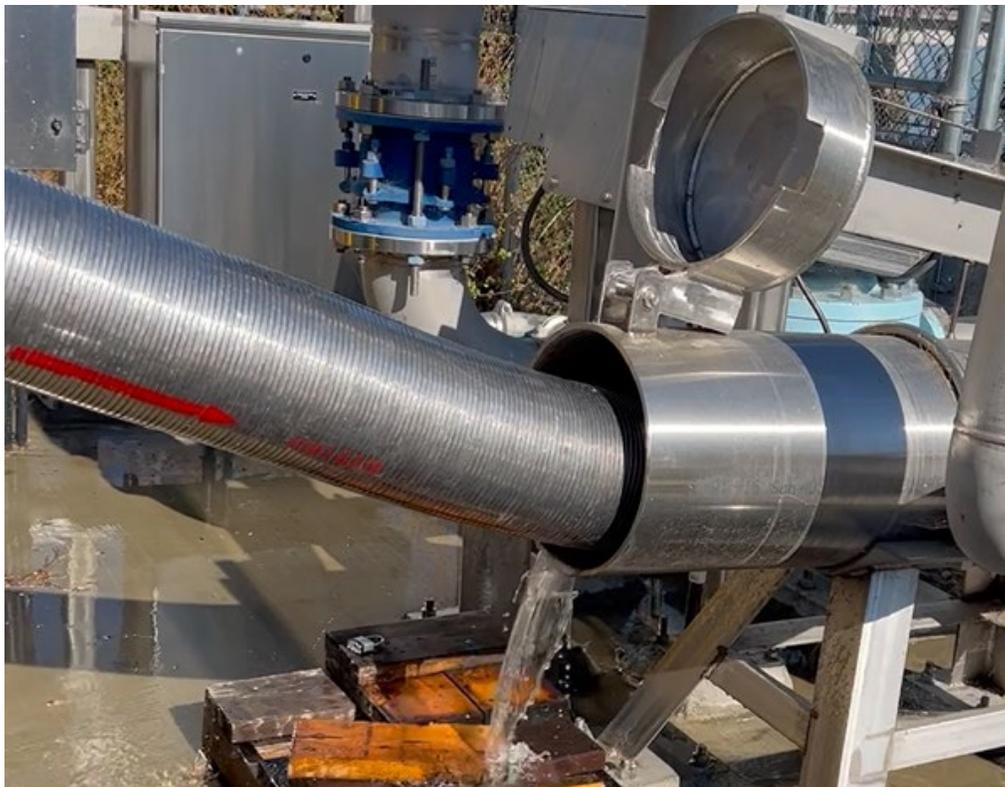


Figure 3.—Example of a small leakage flow during release operations with the flex trunk inserted into the pipe.

During a full truck release without any auxiliary flow, no leakage was observed from the inlet of the release pipe. When tested with the required auxiliary flow, there was leakage. In one trial, the leakage behaved much more like blowback (characterized by surging flow caused by aeration within the flow) than with the pipe inlet elevation being too low for the flow.

Using a 2-inch x 6-foot (ft) board found on site, a partial blockage of the release pipe was created by inserting the board under the flex pipe to observe the effect of raising the inlet. There was an improvement but there was still some leakage with the flex trunk wedged into the top of the pipe. Furthermore, this showed that any partial blockage of the opening to artificially raise the inlet would not be feasible due to the space required to maneuver the flex trunk.

Additionally, the air vent on the release pipe was consistently blowing air from its opening, indicating a high pressure inside of the pipe. The air vent was also observed spraying water during the largest spillage which indicated the potential for blowback in the system.

Analytical Assessment

To investigate the air vent behavior observed during the site visit, numerical spreadsheet analysis was done to investigate the significance of blowback and air entrainment on spillage observed in the pipe. When air is entrained in closed conduit flow and is not cleared out of the downstream side of the pipe it can form large pockets of air which can surge rapidly upstream when buoyancy forces overcome the momentum of the flow. This condition is known as blowback and is a concern for operational issues and in some cases safety concerns (Falvey 1980). This phenomenon was not seen in the 1:4 scale physical model study.

Using an approach from *Engineering Monograph 41: Air-Water Flow in Hydraulic Structures* (Falvey 1980) the four different slope and flow combinations from the physical model and prototype structure were plotted against known observations of air entrainment in figure 4. The flow rates analyzed were 3 ft³/s for the flushing flow and 1.75 ft³/s for the auxiliary flow and the slopes were 10° for the steep slope and 1.67° for the shallow slope. On the figure, the plotted points for the steep and shallow slope are functionally the same for the physical model and the as-built structure. This is because the difference in slope is so small that they are within the marker on the graph. The dimensionless flow rate is also equal between model and prototype. For blowback free operation, air that is entrained into the flow will move either upstream or downstream consistently, as shown by the green, blue, and red points. Instabilities can occur in conditions where air bubbles move downstream and coalesce into air pockets which then can rapidly surge upstream as shown by the yellow point. This aligns with the field observations when the flushing flow caused a blowback event to occur.

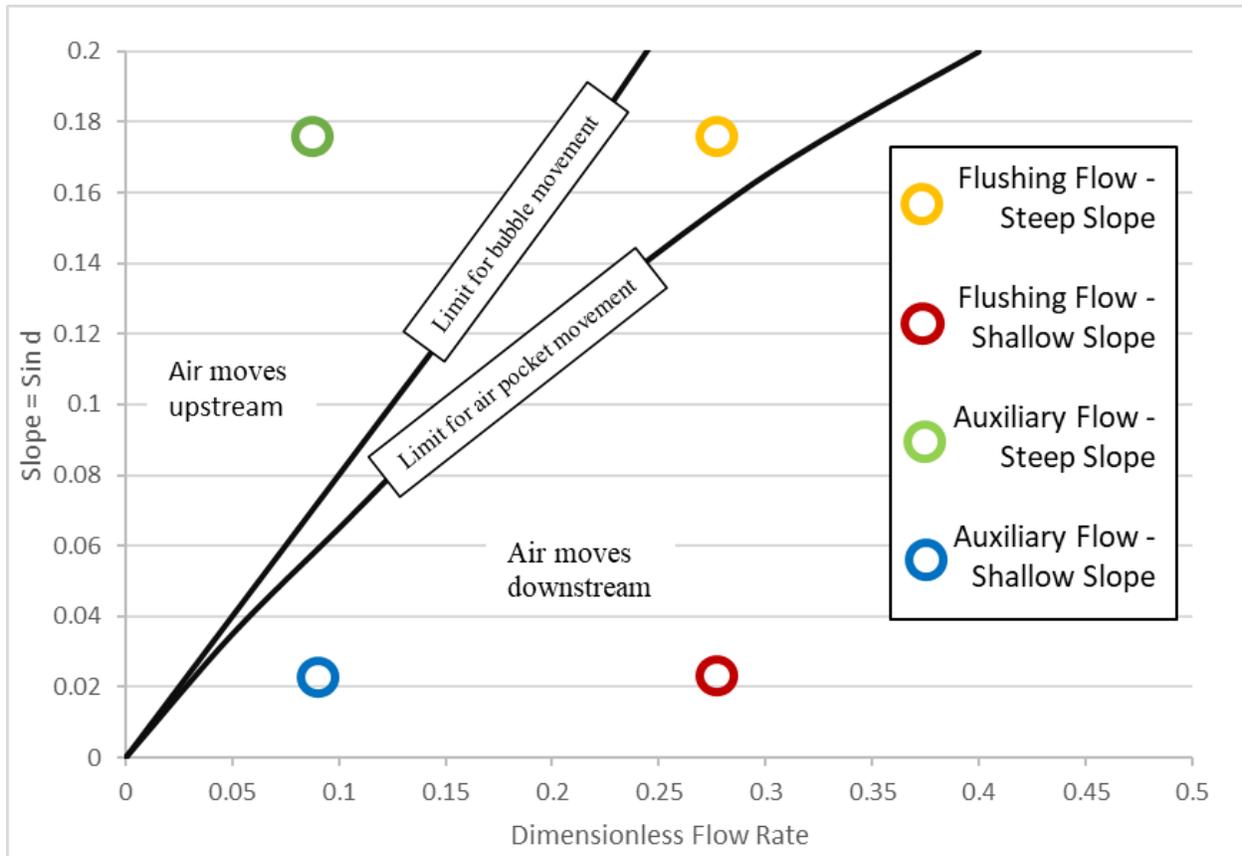


Figure 4.—Antioch Release Pipe operating conditions compared to blowback guidance (Falvey 1980). Air is entrained and moves downstream in the shallow sloped section of pipe. Air can move downstream or upstream depending on the flow in the steeper section of pipe.

Primarily in the first half of the pipe, where the slope is the steepest, there is a chance for air pockets to build up and surge upward during the 1.75 ft³/s auxiliary flow. This is likely what is causing the occasional blowback that was observed in the field, but the constant overflow is likely due to other factors. Since blowback was not found to be a concern in the physical model, the blowback observed in the field is likely due to an operational difference in how the model and prototype pumps work. This led to an investigation of the confluence of the auxiliary flow from the two legs of the manifold with the flushing flow, combined with a shallower pipe slope, as a potential cause of the overflow. This was the primary focus of the computational fluid dynamics (CFD) model.

Computational Fluid Dynamics (CFD) Model

A computational fluid dynamics (CFD) model was used to evaluate hydraulic conditions within the Antioch release pipe as well as potential modifications, with the intent of preventing overflow by improving the confluence of flow from the upstream release pipe (flushing flow from haul truck) with auxiliary flow from the pipe manifold. The commercially available software package FLOW-3D[®] developed by Flow Science Inc. was used for the analysis. FLOW-3D[®] solves the Reynolds-averaged Navier-Stokes (RANS) equations for fluid flow. The Volume of Fluid Method (VOF) developed by Nichols and Hirt (1981) is used to track fluid throughout the structured cartesian mesh. This code was selected due to the ease of use and efficient computation time due to the structured grid approach. This study aimed to create an air entrainment model which simulates the air bubbles being entrained within the fluid. Due to the observed air venting, the full two phase (e.g., air and water) solver was used to better account for the impacts of air within the domain. This allowed for a better representation of the physics and more accurate flow behavior.

Model Extents

The model domain included the release pipe, the flex trunk from the truck, and the manifold for auxiliary flow (figure 5). The exit boundary of the model was approximately 4 feet downstream from the point where the diameter is reduced to 1 foot.

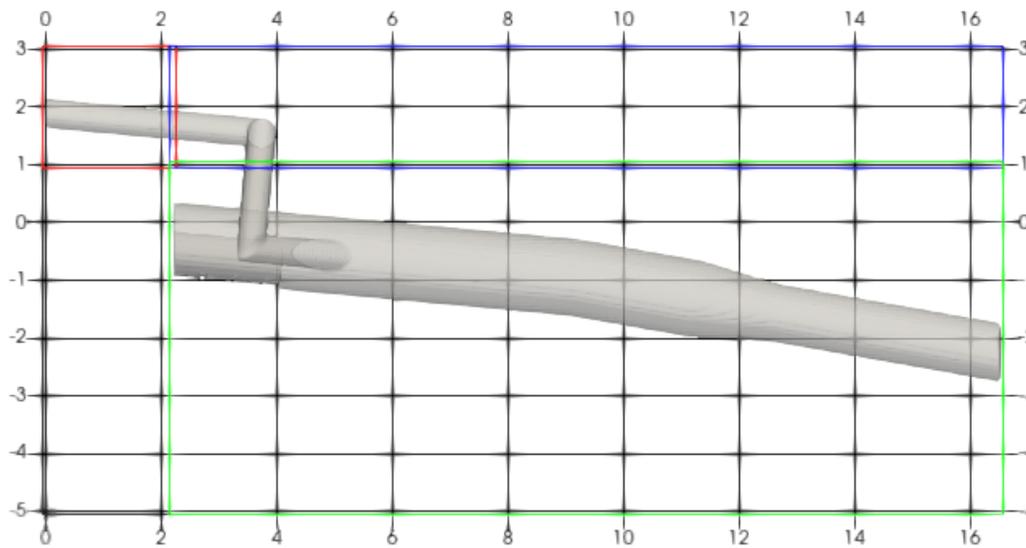


Figure 5.—Model domain from Flow-3D[®]. Units on the X and Z axis are in feet. Initial model runs were done with the flex trunk interfering with the auxiliary flow, but all final model runs were done with it out of the confluence.

A uniform mesh cell size of 0.05 ft was used which allows approximately 10 cells across the diameter of the auxiliary manifold, the smallest diameter pipe in the system (6 inch diameter). Finer resolutions were explored as part of model setup, but the substantially increased runtimes made this infeasible based on project schedule. The chosen mesh accurately represented the hydraulic conditions for the purposes of this model study as the finer resolutions showed similar trends as the coarser mesh size.

Boundary Conditions

The model used several boundary conditions and locations. Fluid can enter the domain through the manifold inlet as well as the fill pipe inlet and water can leave the domain through the outlet as well as the overflow (i.e., spillage) location shown in figure 6.

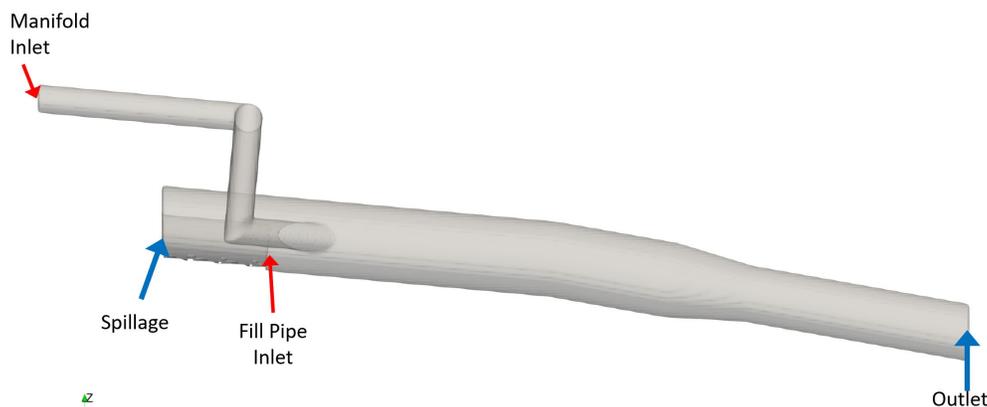


Figure 6.—Red arrows indicate model boundaries where the flow could enter the model domain. Blue arrows indicate boundaries where flow could exit the domain.

The exact boundary conditions varied based on the simulation with different inlet flow rate used to represent the auxiliary flow rate. The auxiliary flow was routed through the manifold at a rate of 3 ft³/s for the pre-flush, 1.75 ft³/s for the supplemental flow during the truck release, and 3 ft³/s for the post-flush. The truck flow was introduced via the flex trunk with the time series shown below in figure 7. A measurement plane was placed at the upstream end of the pipe to capture the volume of the overflow during each trial. The downstream boundary was set to free discharge.

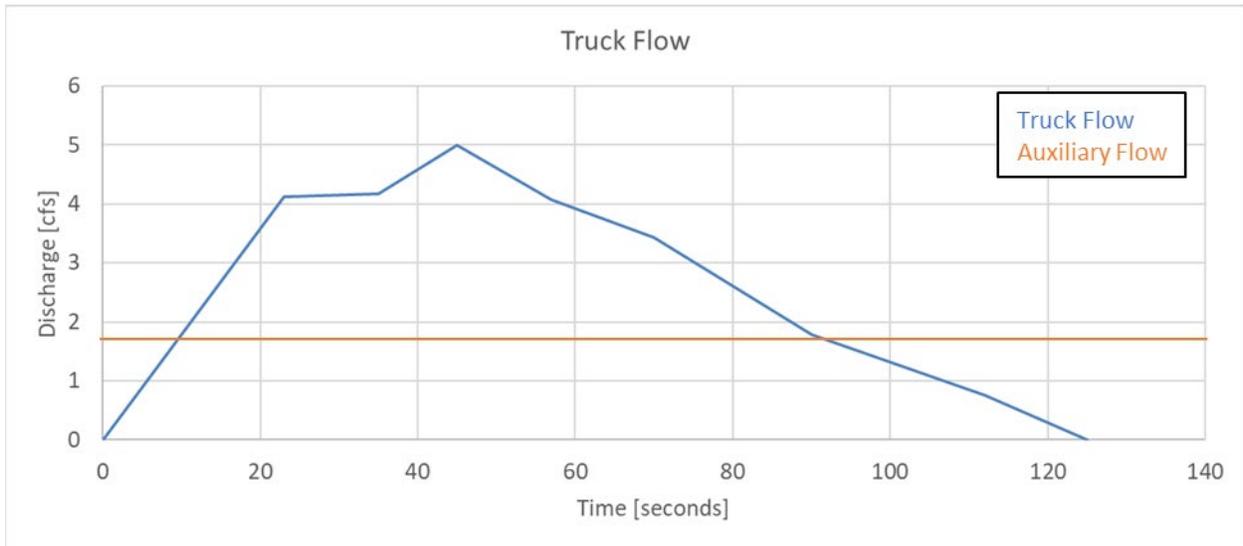


Figure 7.—Truck discharge over time (truck hydrograph). The peak flow during a truck release is 5 ft³/s and the truck takes just over 2 minutes to empty.

Results

Best estimates of numerical settings were used to qualitatively match what was recorded in the videos from the site visit. A series of simulations were performed to better understand existing conditions as well as the effects of operational changes and structural modifications. The simulations are tabulated below in table 2.

Table 2.—Simulation summary

Simulation	Manifold Flow	Truck Hydrograph	Fill Pipe Geometry
Baseline 1	Flushing flow 3.0 ft ³ /s	No	Existing conditions
Baseline 2	Hold constant 1.75 ft ³ /s	Yes	Existing conditions
Operation 1	Hold constant 1.0 ft ³ /s	Yes	Existing conditions
Operation 2	Hold constant 0.5 ft ³ /s	Yes	Existing conditions
Modification 1	Hold constant 1.75 ft ³ /s	Yes	Inlet extend 3 ft at 5°
Modification 2	Hold constant 1.0 ft ³ /s	Yes	Inlet extend 3 ft at 5°
Modification 3	Hold constant 1.0 ft ³ /s	Yes	Inlet extend 2 ft at 10°

Baseline Simulations

Two simulations were performed for the baseline scenarios. First, a flushing operation was simulated where the manifold flow rate was linearly increased from zero to 3.0 ft³/s and held constant. The results show that a few seconds after the full manifold flow rate of 3.0 ft³/s was

attained there was a larger pulse of water spilling from the fill pipe and then the amount of overflow was reduced but did not drop to 0 ft³/s during the simulation (figure 8).

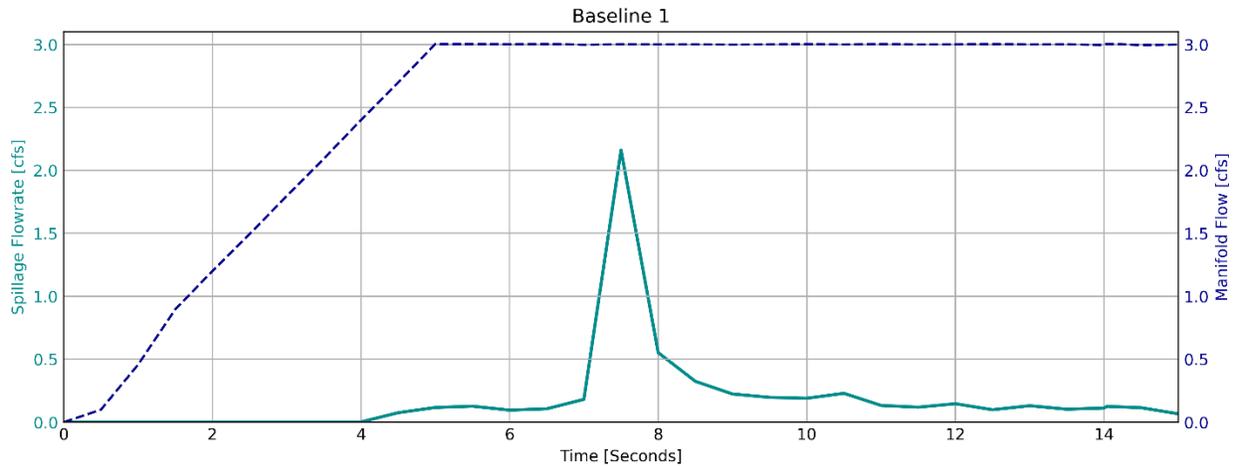


Figure 8.—Baseline 1 simulation, flow was ramped up to 3 ft³/s over 5 seconds and held for 10 seconds. A substantial spillage occurred during this model run.

Baseline 2 simulated a truck emptying where the simulation was initialized with a constant auxiliary flowrate of 1.75 ft³/s from the manifold and then the flushing flow hydrograph from the truck was run through the release pipe (figure 9). The results showed that there was some leakage once the truck discharge began and the overflow increased with truck flow and continued for the duration of the simulation.

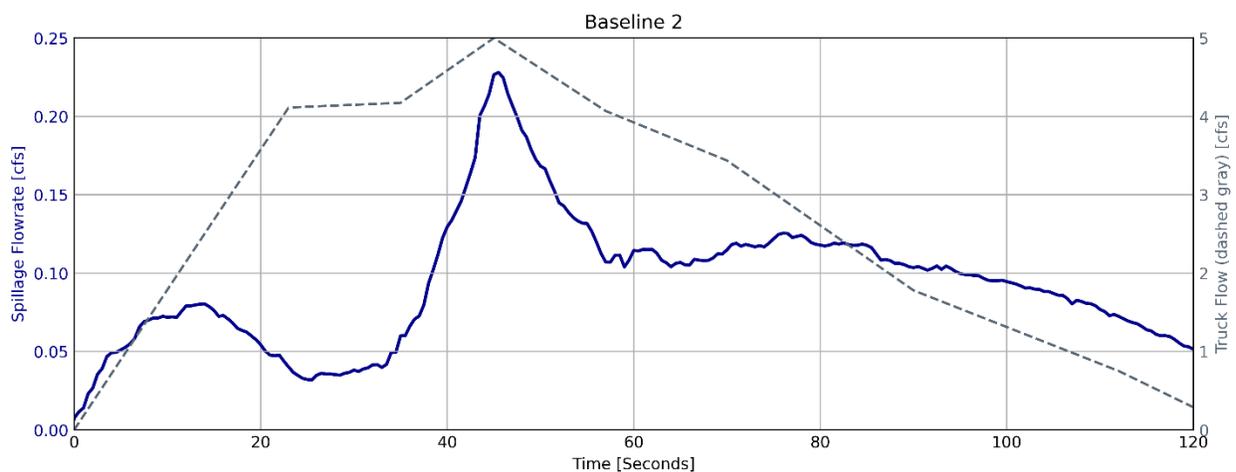


Figure 9.—Baseline 2 simulation, truck flow rate shown with a dashed line on the right axis. Spillage is shown with a dark blue line on the left axis. Note the maximum spillage rate is much lower than that of figure 8.

Figure 10 shows an isometric profile view of the flow field for the second baseline simulation at $t = 50$ seconds, the peak of the potential spillage flow rate. The result of this simulation shows the pipe completely full, and the water surface elevation is above the inlet of the fill pipe, which is consistent with overflow observed in the prototype.

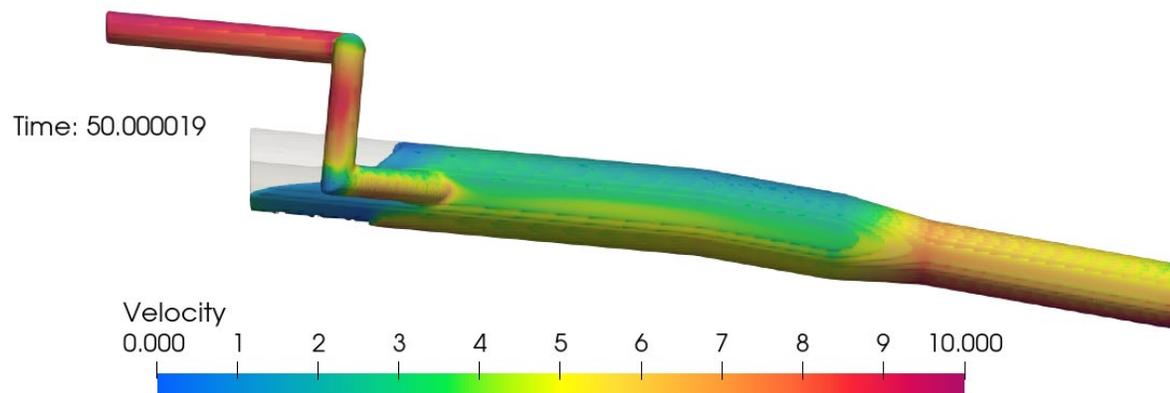


Figure 10.—Baseline 2 simulation CFD flow field. The release pipe flows full with the 5-ft³/s truck flow plus the 1.75-ft³/s auxiliary flow. The spillage flow is consistent throughout the release due to the water surface being above the pipe inlet.

The water surface increase caused by the two jets of the auxiliary flow manifold was found to be the primary reason for the overflow. When the truck flow entered the pipe the increase in volume resulted in water spilling out the back. This aligns with the field observations and provides confidence that the trends shown by the CFD model are consistent with prototype observations.

Operational Simulations

To understand the effect of different operational changes, two simulations were run with different constant manifold flow rates. The results of these simulations are shown in figure 11. The results show that there is a significant reduction in peak spillage with both flow rates compared to Baseline 2. There is a greater reduction going from 1.75 ft³/s to 1.0 ft³/s than there is from 1.0 ft³/s to 0.5 ft³/s.

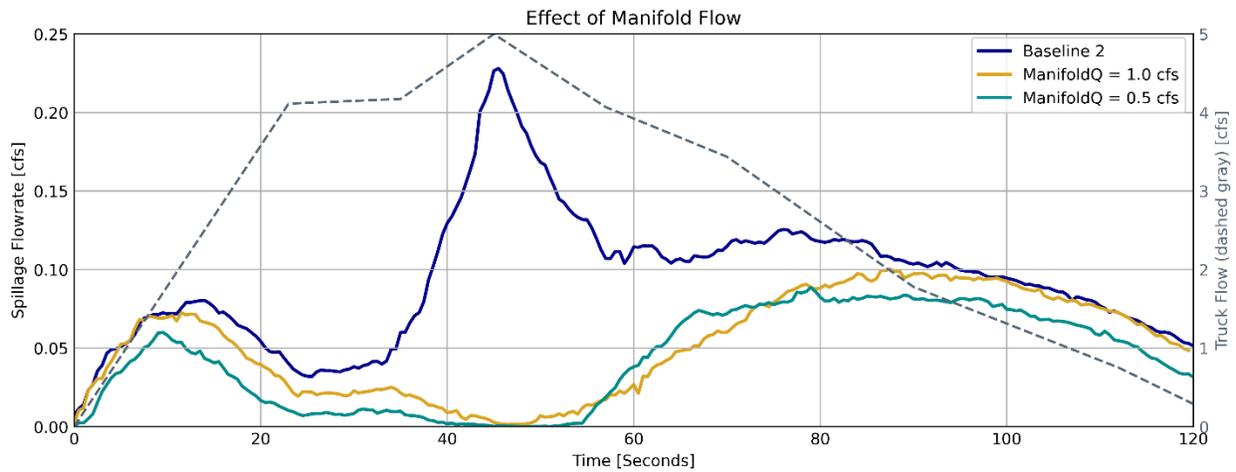


Figure 11.—Operational simulations with the baseline shown in dark blue, the manifold flow reduced to 1 ft³/s shown in yellow and the manifold flowrate reduced to 0.5 ft³/s shown in green.

Modification Simulations

The goal of the modified simulations was to raise the inlet elevation to determine what effect it would have on spillage flow rate. Three different simulations were run with two different geometries and two different flow rates to minimize the leakage flow rate as much as possible. These modifications along with operational changes showed the greatest reduction in leakage of any of the options tested.

The slope of the as-built fill pipe was approximately 5°. One tested geometry extends this slope 3 feet. The second tested geometry extends the fill pipe 2 feet but at a slope of 10°. This slope terminates at the inlet elevation recommended by the design. The results of the simulations are shown in figure 12.

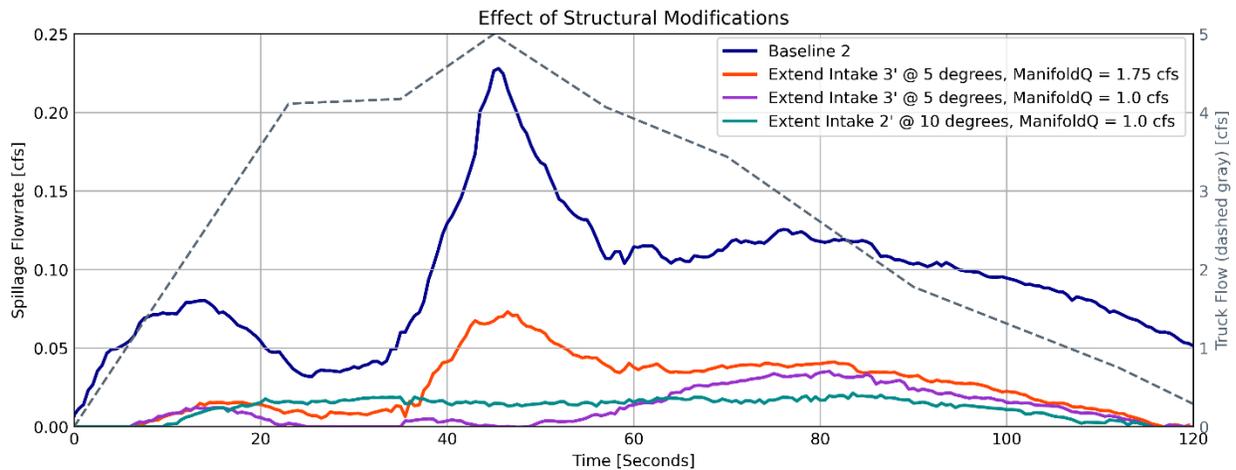


Figure 12.—Modification simulations showed the greatest change in the spillage flow. The baseline is shown in dark blue, the 3-ft extended intake at 5 degrees with the design 1.75-ft³/s release flow shown in orange, the 3-ft extended intake at 5° with the reduced 1-ft³/s release flow shown in purple, and the 2-ft extended intake at 10° shown in dark green.

Additional CFD Simulations of Final Proposed Geometry

Purpose

Additional CFD simulations were run for the final proposed geometry of a 3-ft extension at a 15° incline after feedback from the Tracy Office. Figure 13 illustrates the different geometries for this project where the existing condition is shaded grey and the original design is blue. Green and cyan show two preliminary geometries that were tested above, where green is a 2-ft extension at 10° and cyan is a 3-ft extension at 5°. Red is the current proposed geometry of a 3-ft extension at 15°.

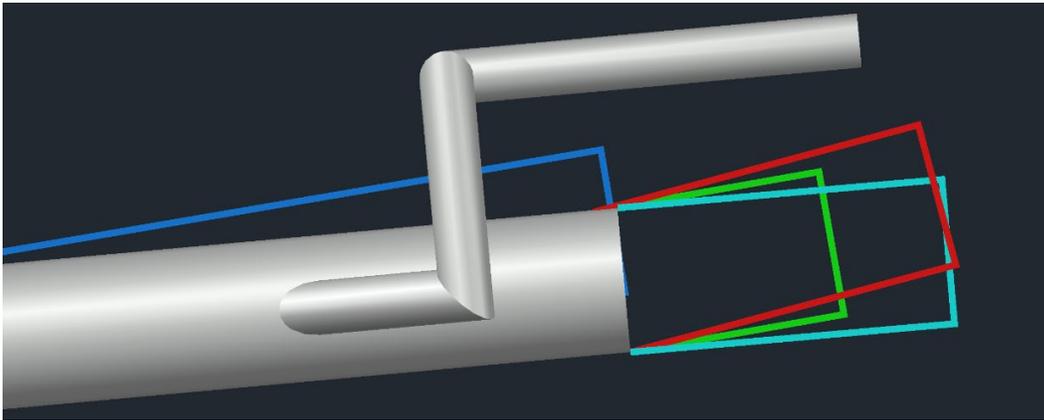


Figure 13.—Summary of tested geometries. The original design geometry is shown in dark blue, the constructed geometry is shown in grey, the preliminary geometries are shown in cyan and green, and the final proposed geometry is shown in red.

Numerical Model – Open FOAM

The previous CFD simulations for the project used the commercial software FLOW-3D[®]. A software update to the code made it no longer possible to run simulations using the same air entrainment settings that were used for those previous simulations. After some difficulties using different parameters with FLOW-3D[®] it was decided to switch to OpenFOAM, a publicly available open-source CFD code. OpenFOAM uses similar numerical methods to approximate the Navier-Stokes equations for fluid flow. OpenFOAM also uses the same VOF method for multiphase flow with the distinction that the code is developed to run on unstructured meshes which better conform to complex geometry. The interFoam solver was selected for these simulations since it is a true two-phase solver where air and water are explicitly simulated throughout the computational domain. The K-OmegaSST-DES turbulence model was used for the final simulations since it provided a better approximation of transient, turbulent flow dynamics. The computational domain was extended further downstream to reduce and influence the outlet boundary may have on the final solution. The resolution of the mesh was increased by decreasing the cell size to 0.03 ft or 0.4 inches. Figure 14 below shows the expanded computational domain.

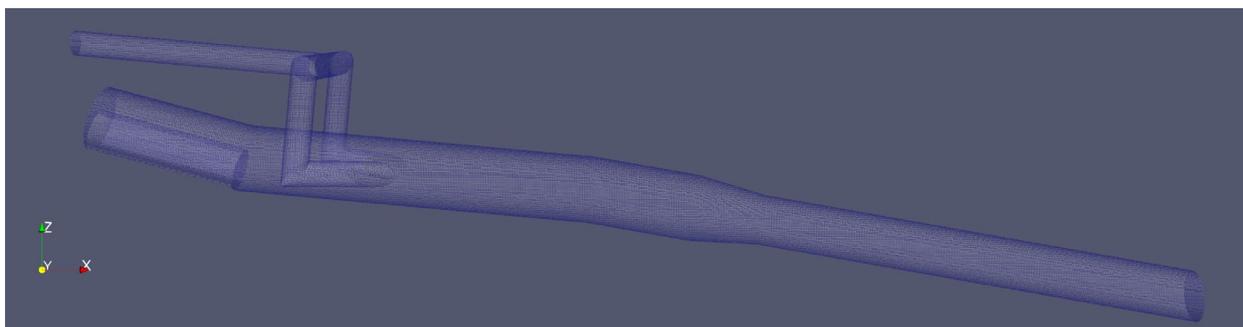


Figure 14.—Computational domain for the OpenFOAM simulations.

Results

These simulations indicated that water did not flow out of the inlet pipe with the proposed 3 ft extension at 15°. The primary reason for this is that the invert of the pipe at the inlet was raised above the level of the wave that is formed in the area where the manifold pipes converge. This is shown by a red line at the elevation of the peak of the wave in figure 15 below.

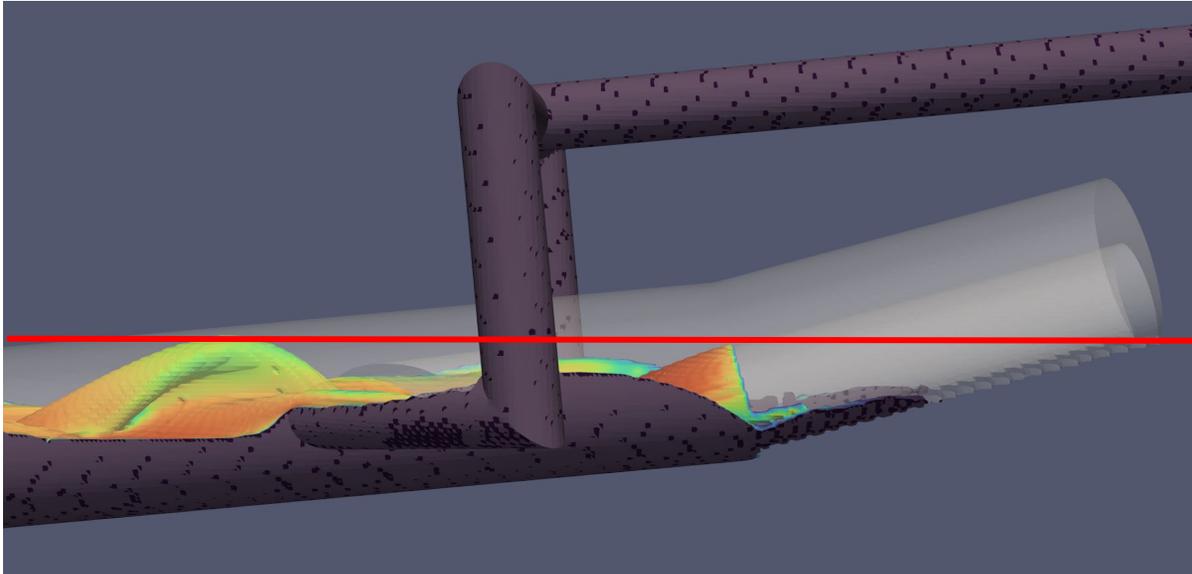


Figure 15.—The inner hydraulics of the pipe with the peak wave height shown by a red line that matches in invert elevation of the final proposed geometry.

With a true two-phase simulation that OpenFOAM utilizes it is possible to quantify the air demand coming from the inlet. Figure 16 illustrates the air demand at the inlet versus the total flow rate for a truck emptying hydrograph. Note that the total flow presented is the truck flow rate plus the constant 1.75 ft³/s coming through the auxiliary flow manifold.

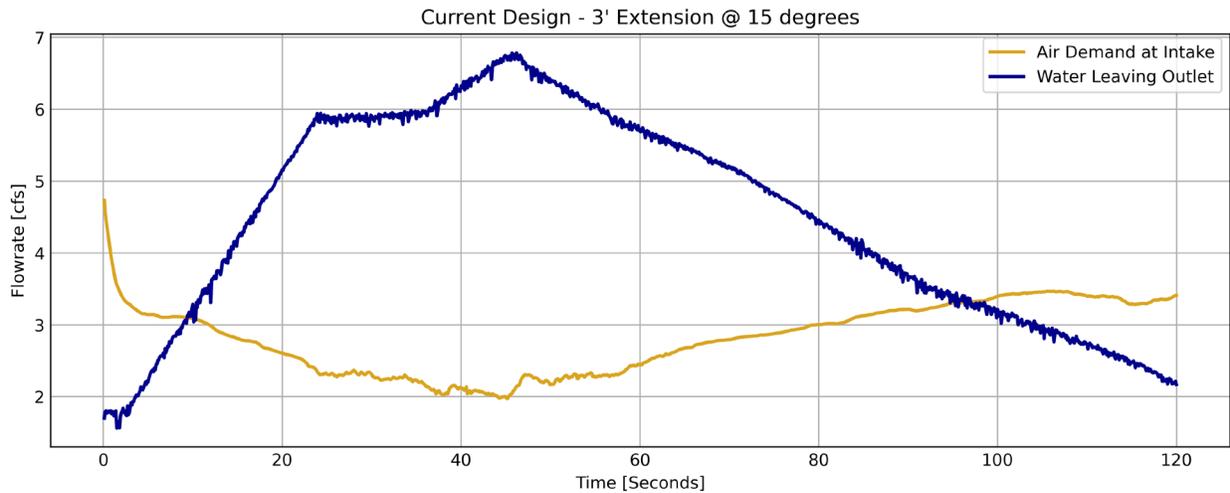


Figure 16.—Air demand of the inflow pipe and flow rate of the truck plus the auxiliary flow manifold.

Summary

The simulations performed for the final proposed geometry suggest that there is a very low risk of water back flowing out of the inlet pipe. The primary reason for this improvement is that the extension raises the invert of the fill pipe above the water level in the manifold pipe convergence zone.

To match what was simulated in the numerical model there is an operational suggestion to limit the insertion of the flex pipe to 3 ft to allow room for water to spread out in the convergence zone of the fill pipe and manifold flow.

Conclusions and Recommendations

Due to differences in the constructed release pipe from those proposed for design by a physical hydraulic model, the release pipe at the Antioch Fish Release Site overflowed at the upstream end through the inlet during release operations, creating concerns for safe fish release. The aim of this study was to find the cause of the overflow as well as a viable solution to minimize this problem.

The primary issue causing the overflow at the Antioch Fish Release Site was the low elevation of the upstream pipe inlet relative to the auxiliary flow manifold and confluence with the jets. Additionally, there was some air entrainment blowback that caused temporary surging and increased the overflow. Results from a field test conducted in May 2023 indicate that air entrainment can largely be controlled operationally by reducing the pre-flush flow rate below the current threshold of 3 ft³/s.

In the absence of calibration data, the CFD results should be used for qualitative and comparative analysis, not to provide absolute values. However, the simulations indicate that a significant reduction in overflow can be achieved by reducing the manifold flow rate, as corroborated by field observations. Several geometries were tested using FLOW-3D[®] that showed significant improvement. This includes extending the upstream intake 3 ft at a 5° incline or extending the intake 2 ft at a 10° incline. After consulting with the Tracy Office, additional testing was completed using OpenFOAM to test a 3-ft extension at 15° that showed no leakage from the entrance to the release pipe when the auxiliary flow rate was set to 1.75 ft³/s and the flex pipe inserted 3 ft into the release pipe. This configuration is recommended for construction during final modification of the release pipe.

References

- Carter-Gibb, J. 2020. Antioch Fish Release Site Replacement Physical Hydraulic Model. Bureau of Reclamation, Technical Service Center, Denver Colorado.
- Falvey, H.T. 1980. Engineering Monograph 41: Air-Water Flow in Hydraulic Structures. U.S. Department of the Interior, Bureau of Reclamation, Water and Power Resources Service, Denver, Colorado.
- Nichols, B.D., and C.W. Hirt. 1981. Volume of Fluid (VOF) Method for Dynamics of Free Boundaries. *Journal of Computational Physics* 39(1): 201–225.