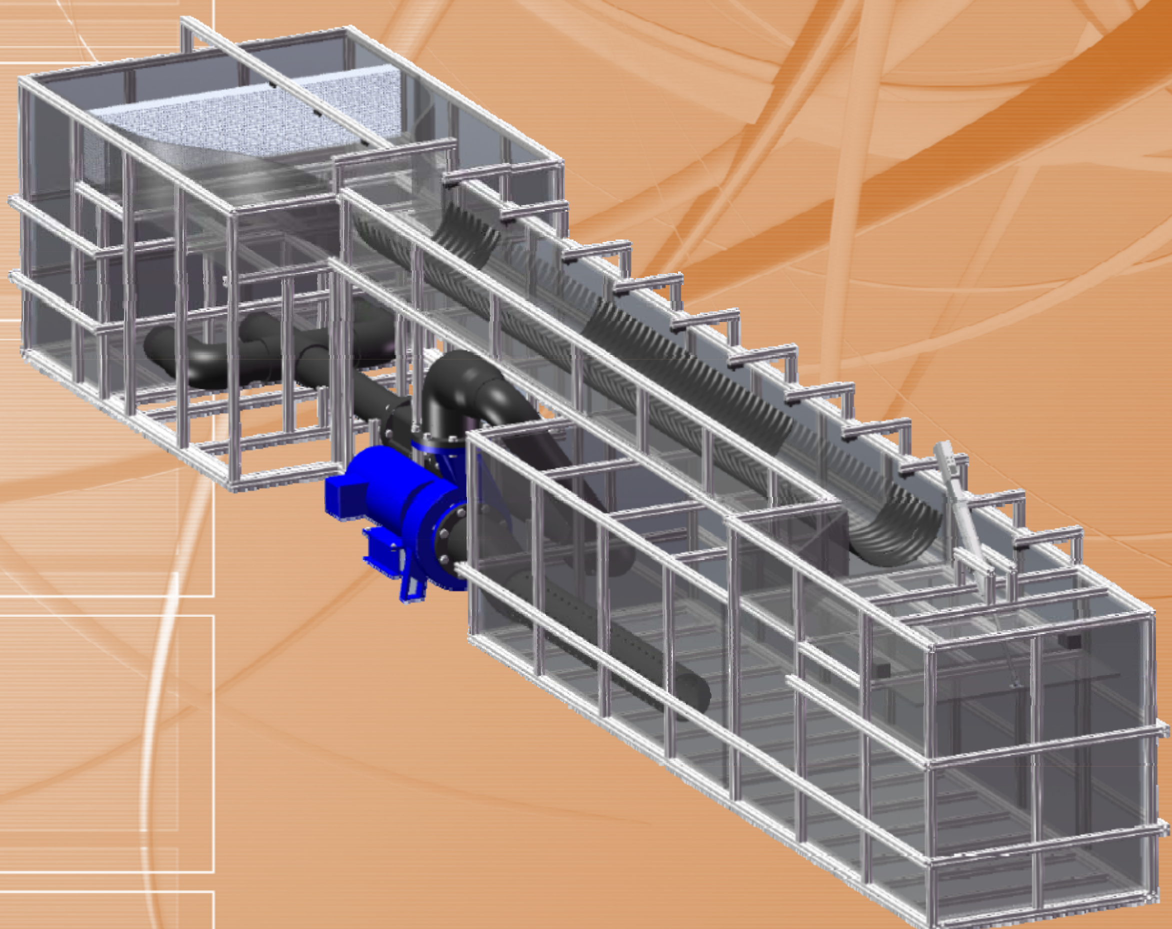
NATIONAL LABORATORY

Computational Mechanics Research and Support for Aerodynamics and Hydraulics at **TFHRC**



Culvert Analysis Quarterly Report

October through December 2011

Computational Fluid Dynamics Modeling of Flow through Culverts
2011 Quarter 4 Progress Report

Transportation Research and Analysis Computing Center (TRACC)
Energy Systems Division (ES)
Argonne National Laboratory (ANL)

Principal Investigator:
Steven A. Lottes, Ph.D.

Contributing CFD Investigators:

Sudhir Lanka Venkata
Pradip Majumdar, Ph.D.
Northern Illinois University

Yuan Zhai
University of Nebraska

Submitted to:
Federal Highway Administration

Kornel Kerenyi, Ph.D.
Turner-Fairbank Highway Research Center
Federal Highway Administration
6300 Georgetown Pike
McLean, VA 22101

January, 2012

Table of Contents

List of Figures

List of Tables

1. Introduction and Objectives

This project was established with a new interagency agreement between the Department of Energy and the Department of Transportation to provide collaborative research, development, and benchmarking of advanced three-dimensional computational mechanics analysis methods to the aerodynamics and hydraulics laboratories at the Turner-Fairbank Highway Research Center (TFHRC) for a period of five years, beginning in October 2010. The analysis methods employ well-benchmarked and supported commercial computational mechanics software and also include user subroutines, functions, and external software programs and scripts written to automate the analysis procedures. Computational mechanics encompasses the areas of Computational Fluid Dynamics (CFD), Computational Wind Engineering (CWE), Computational Structural Mechanics (CSM), and Computational Multiphysics Mechanics (CMM) applied in Fluid-Structure Interaction (FSI) problems.

This quarterly report documents technical progress on the CFD modeling and analysis of flow through culverts for the period of October through December 2011. The focus of effort for the work this year is on improving methods to assess culvert flows for fish passage.

2. Computational Modeling and Analysis of Flow through Large Culverts for Fish Passage

Culverts are fixed structures that do not adapt to changing streams and may instead become barriers to fish movement. The most common physical characteristics that create barriers to fish passage include excessive velocity, insufficient water depth, large outlet drop heights, turbulence within the culvert, and accumulation of sediment and debris. Primary variables that control the possibility of fish passage are: flow rates during fish migration periods, fish species, roughness, and the length and slope of the culvert.

The objective of this work is to develop approaches for CFD modeling of culvert flows and to use the models to assess suitability of the culvert for fish passage under a variety of flow conditions. The flow conditions to be tested with CFD analysis are defined in the tables of a work plan from TFHRC [6]. The CFD models are verified by comparing computational results with data from experiments conducted at TFHRC. A primary goal of CFD analysis of culverts for fish passage is to determine the local cross section velocities and flow distributions in corrugated culverts under various flow conditions. In order to evaluate the ability of fish to traverse corrugated culverts, the local average velocity from the region adjacent to the culvert wall out to the centerline under low flow conditions will be determined.

3. Modeling Culvert Flow above a Gravel Bed

One of the objectives of the CFD analysis during this quarter is to investigate methods to model gravel bed in the culvert. The test matrix in the TFHRC work plan includes tests with the bed height at 15% and 30% of the culvert diameter. The culvert bed material is coarse gravel with a median diameter of D_{50} = 12 mm. Since the experiments in the TFHRC were conducted by gluing a layer of gravel on the flume bed, the CFD modeling culvert flow over a gravel bed is done by producing meshing that simulates the rough bed contour created by the top layer of gravel.

4. Tests scenarios

During this quarter, both experiments and CFD model simulated the culvert flow with sediment bed. The flow velocities are 0.71ft/s and 1.1ft/s, and the water depths are 4.5 inch, 6 inch and 9 inch. The proposed gravel has a median diameter, D_{50}, of 12 mm. This size was used as a reference in CFD modeling, although sieve analysis showed a slightly lower value of D_{50} (10.5mm). The flume with the corrugated pipe is illustrated in Figure 1.

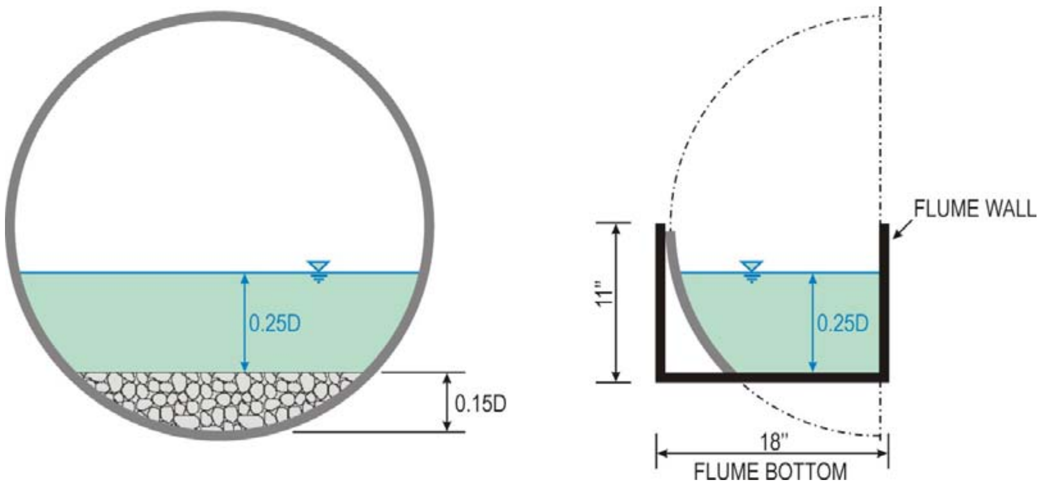


Figure 1 **Bed elevation at 0.15 culvert diameter and symmetrical half of flume culvert model**

5. CFD modeling methodology

The method of simulating the gravel in CFD model is to produce meshing that mimics the contour of the top layer of the gravel. The CAD model was created in Pro-ENGINEER. Figure 2 illustrates the method of creating a D_{50} =24mm gravel. In order to avoid numerical error caused by sharp angles in cells in the grid, two quarter-circular arcs with the diameter of D_{50} are used to get smooth connections. Figure 3 shows a part of the model with such surface features simulating gravel bed.

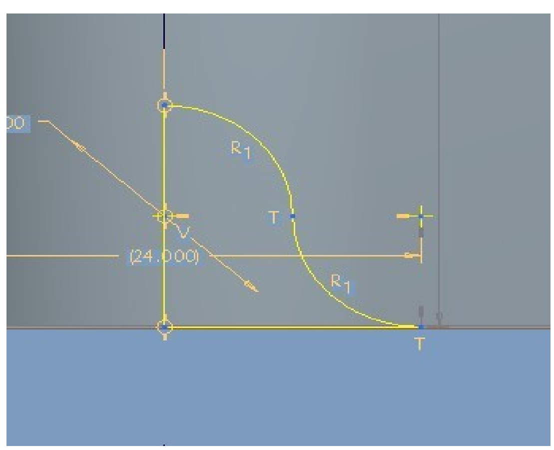


Figure 2 **Illustration of the gravel creation method in the CAD model**

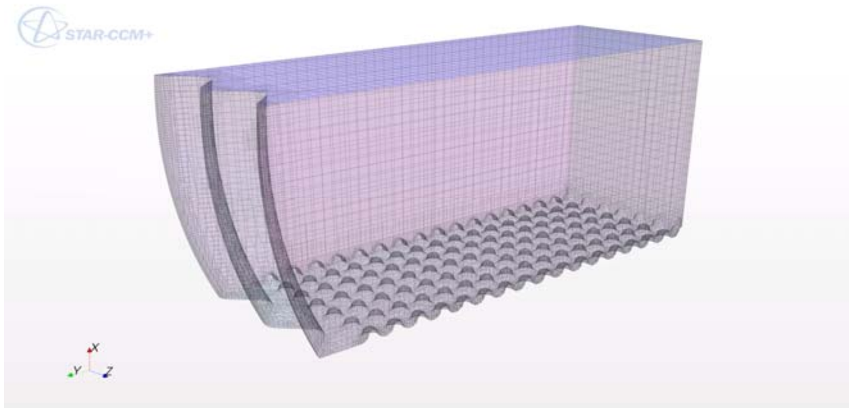


Figure 3 **CAD model with the gravel bed surface created by the gravel geometry**

6. Comparison between CFD and experimental results

In order to verify the CFD model by comparison with experimental results, comparison was made among CFD, PIV, and ADV data in corresponding areas of a plane cross section normal to the primary flow direction. Grids of data are produced for comparison by interpolation from the original computational cell data and observed data. The depth-averaged velocity and cumulative average velocity curves are calculated based on the CFD data. The comparisons between CFD and experimental results and the velocity curves computed for different flow conditions are shown in Figure 4 through Figure 15. Note that the CFD simulation uses cyclic boundary conditions between inlet and outlet planes. The cyclic boundary makes the CFD results correspond to a fully developed flow condition, while the test section in the experimental flume is only a couple of meters downstream of the culvert inlet, which is not sufficient to attain a fully developed flow condition.

1) Velocity is 0.71 ft/s, and the water depth is 4.5 inch

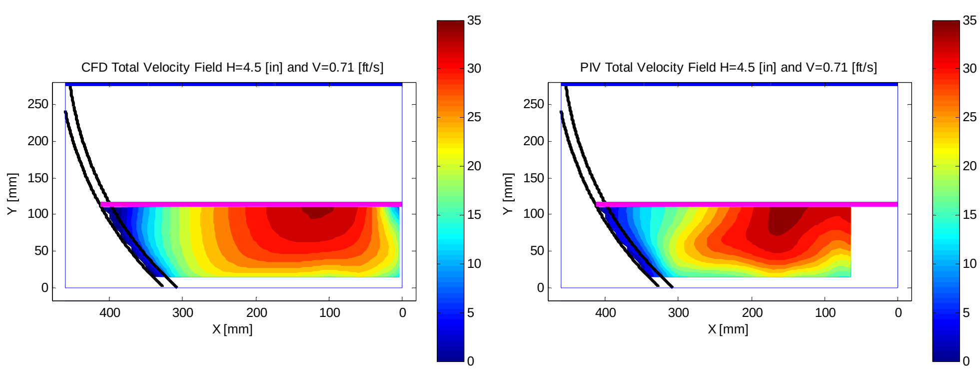


Figure 4 **Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 0.71fts and 4.5inch water depth**

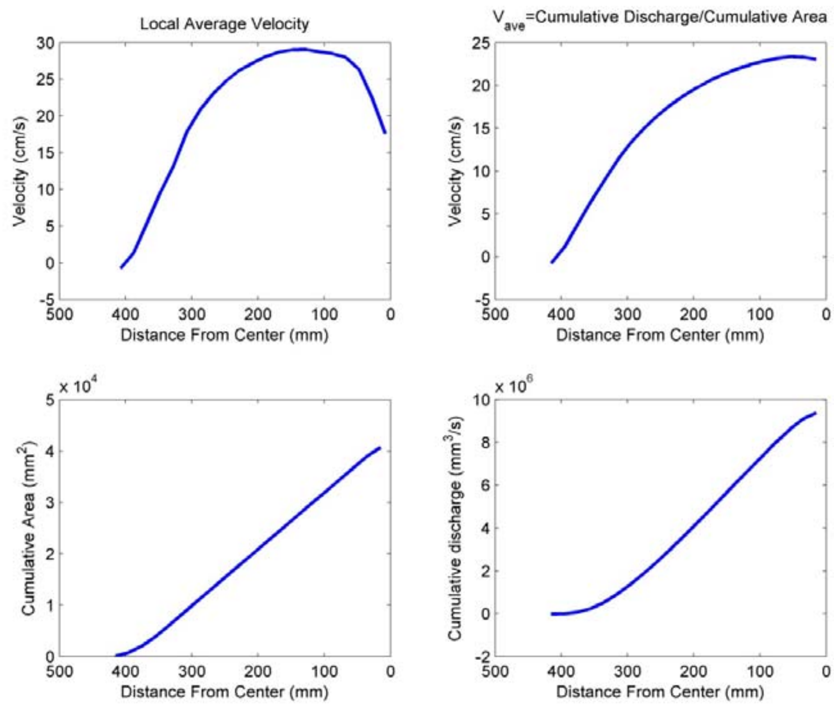


Figure 5 Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 0.71fts and 4.5inch water depth

2) Velocity is 0.71 ft/s, and the water depth is 6 inch

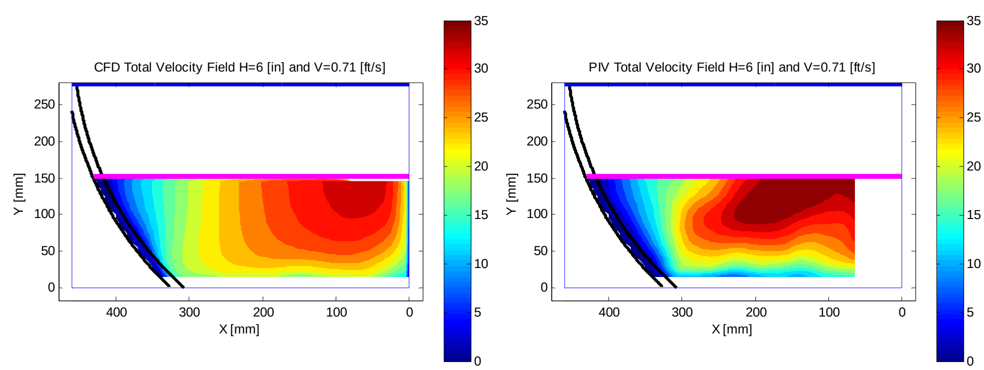


Figure 6 Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 0.71fts and 6 inch water depth

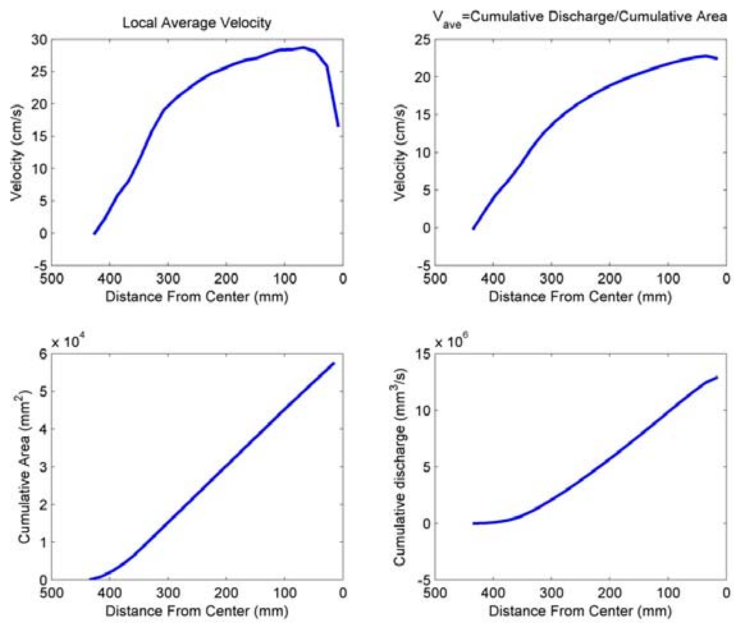


Figure 7 Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 0.71fts and 6 inch water depth

3) Velocity is 0.71 ft/s, and the water depth is 9 inch

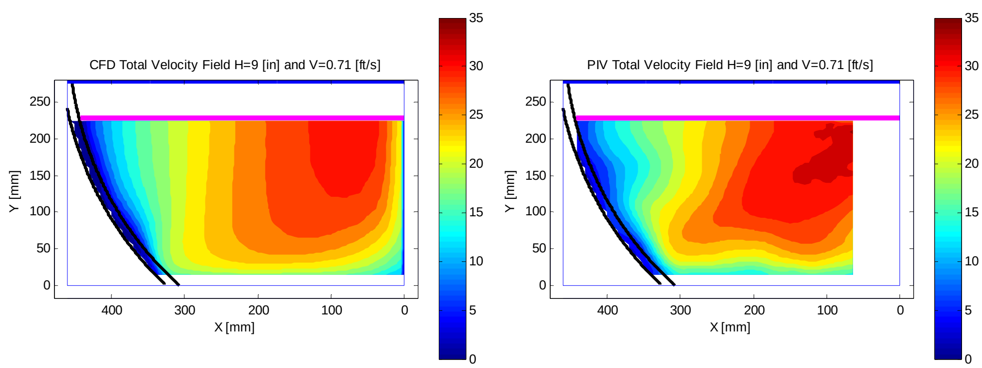


Figure 8 Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 0.71fts and 9 inch water depth

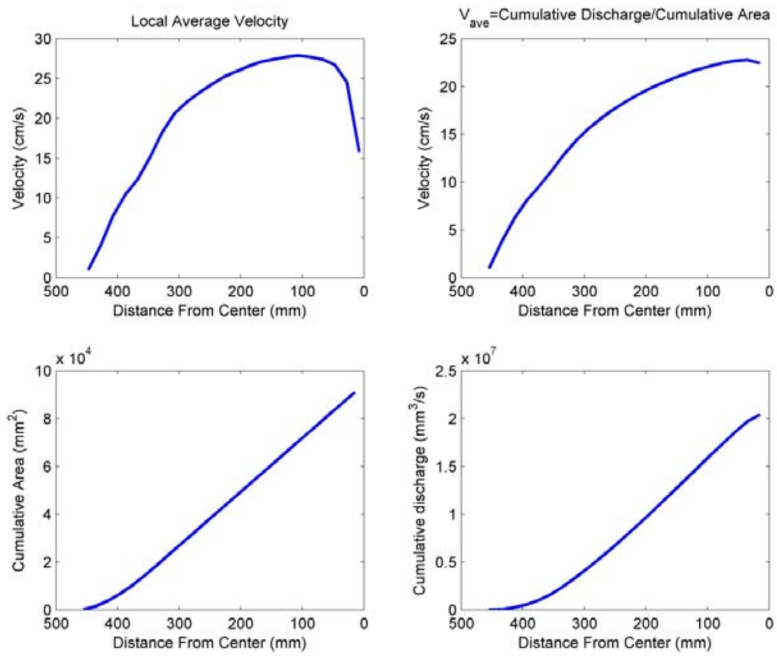


Figure 9 Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 0.71fts and 9 inch water depth

4) Velocity is 1.1 ft/s, and the water depth is 4.5 inch

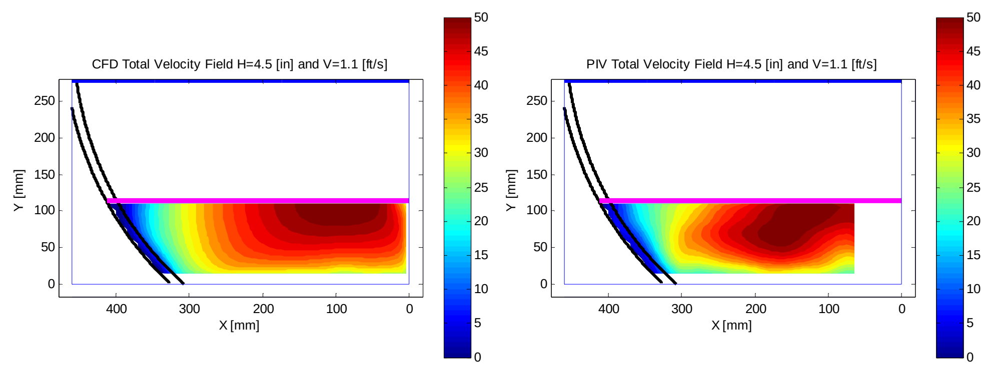


Figure 10 Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 1.1fts and 4.5 inch water depth

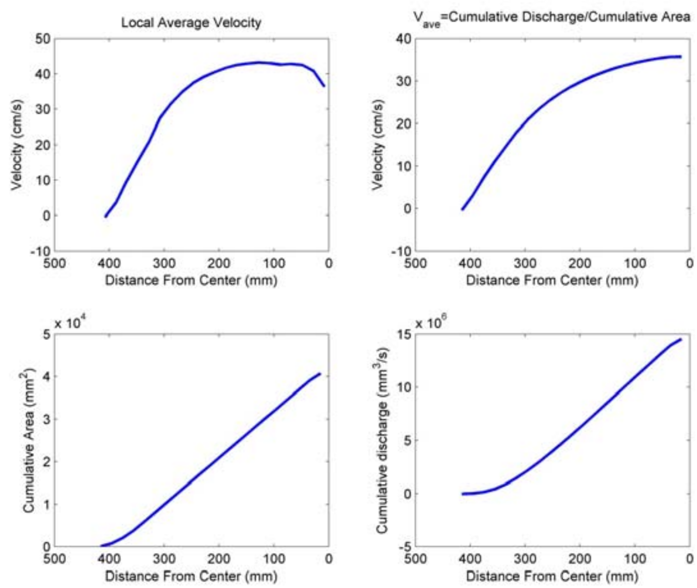


Figure 11 Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 1.1fts and 4.5 inch water depth

5) Velocity is 1.1 ft/s, and the water depth is 6 inch

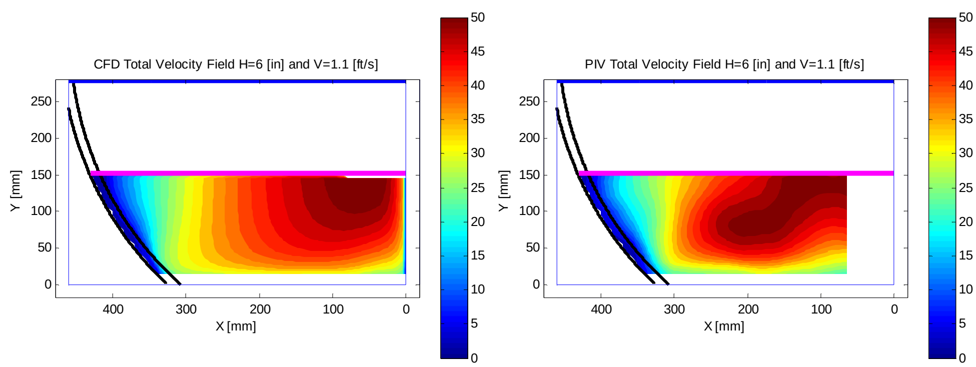


Figure 12 Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 1.1fts and 6 inch water depth

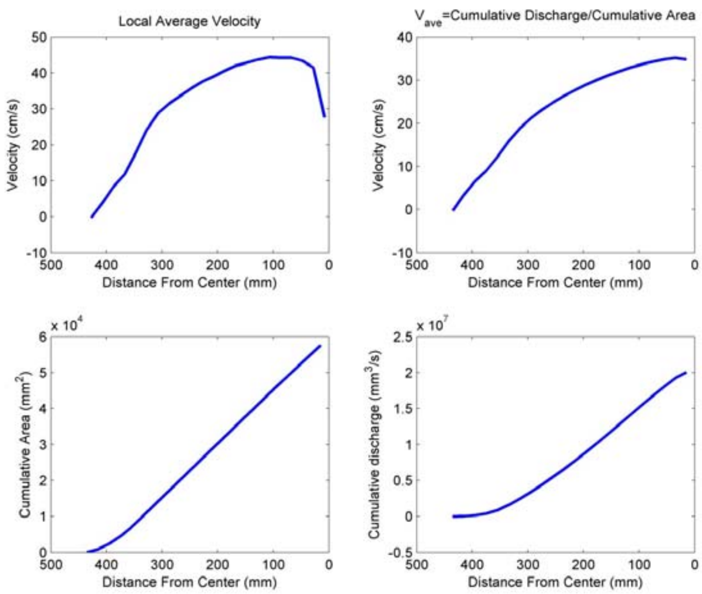


Figure 13 Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 1.1fts and 6 inch water depth

6) Velocity is 1.1 ft/s, and the water depth is 9 inch

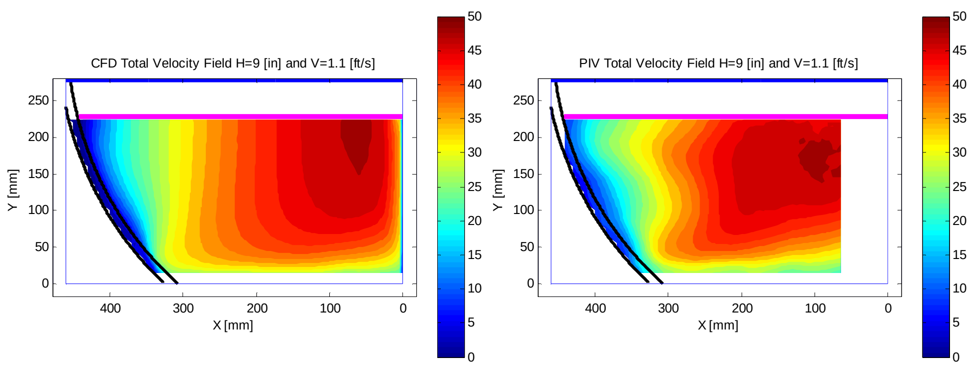


Figure 14 Comparison of the velocity distribution contour between CFD and experimental data under the flow condition of 1.1fts and 9 inch water depth

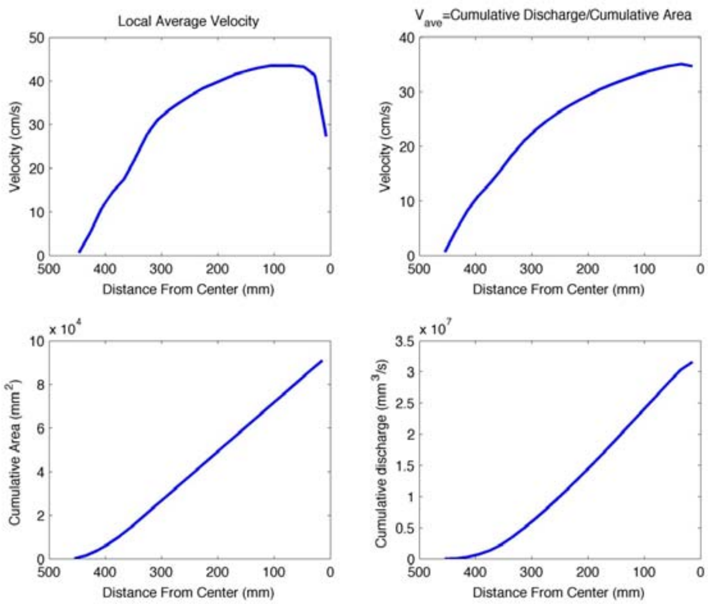


Figure 15 **Depth-averaged velocity and cumulative average velocity curves development under the flow condition of 1.1fts and 9 inch water depth**

7. Data process and error analyses

Original velocity data from CFD for each cell are exported and interpolated into uniform grids of 5 mm×5 mm grid in MATLAB. The PIV and ADV data can also be interpolated into uniform grids of 5 mm × 5 mm for the convenience of comparison. Root–mean–square deviation (RMSD) is used to assess error. RMSD is a frequently used measure of the difference between values predicted by a model or an estimator and the value actually observed from the thing being modeled or estimated. RMSD is a good measure of accuracy. These individual differences are also called residuals, and the RMSD combines these error values into a single measure of overall error. Specifically, the RMSD is calculated by the function

RMSD= sqrt (mean(sum of the squares of errors between corresponding grids)).

Based on the 5mm ×5 mm grid data, the RMSD values were calculated for each of the cases, and are listed in the below Table 1.

Table 1 RMSD number between CFD and experimental data

Water depth (inch)	Velocity (fps)	PIV and CFD		ADV and CFD	
		RMSD	Relative error	RMSD	Relative error
4.5	0.71	2.4017	0.11	4.2799	0.20
6	0.71	4.532	0.21	4.3604	0.20
9	0.71	2.2145	0.10	3.8758	0.18
4.5	1.1	3.1694	0.09	6.258	0.19
6	1.1	4.749	0.14	7.3586	0.22
9	1.1	3.4901	0.10	5.7846	0.17

8. Simulating the Gravel Bed with a Porous Media Model

An alternative method that simulates the roughness from gravel bed is using porous media. In a real gravel bed there is some shear induced flow within the bed. Depending upon the permeability of the bed material, various levels of sub-surface flow are possible. The porous media approach not only provides a method to simulate surface roughness, but also includes the effect of the flow under bed surface.

To model flow through a porous media, the STAR-CCM+ User Guide notes that it is not the details of the internal flow in the porous region that is of interest, but rather the macroscopic effect of the porous medium on the overall fluid flow. The effect of the porous medium on the flow is defined using lumped parameters, which are typically taken to be resistance coefficients for two momentum sink terms in the momentum equation. One of them is linearly proportional to velocity and the other proportional to velocity squared. These parameters account for the effect of the distributed surface area of the porous media as a flow resistance on the bulk flow in the porous media and consequent effect on the flow that borders the porous region. As a gravel bed in a culvert, the porous region should have the effect of a very rough wall with a small slip condition at the interface that is determined by the flow resistance of the porous media. Various CFD tests have been conducted to determine if a porous media model appears to be a viable way to model large diameter gravel in the bed of a culvert. Some of the results obtained appeared to be reasonable while other results appeared to be physically unrealistic. The physically unrealistic appearing results did not appear to have viscous transport of momentum from the open flow region into the porous region and prompted a question for CD-adapco technical support regarding the terms included in the porous media model of the momentum because a variety of equations are in common use. In particular, the convective term and/or the viscous term from the full Navier-Stokes equations are often not present in the equations governing bulk flow through proous media and there is not agreement in the literature on equations and interface conditions governing flow parallel to a porous media. Good engineering results have been obtained using the full Navier-Stokes equations with two momentum sink terms, one linear in velocity and the other quadratic in velocity to account for the resistance to flow of the distributed surfaces of the porous media in the volume. Presence of the viscous term in particular is essential to using the porous media model for flow over a gravel bed in a culvert. CD-adapco technical support contacted the developers responsible for the porous media models and determined that the porous media model does use the full Navier-Stokes equations augmented by porous media flow resistance sink terms. A switch in the user interface also allows the convective transport term to be turned off. The physically unrealistic results observed in modeling of porous media in a culvert section were therefore most likely a consequence of mesh refinement in the vicinity of the porous media interface. Because a boundary layer with a small slip condition does exist at the porous media interface, and wall functions for this model are not available to determine shear stress at the interface, using a sufficiently refined grid to resolve the velocity profile at near the interface is important. Further tests to resolve this issue are currently being conducted.

9. References

1. Matt Blank, Joel Cahoon, Tom McMahon, “Advanced studies of fish passage through culverts: 1-D and 3-D hydraulic modeling of velocity, fish expenditure and a new barrier assesment method,” Department of Civil Engineering and Ecology, Montana State University, October, 2008 .
2. Marian Muste, Hao-Che Ho, Daniel Mehl,“Insights into the origin & characteristic of the sedimentation process at multi barrel culverts in Iowa”, Final Report, IHRB, TR-596, June, 2010.
3. Liaqat A. Khan, Elizabeth W.Roy, and Mizan Rashid, “CFD modelling of Forebay hydrodyamics created by a floating juvenile fish collection facility at the upper bank river dam”, Washington, 2008.
4. Angela Gardner, “Fish Passage Through Road Culverts” MS Thesis, North Carolina State University, 2006.
5. Vishnu Vardhan Reddy Pati, “CFD modeling and analysis of flow through culverts”, MS Thesis, Northern Illinois University, 2010.
6. Kornel Kerenyi, “Final Draft, Fish Passage in Large Culverts with Low Flow Proposed Tests” unpublished TFHRC experimental and CFD analysis of culvert flow for fish passage work plan, 2011.
7. CD-adapco, *User Guide STAR-CCM+ Version 6.04.014*, 2011