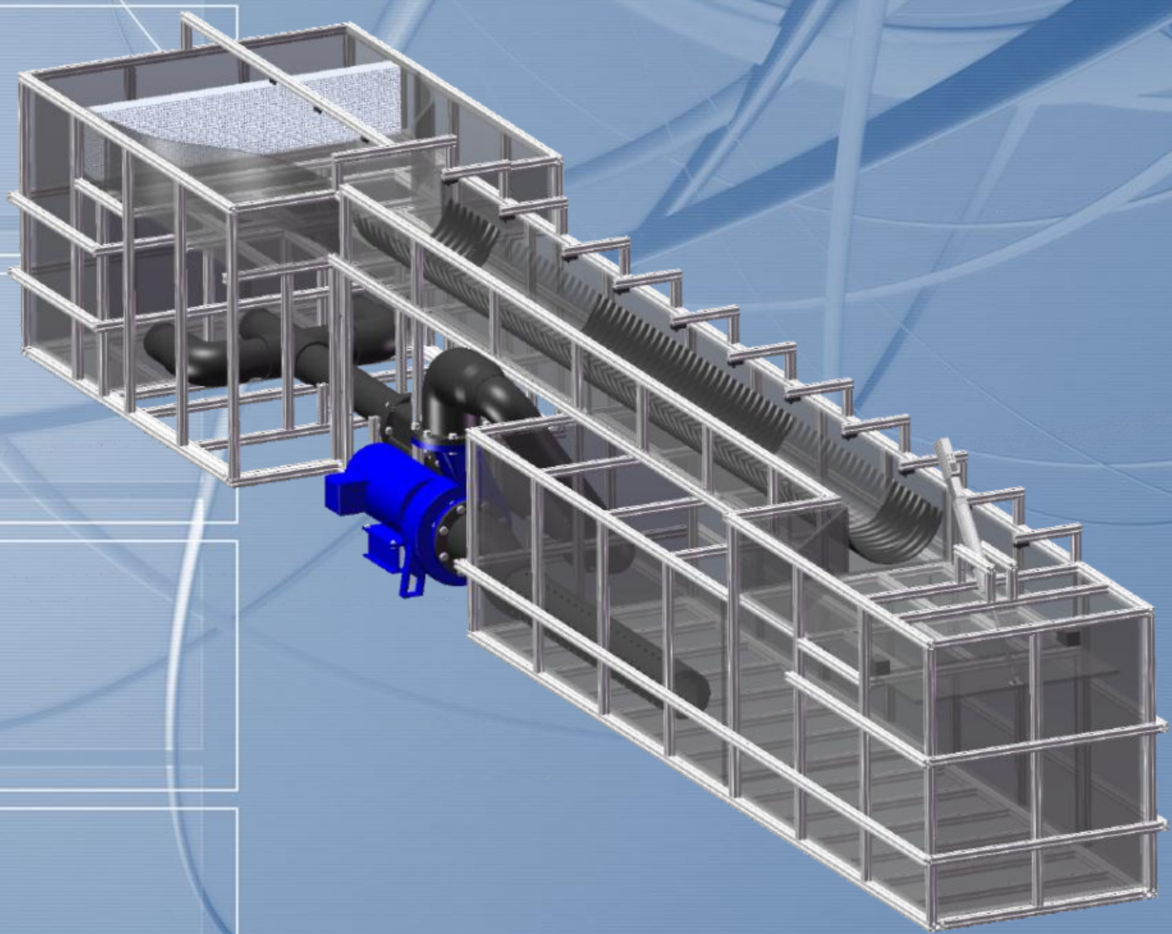
Argonne
NATIONAL LABORATORY

Energy Systems Division

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



# Culvert Analysis Quarterly Report

April through June 2012

**Computational Fluid Dynamics Modeling of Flow through Culverts**
**2012 Quarter 3 Progress Report**

**Energy Systems Division (ES)**
**Argonne National Laboratory (ANL)**


**Principal Investigator:**
**Steven A. Lottes, Ph.D.**
**Argonne National Laboratory**

**Contributors:**

**Yuan Zhai**
**University of Nebraska**

**Jerry Shen, Ph.D.**
**Turner-Fairbank Highway Research Center**


**Submitted to:**
**Federal Highway Administration**


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


**July 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.

This quarterly report documents technical progress on the CFD modeling and analysis of flow through culverts for the period of April through June 2012. 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

Fish passage through culverts is an important component of road and stream crossing design. As water runoff volume increases, the flow often actively degrades waterways at culverts and may interrupt natural fish migration. Culverts are fixed structures that do not change with changing streams and may instead become barriers to fish movement. The most common physical characteristics that create barriers to fish passage include excessive water velocity, insufficient water depth, large outlet drop heights, turbulence within the culvert, and accumulation of sediment and debris. Major hydraulic criteria influencing 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 to CFD modeling of culvert flows and to use the models to perform analysis to assess flow regions 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 being 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 varying flow conditions. In order to evaluate the ability of fish to traverse corrugated culverts, the local average velocity in vertical strips from the region adjacent to the culvert wall out to the centerline under low flow conditions will be determined.

A primary goal of the CFD analysis during this quarter has been the detailed comparison among the results from CFD and those from Particle Image Velocimetry (PIV) and Acoustic Doppler Velocimetry (ADV). The challenge of this task included the variation of measurable area over the entire cross section by the three methods, the difference in original data grid format, and finding a simple representation of the discrepancies in velocity distribution. Most of the comparisons were done between CFD and PIV data. While ADV measurements were limited due to the significant cropping of the flow section, the ADV was considered a very reliable tool and therefore was used to cross-check the comparison done between CFD and PIV under deep water conditions. Good agreement was observed among these three methods.

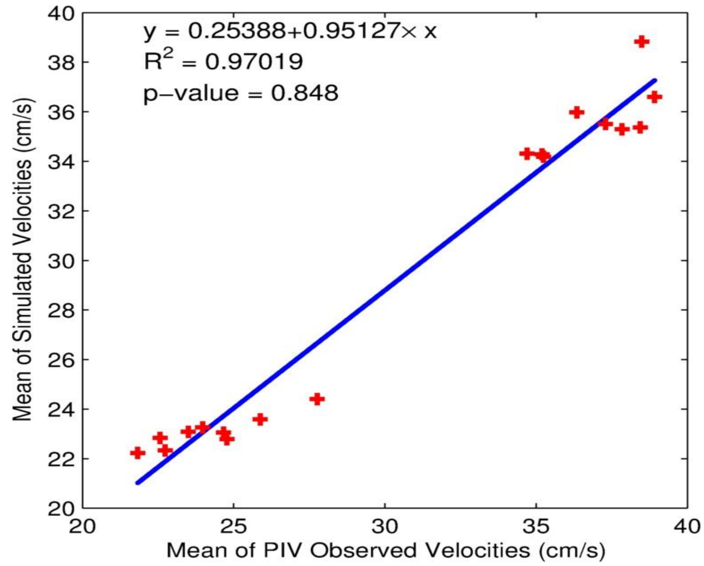
## 2.1. Validation of the CFD models

Particle Image Velocimetry (PIV) and Acoustic Doppler Velocimetry (ADV) were two methods used to obtain the velocity data from the physical modeling. The data from physical modeling provided a reliable means to calibrate and validate the CFD modeling. For each flow condition specified in the test matrix for physical modeling [6], comparisons were made between velocity data from CFD modeling and those from physical modeling. The results of the comparison verified adequacy of the CFD modeling and helped in fine-tuning the models to better simulate the corrugated metal pipe culvert in low flow conditions. A large number of CFD model cases beyond the range of the physical modeling is in progress to extend the ranget of the findings to a greater variety of culvert geometry, including culvert diameters that are too large to test in the laboratory, and flow conditions with a good level of confidence.

The model needs to be validated in order to address all critical issues that potentially affect the accuracy of CFD results. This is an important step that makes the CFD results adhere to the physical phenomena. A series of physical modeling and CFD modeling tests are conducted for validation under various flow conditions. A total test set with combinations of three water depths, two flow velocities, and three bed elevations produce an eighteen-case test matrix. For each case, the PIV measurements can be compared with CFD simulated results. The data sets can be presented as $(u_i, v_i)$, where i = 1, 2, 3, …, 18. $u_i$ is the average velocity of PIV measurements; $v_i$ is the averaged velocity from CFD simulation. A linear regression yields a best-fit equation as shown in Equation 2.1 and Figure 2.1:

$$v = 0.25388 + 0.95127u \qquad (2.1)$$

The intercept is close to zero and the coefficient is close to one. The coefficient of determination, $R^2$, describes how well the CFD modeling fits the PIV observations. Limited by the relatively small measurable area in the culvert cross section (especially in shallow conditions), the ADV measurements do not compare very well with the average velocity from CFD results.



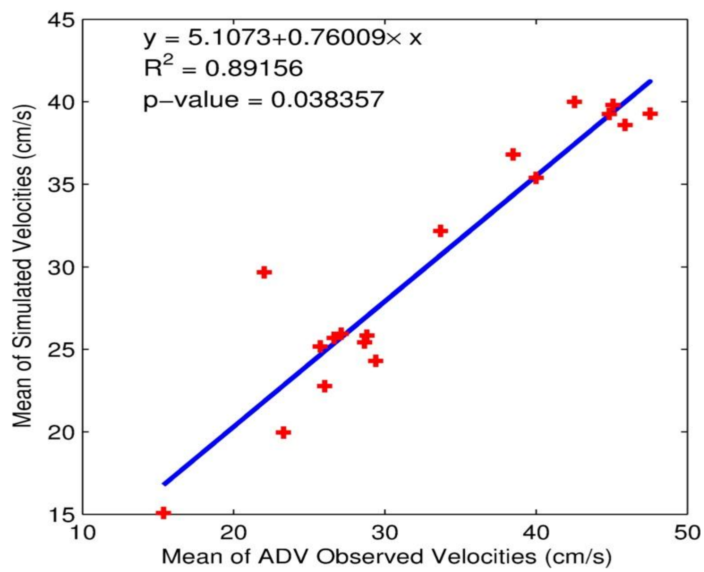




**Figure 2.1 CFD simulated vs. PIV observed mean velocities (upper) and CFD simulated vs. ADV observed mean velocities (lower)**

## 2.2. Calibration of Scaled Models

Full-scale physical modeling is often very costly, if not impossible. Therefore, conducting scaled model experiments in the laboratory to reveal the principle properties of the flow field and resolve the hydraulic problems is a very important tool. A model is a representation of a system that may be used to predict the behavior of a real-world system in some desired respect. The physical system for which the prediction is to be made is called the prototype (Young, D.F. et al., 2007). Usually a model used in bridge hydraulics study is much smaller than the prototype. The similitude theory is used to predict the prototype's performance from model observations. The similitude theory used in hydraulics study is based on the hypothesis that functional relationships exist among the non-dimensional parameters describing a physical system. The functions themselves must be determined empirically (Randall, 2012). Geometric similarity, kinematic similarity, and dynamic similarity are considered in the design of physical models.

One of the important parameter is the Froude Number. It is a dimensionless number defined as the ratio of a characteristic velocity to gravitational wave velocity. In open channel flow, the Froude Number is sometimes used in relationship with the resistance of a partially submerged object moving through water, and permits the comparison of objects of different scale. The Froude Number is defined as:

$$F_r = \frac{V}{\sqrt{gL}} \qquad \textbf{2.2}$$

where $v$ is a characteristic velocity, g is the acceleration of gravity, and $L$ is a length. The Froude Number is named in honor of William Froude (1810-1879), a British civil engineer, mathematician and naval architect. For the problems involving free-surface flows, both gravitational and inertial forces (represented by velocity) are important and Froude Number becomes an important similarity parameter. Therefore, Froude number similitude is usually required for models involving open channel flow.

In section 2.1, the experimental data from the physical modeling are used to validate the CFD models' accuracy and sensitivity to scale. The sensitivity to parameters such as shape, size and corrugation dimensions is tested in this section. Since the flow is open channel flow with relatively large values of roughness, it is in a fully turbulent condition. Froude Number similitude criterion is used to correlate flume tests (scale model) and field conditions (prototype). That is, the hydraulic condition in the field is considered to be similar to that of the model tests if both flows have same Froude number. For example, if a 24 inch diameter culvert is used for modeling a 96 inch structural plate pipe culvert, the linear scale ratio of prototype to model is 4. The velocity scale ratio then becomes the square root of 4, i.e. 2. Using this approach, the velocity of 3 ft/s measured in the laboratory flume corresponds to a flow velocity of 6 ft/s for the 96 in structural plate pipe culvert.

A length scale of 1:2 is used to study the scale effect. The 36 in diameter laboratory scale model culvert and a larger 72 in diameter pipe are both produced with Pro-Engineer for CFD simulation. The laboratory model culvert corrugation is 1 inch high, with spacing of 3 inches. Correspondingly the larger size corrugation (72-inch pipe) is 2 inches high, with spacing of 6 inches. The bed elevations, test water depths and river bed gravel size are all in a scale of 1:2. The flow velocities in the lab model testing are 0.71 fps and 1.1 fps which corresponds with 1 fps and 1.556 fps for the 72-inch pipe.

In the large-scale CFD modeling, the material physics, the boundary conditions and the meshing methods are kept the same as those used with lab scale model. If the base size of the meshing cells remains the same as that used in the lab-scale (36-inch) model, the required computation effort increases significantly for the large scale (72-inch) model. A parametric study is conducted to identify the optimum base size of the mesh and the refinement needed in the corrugated section (Venkata, 2011).

The depth-averaged velocity is an important characteristic velocity in this project. The local depth-averaged velocity V and the local depth y of the flow at any point in the culvert cross-section can be determined by the procedure illustrated in Figure 2.2. In the numerical modeling, the culvert cross-section is divided into evenly spaced strips, and then the discharge and area in each strip are determined by integrating over the strip. The ratio between the integrated discharge and area is the depth-averaged velocity.

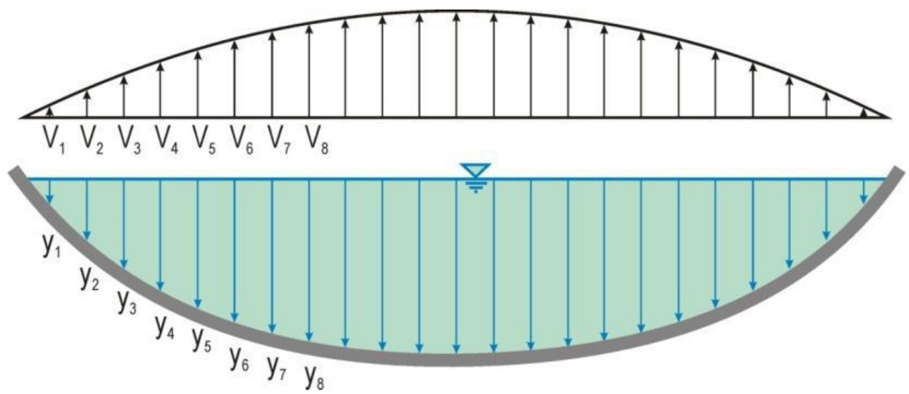



**Figure 2.2 Variation of flow velocity and depth in a cross-section of a corrugated metal pipe**

The proposed fish passage design is based on the local depth-averaged velocity curve. One end of the path is defined by the culvert wall. The other end of the path, towards the center of the culvert, is defined by the point where the local depth-averaged flow velocity V is equal to the maximum fish swimming velocity. The depth-averaged velocity has been normalized by the cross-sectional average velocity. The position of the velocity is defined as the distance to the center of the culvert and normalized by the culvert diameter. Using the normalized position and velocity as the x axis and y axis, respectively, the scaled culvert model and the original laboratory culvert model velocity curves can be plotted in the same figure. Figure 2.3 to Figure 2.5 illustrate the comparison of the normalized local depth-averaged velocity curve of the original model and that of the scale model. The bed elevation varies from 0 in to 10.8 in.

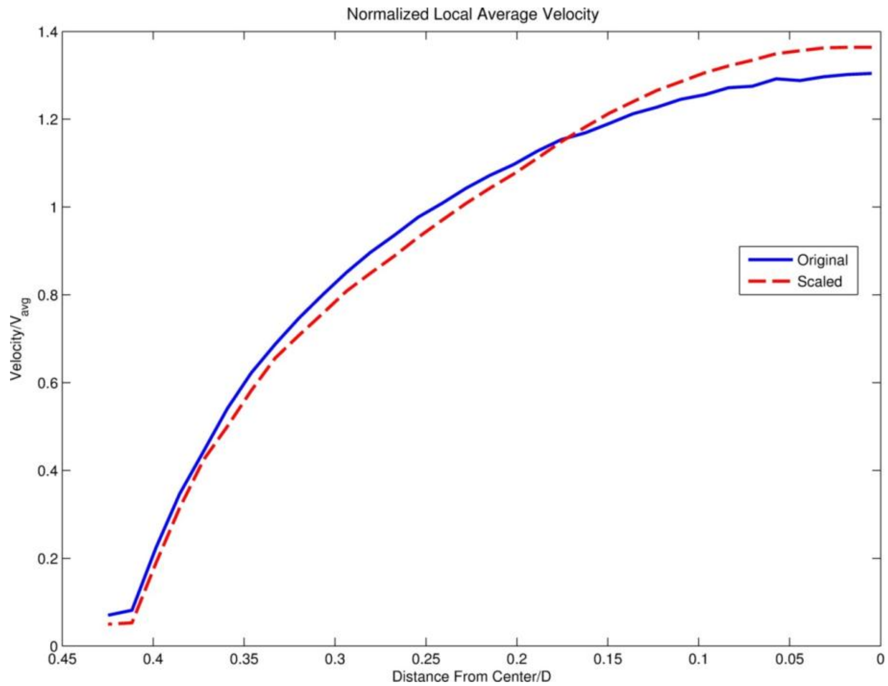



**Figure 2.3 Comparison of original CFD model and scaled CFD model normalized depth-averaged velocity curves at 0 in bed elevation**

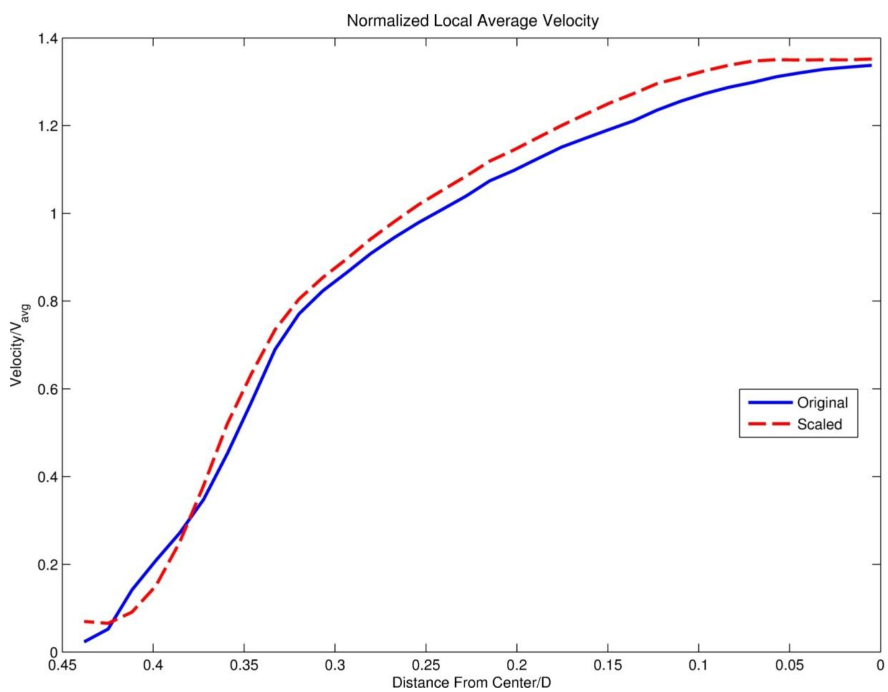



**Figure 2.4 Comparison of original CFD model and scaled CFD model normalized depth-averaged velocity curves at 5.4 in bed elevation**

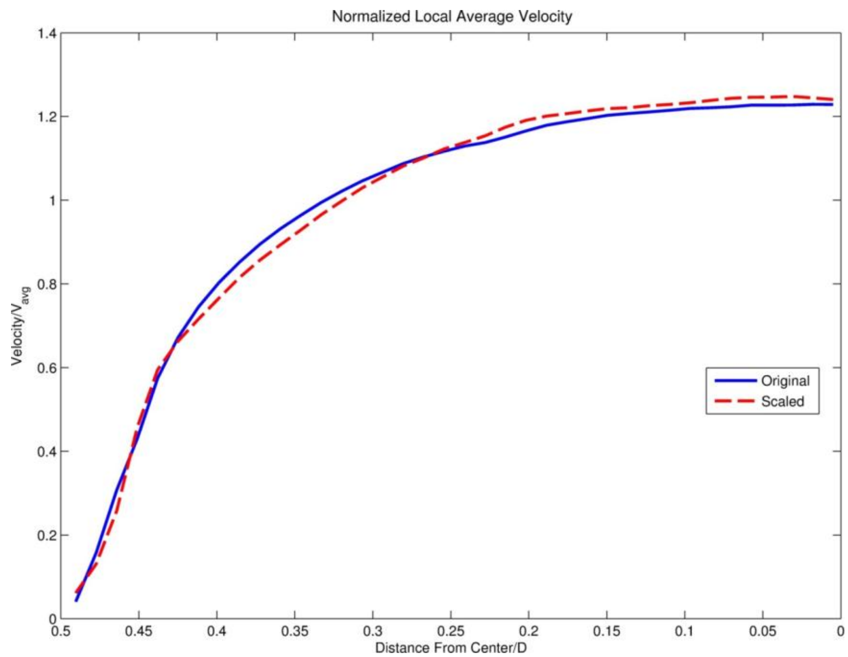



**Figure 2.5 Comparison of original CFD model and scaled CFD model normalized depth-averaged velocity curves at 10.8 in bed elevation**

## 2.3. Application of the Prototype Models

Once the truncated single phase CFD culvert model is validated and calibrated, it is applied to the prototypical size. The prototype culvert diameter is 8 ft, and has the annular corrugations of 2 inches high and 6 inch spacing. The flow depths are still 0.08D, 0.16D and 0.25D, which correspond to 0.64 ft, 1.28 ft, and 2 ft, respectively, for the 8-ft pipe. The velocities are chosen as the fish passage critical velocity 1 fps and 3 fps, and all the flow conditions were tested in the corrugated pipe and pipes with 24 mm $D_{50}$ gravel. The prototype velocity distribution contours are displayed through Figure 2.6 to Figure 2.8. All of the iso-velocity contour plots demonstrate that there are steep velocity gradients close to the bed and walls, which is characteristic of turbulent flow with fully developed boundary layers.

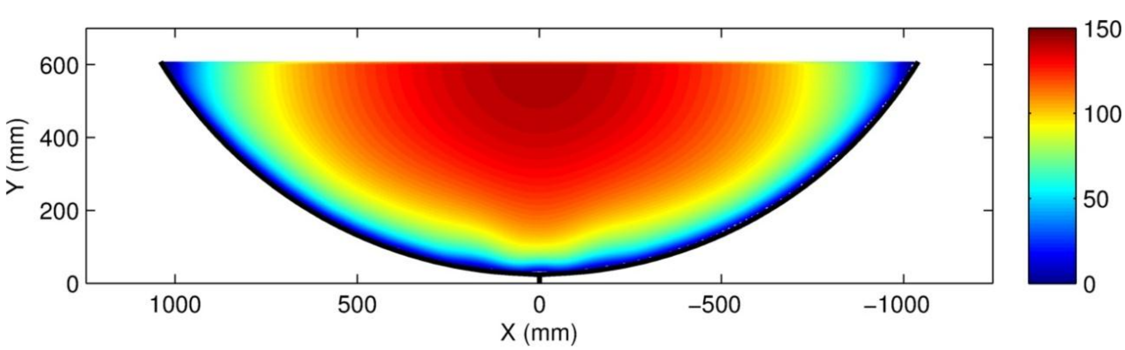



**Figure 2.6 Velocity distribution (cm/s) of the prototypical culvert (trough section) for 2 ft water depth without bed elevation**

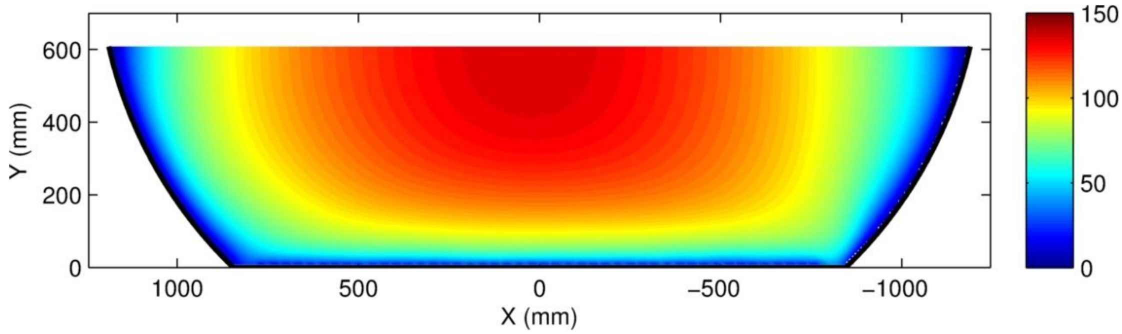



**Figure 2.7 Velocity distribution (cm/s) of the prototypical culvert (trough section) for 2 ft water depth with 0.15D bed elevation**

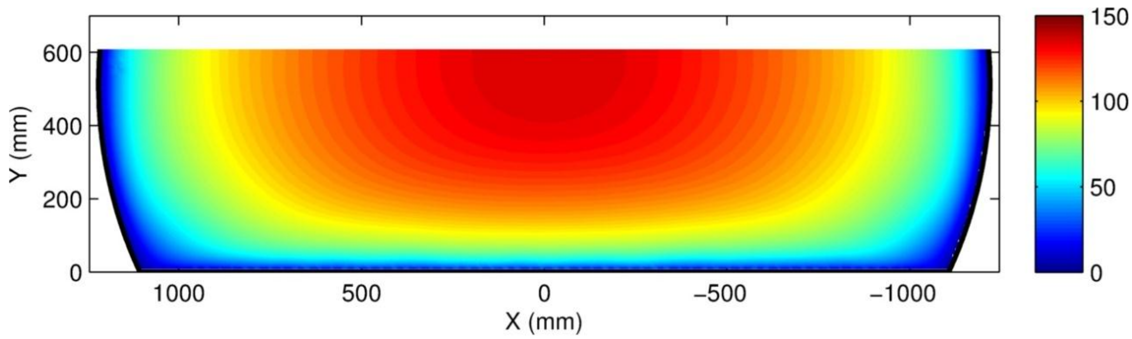



**Figure 2.8 Velocity distribution (cm/s) of the prototypical culvert (trough section) for 2 ft water depth with 0.3D bed elevation**

## 2.4. Design Aids Based on the Velocity Contour Results

The validated truncated periodic boundary single phase CFD model provides the velocity distribution contour under a variety of flow conditions reliably and cost-efficiently. And the velocity of each numerical calculation grid represents the local velocity when the CFD model mesh is fine enough. The available velocity data read from the Star-CCM+ tabulated data is adequate to do the velocity analysis and extrapolation.

In this study, a trimmed cell mesh was used to generate an extremely high quality hexahedral-based mesh for the culvert geometry. This kind of mesh model gives a structured mesh across the culvert section that is well suited for computing strip averages because the grid can be built to align cell faces with strip boundaries. When these boundaries are not aligned with cell faces, velocity data in uniform vertical strips in the cross-section are generated by interpolating the original grid velocity data from STAR-CCM+. The exported data are then used to obtain the depth averaged velocity distribution over the strips by averaging the velocities falling in each strip (using MATLAB). Table 2.1 shows the current recommendations of the design standards for fish passage in Maryland.

**Table 2.1 General design requirements for fish passage in the state of Maryland**

| | Flow Velocity | Minimum Flow Depth |
|---|---|---|
| Non-trout streams | Up to 1 fps | 4 to 6 in |
| Trout streams | Up to 3 fps | 12 in |

The next step is to define the path in the culvert available for fish passage for the given conditions. In Figure 2.9, one end of the path is defined by the culvert wall. The other end of the path, towards the center of the culvert, is defined by the point where the local depth-averaged flow velocity V is equal to the maximum fish swimming velocity, $V_F$, as defined by the appropriate standard. In this illustration example $V_8 = V_F$. Note that another limit to the fish path can be the water depth when the depth in a strip is less than that required for larger species of fish, such as trout or salmon.

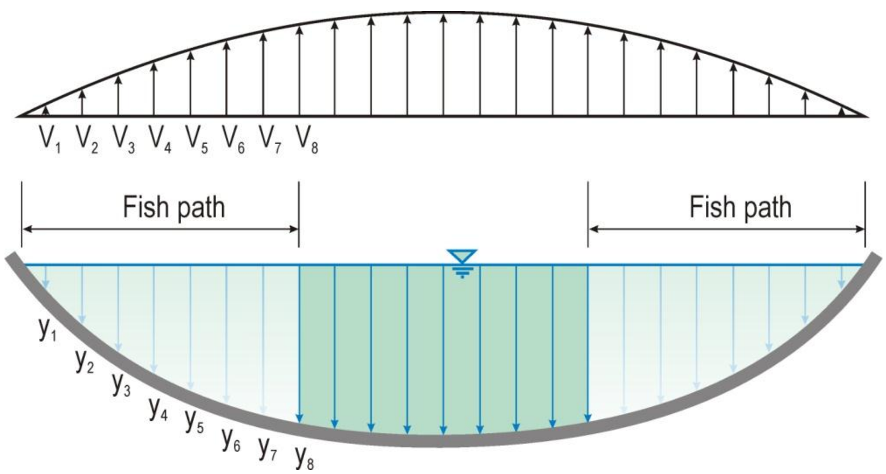
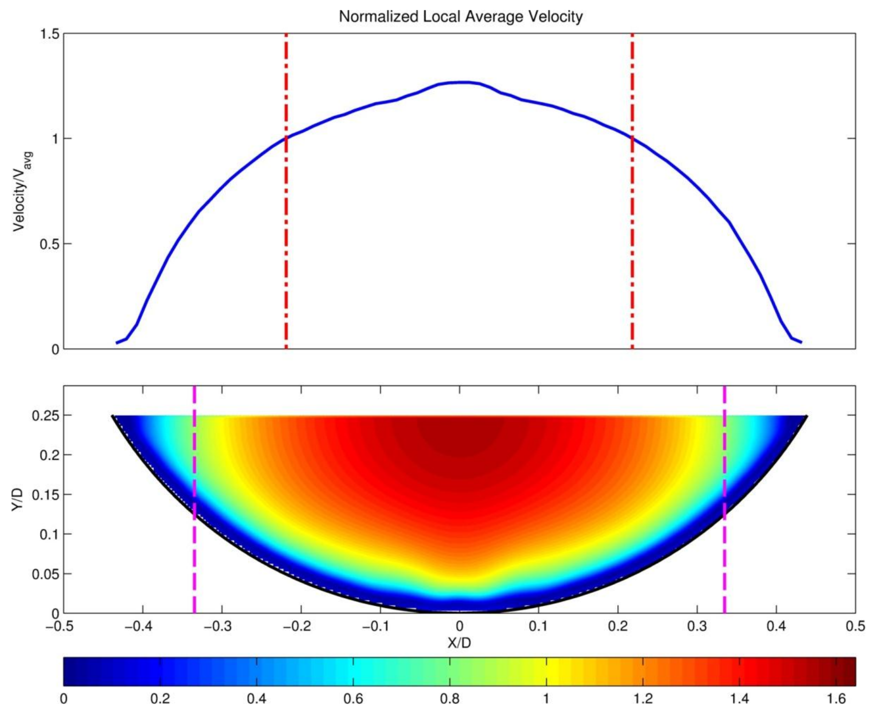
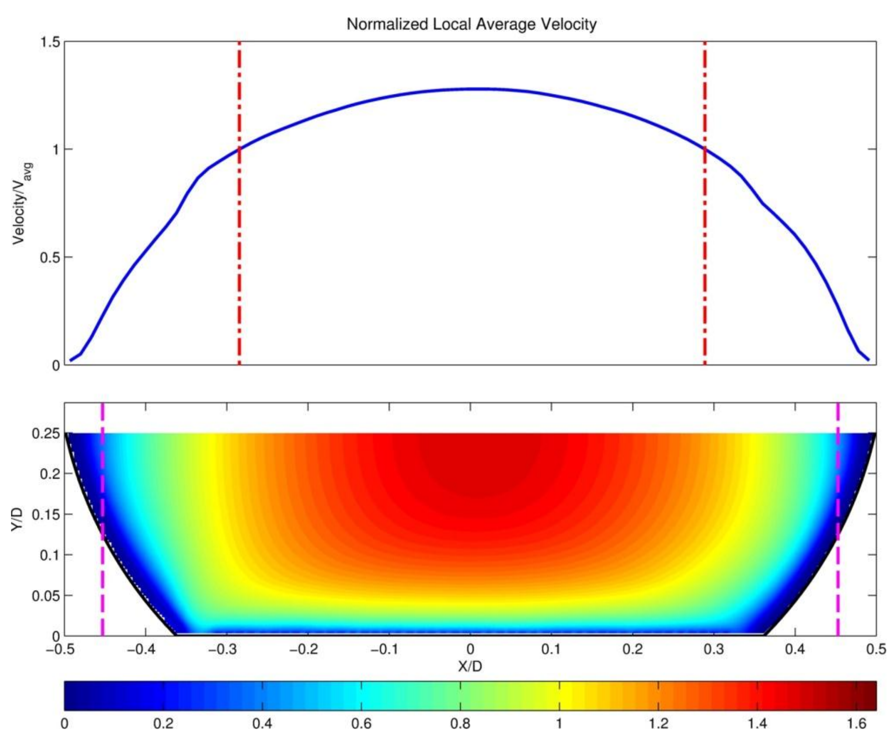



**Figure 2.9 Flow path for the selected fish design criteria of velocity and depth**

Using the above-mentioned method, the fish path in this project culvert condition is defined. Based on the prototype CFD simulation data, the local depth-averaged velocity curve was created. Take trout as the design objective (see Figure 2.10), the red dot-dashed lines illustrate the maximum velocity of 3 ft/s ($V/V_{avg}$=1), and the purple dashed lines indicate the minimum flow depth of 12 in required by trout. The area between the two curves is suitable for the swimming capacity of trout, which qualifies the flow condition for fish passage. The upper figure (Figure 2.10a) is the condition without gravel bed, while the lower figure (Figure 2.10b) is the condition with 0.15D bed elevation. If the purple dashed lines are between the red dot-dashed lines, the flow condition does not qualify for fish passage.




(a) Without gravel bed




(b) With 0.15D bed elevation

**Figure 2.10 Define fish path based on the combination of dimensionless water depth and flow velocity**

## 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. Young, D. F., Munson, B. R., Okiishi, T. H., and Huebsch, W. W. (2007). “A brief introduction to fluid mechanics: fourth edition.” John Wiley and Sons, Inc.
8. Randall, D. (2012). “Dimensional Analysis, Scale Analysis, and Similarity Theories”, Revised February 13, 2012.
9. Venkata, S. L. (2011). “Computational fluid dynamics modeling of flow through culverts”, Master Thesis, Northern Illinois University.
10. CD-adapco, *User Guide STAR-CCM+ Version 6.04.014*, 2011.
11. http://www.ptc.com/support/proengineer.htm
12. http://en.wikipedia.org/wiki/Froude_number