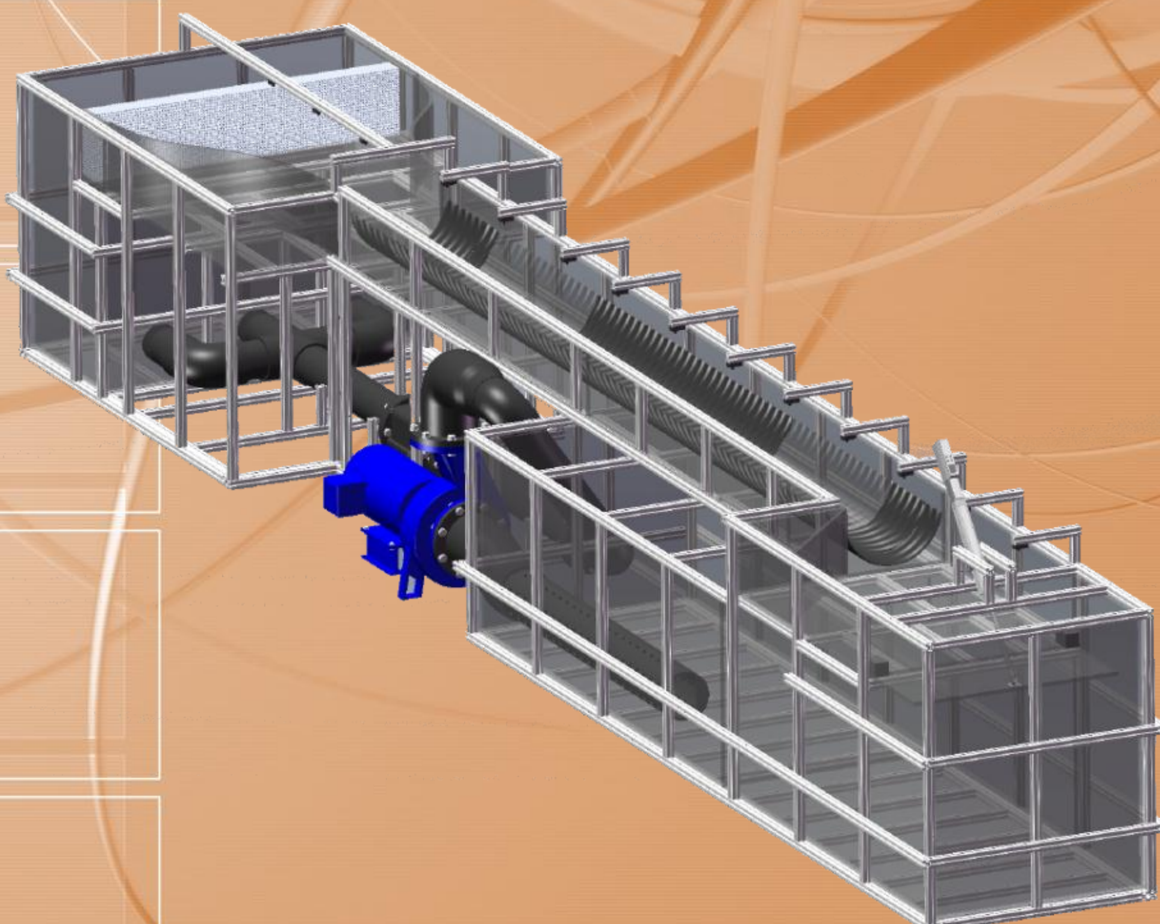
Computational Mechanics Research and Support for Aerodynamics and Hydraulics at **TFHRC**



Culvert Analysis Quarterly Report

January through March 2012

Computational Fluid Dynamics Modeling of Flow through Culverts
2012 Quarter 2 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

April 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 January through March 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 part 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 reliable means in calibrating and validating 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 modeling beyond the range of the physical modeling is in progress to extend the impact of the findings to a greater variety of culvert geometry and flow conditions with good confidence.

2.1.1. Comparison of CFD results with experimental data

The hydraulic flume used for testing culverts in the fish passage study had a width the same as the radius of the selected culvert pipe. It was therefore possible to fit an entire quarter of the pipe into the flume widthwise. The quarter-pipe setup allowed optimal visibility to the flow through the translucent flume wall for the access of laser light sheet and camera that were required by PIV.

The primary validation effort consisted of a comparison of model predictions of velocity distribution from the STAR-CCM+ software against experimental data under various average velocities, flow depths, and gravel bed elevations. Analyses were conducted to quantify discrepancies between CFD output and experimentally measured values, and to assess how these discrepancies affect the qualification of a culvert as fish passable.

As mentioned in previous reports, test scenarios performed in the physical modeling included three different water depths, two velocities, and three bed elevations. CFD models for the calibration process were created precisely following the geometry of the physical models. Single-phase models with cyclic boundary conditions were used. Validation work presented in previous reports showed good agreement between uniform flow results from this highly efficient approach and those from time-consuming full-barrel VOF modeling. Table 2.1 shows the types of boundary conditions specified in the CFD modeling:

Table 2.1: Boundary conditions

boundary	name	type
Face at minimum z (flow direction) value	Inlet	Velocity inlet
Face at maximum z (flow direction) value	Outlet	Pressure outlet
Top of the bounding box	Top	Symmetry plane
Centerline face	Center	No-slip wall
Select all the other faces	Barrel	No-slip wall

Special attention was given to the centerline face. In order to obtain better agreement with the physical model, the centerline face boundary type was set to be a non-slip wall in the quarter culvert models, which imitated the zero velocity at the sidewall of the flume. However it should be changed to symmetry plane in the extended simulation for full size culvert models because the non-slip wall conditions would not exist in a real pipe. Symmetry plane indicates a surface where normal velocity and normal gradient of in-plane velocity are both zero. The effect of the difference between boundary conditions used in the quarter culvert model and those used in full size culvert model will be identified when the extended CFD simulations on full-scale pipes are complete.

Bed elevation is defined as the depth of the culvert that is buried under the gravel bed. The illustration of the comparison between CFD results and experimental data is organized into three sections based on three different bed elevations of 0 inch, 5.4 inch (0.15D, D is the pipe diameter.) and 10.8 inch (0.3D). For each case, two velocities and three flow depths are used. The accuracy of analyses and the sources of error are discussed for each section.

The development of the CFD models from the VOF multi-phase model to the truncated single phase model with cyclic boundary was presented in the previous report. With the premise of the uniform flow, the VOF multi-phase model can be replaced by the single-phase model. As discussed previously, the small increase in water velocity from the single phase model was conservative for the analysis to determine if the flow permits fish passage, and the general velocity distributions were similar between the two approaches. Furthermore, the truncated single phase model with cyclic boundary could provide the same velocity result as single-phase model without tilting and flap gate. Given the large amount of tests in the test matrix, the more efficient truncated single phase model with cyclic boundary was the ideal choice. Meanwhile, finer mesh could be utilized in the shorter model to improve accuracy of the CFD model. The discussion in following sections on the validation of CFD modeling are based on the results from the truncated single phase approach using cyclic boundary conditions.

The STAR-CCM+ models were validated against two independent experimental velocity data sets: velocities measured by ADV and those captured by PIV. There was a significant area near the walls that the ADV probe cannot reliably measure velocity. Although this made the amount of useful ADV data in shallow flow conditions very limited, the ADV measurements still served the purpose as a cross check on the PIV data very well. CFD data results cover the entire flow cross section. Depending upon the relative depth and the bed elevation, the number of mesh cells varied between 58661 and 875087. Meanwhile, more data points were taken near the boundary of the culvert than in the center of the water body in order to obtain more precise flow field data near the corrugated wall. The results showed that the velocity vector was mainly in the flow direction (z-direction) with small components of flow in the x-direction and y-direction. The results were plotted in color-coded contour. Figure 2.2 through Figure 2.22 compare the data from CFD, PIV, and ADV for 3 water depths, 3 sediment elevations, and 2 velocities. These contour plots provide visual evidence in the agreement between CFD simulations and experiments. All figures compare CFD against PIV except Figure 2.8, which compares CFD against ADV. . Table 2.2 shows the flow conditions for each plotting that compares CFD to PIV and CFD to ADV. With a broad band of area near the walls that has no data, the ADV presents sizable contour plot area only when flow depth is 9 inch. The ADV contour plot is a supplementary tool to cross-verify the accuracy of the PIV measurements.

Table 2.2 Contour plots comparing CFD, PIV, and ADV. All figures compare CFD against PIV except Figure 2.8, which compares CFD against ADV.

Velocity	1.1'/s			0.71'/s		
Flow Depth	4.5"	6"	9"	4.5"	6"	9"
Sediment elevation						
0 D	Figure 2.2	Figure 2.3	Figure 2.4 Figure 2.8	Figure 2.5	Figure 2.6	Figure 2.7
0.15 D	Figure 2.10	Figure 2.11	Figure 2.12	Figure 2.13	Figure 2.14	Figure 2.15
0.3 D	Figure 2.17	Figure 2.18	Figure 2.19	Figure 2.20	Figure 2.21	Figure 2.22

(1) Bed elevation at 0 inch

Figure 2.1 shows the experimental model (left) and the Computer Aided Design (CAD) model of culvert section geometry for the use in truncated single-phase modeling (right). The cross section of the pipe at the crest of the corrugation is different from that at the trough of the corrugation. The results shown in Figure 2.2 through Figure 2.7 are taken from a trough section, i.e. the largest cross section.

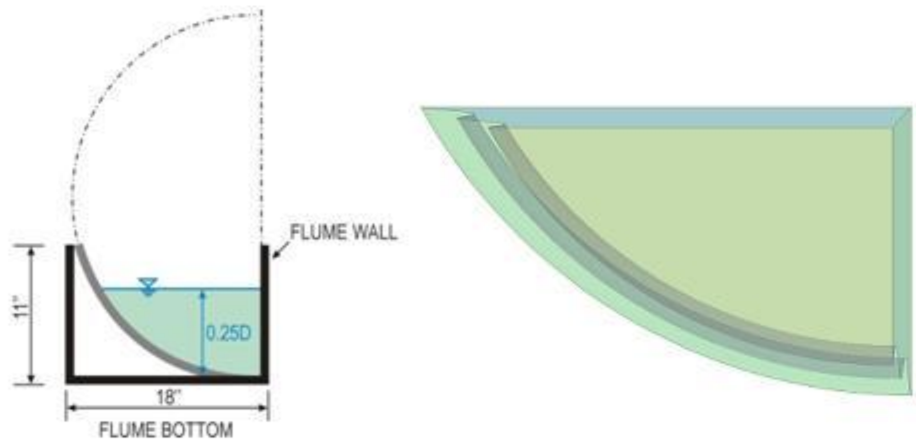


Figure 2.1 Sketch of experimental model (left) and CAD model of culvert section (right) for bed elevation at 0 inch

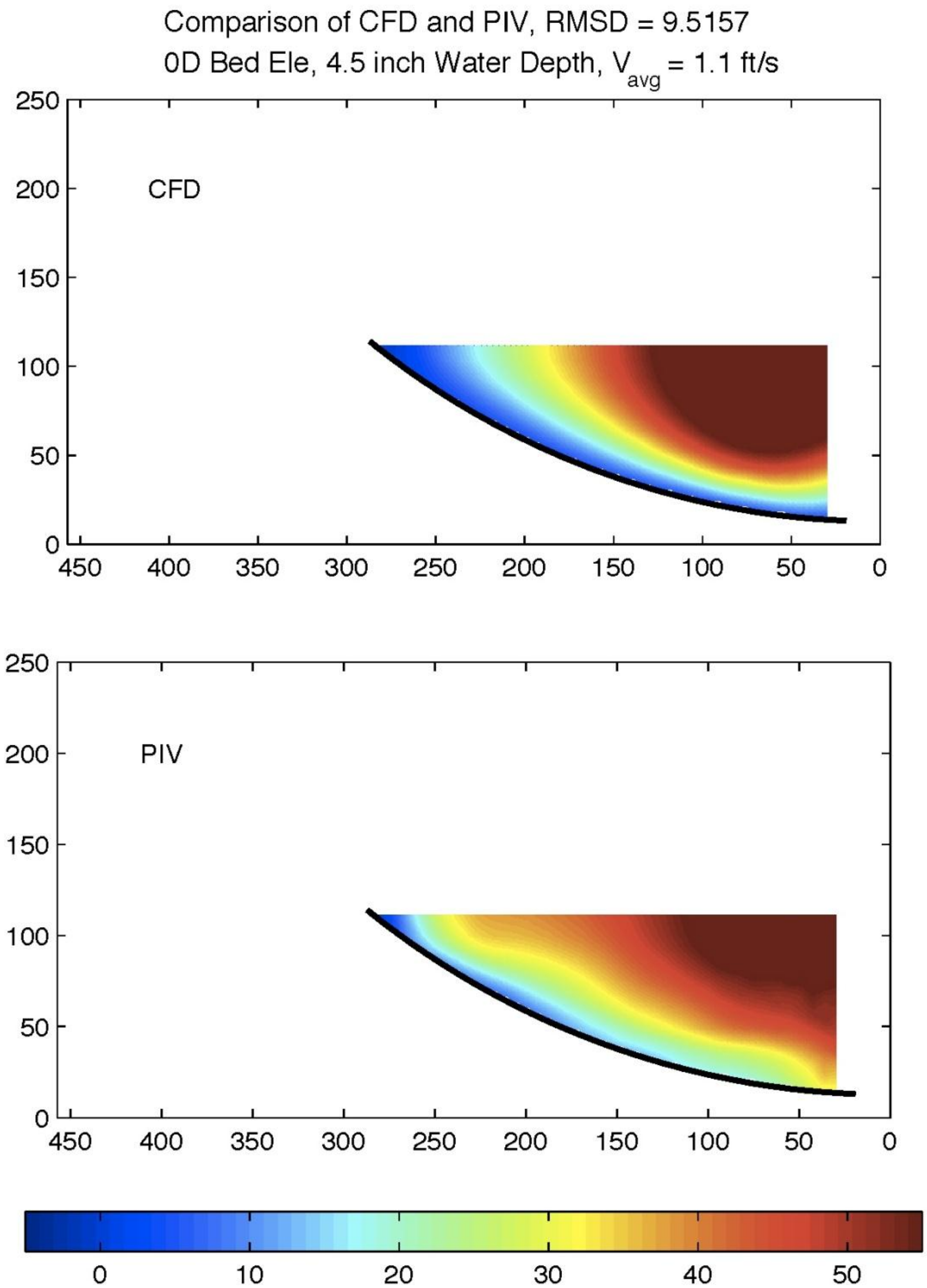


Figure 2.2 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 1.1 fps for 4.5inch water depth (velocity: 33.5 cm/s)

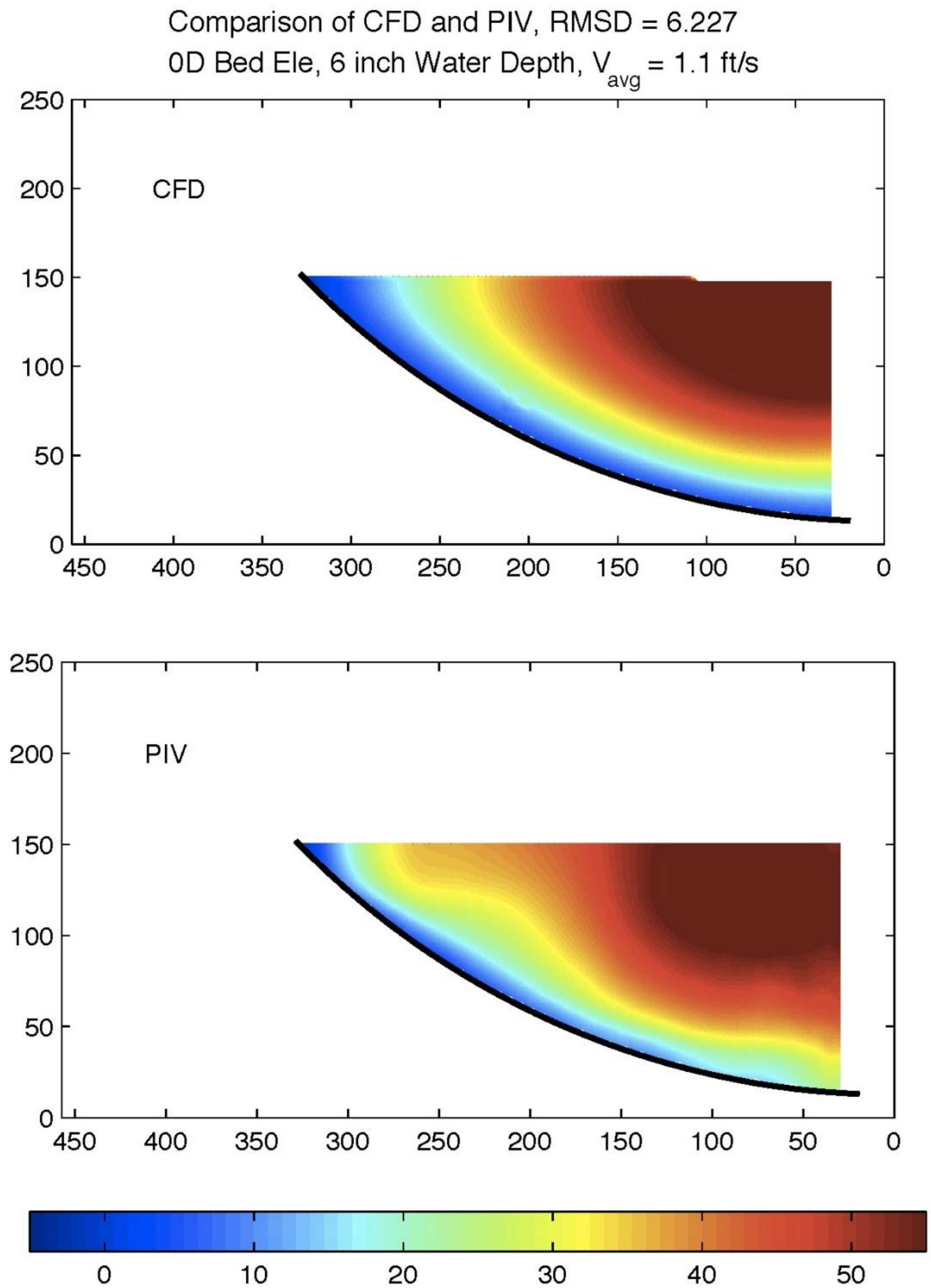


Figure 2.3 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 1.1 fps for 6 inch water depth (velocity: 33.5 cm/s)

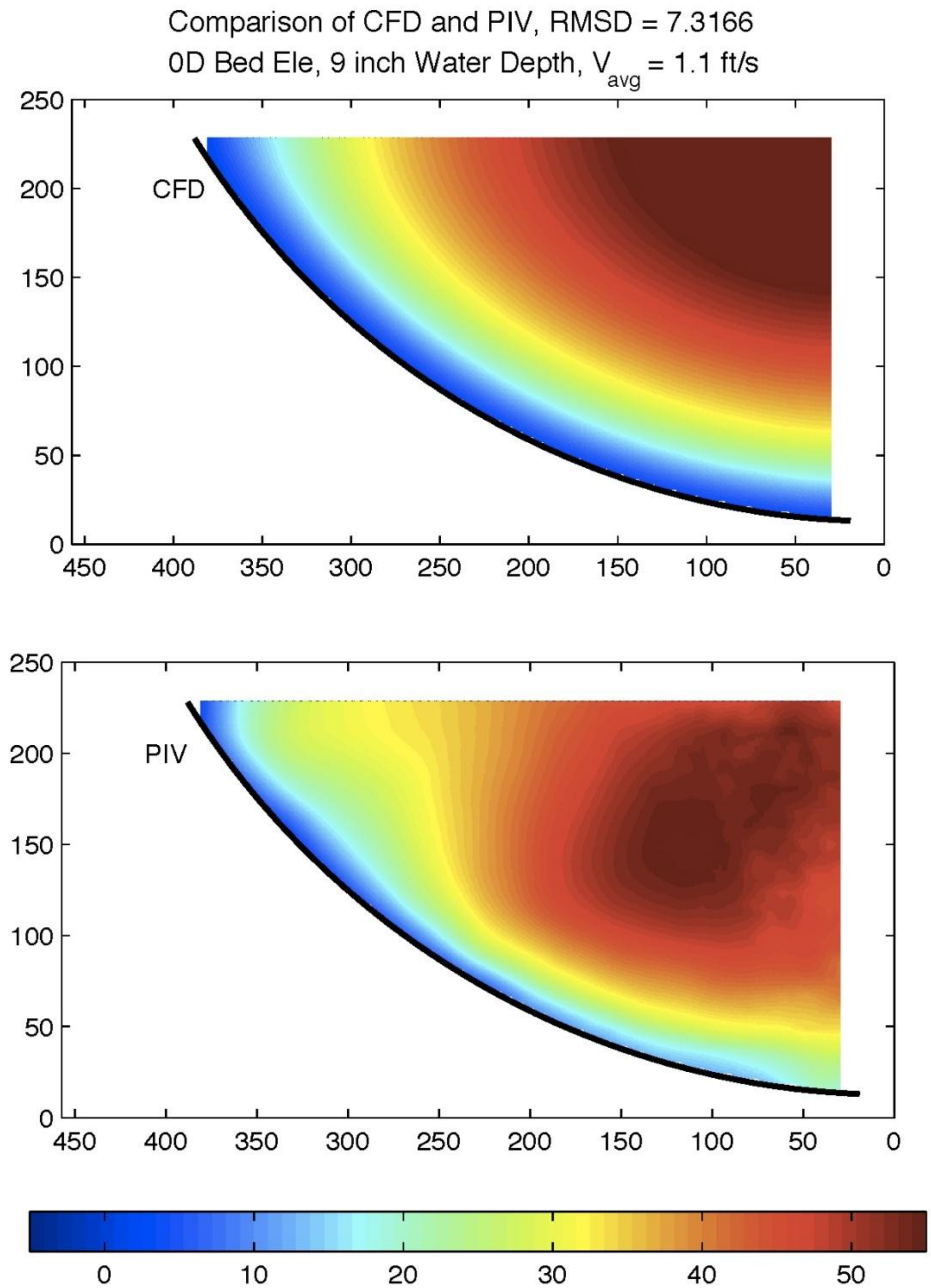


Figure 2.4 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 1.1 fps for 9 inch water depth (velocity: 33.5 cm/s)

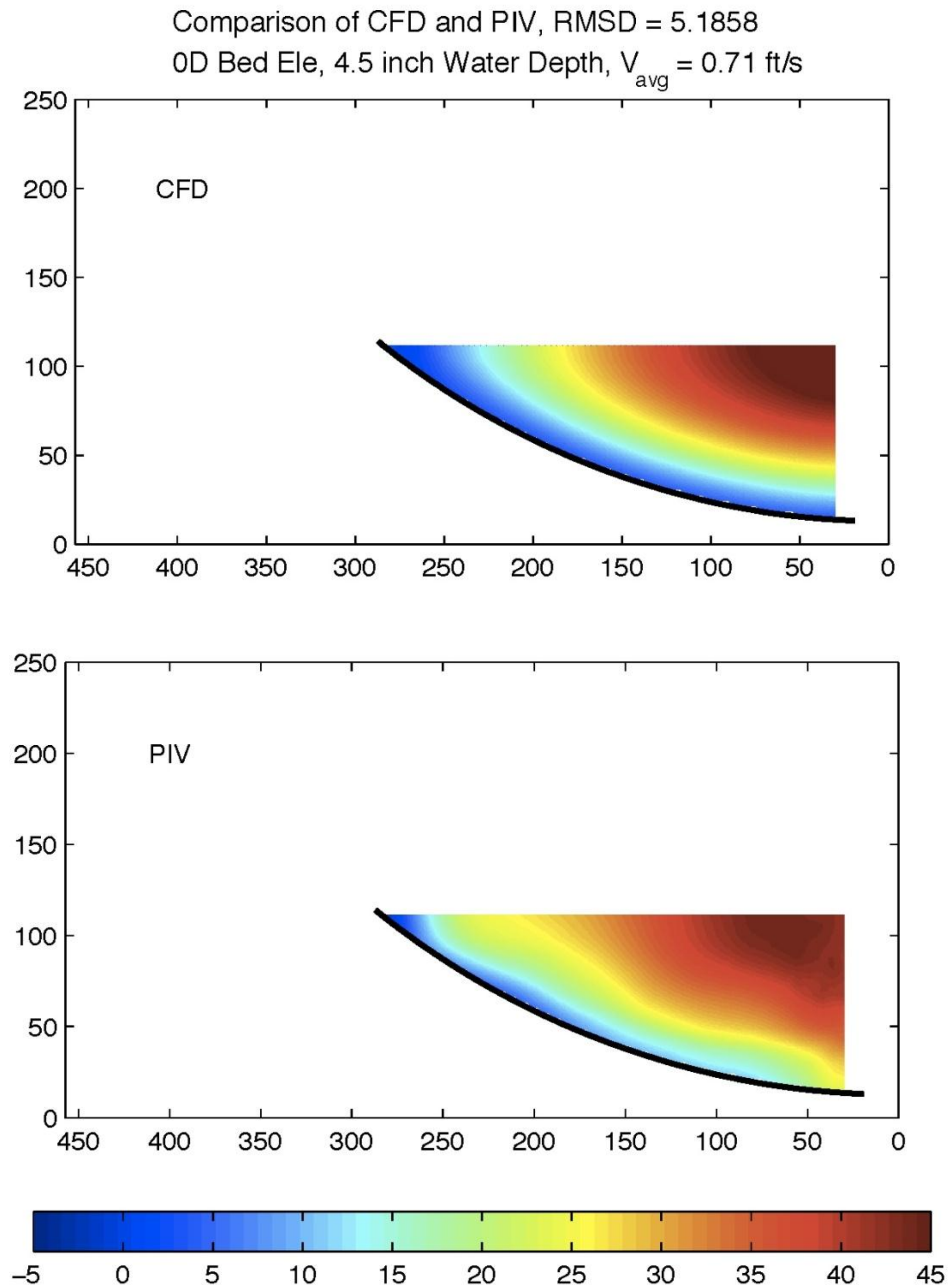


Figure 2.5 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 0.71 fps for 4.5 inch water depth (velocity: 21.6 cm/s)

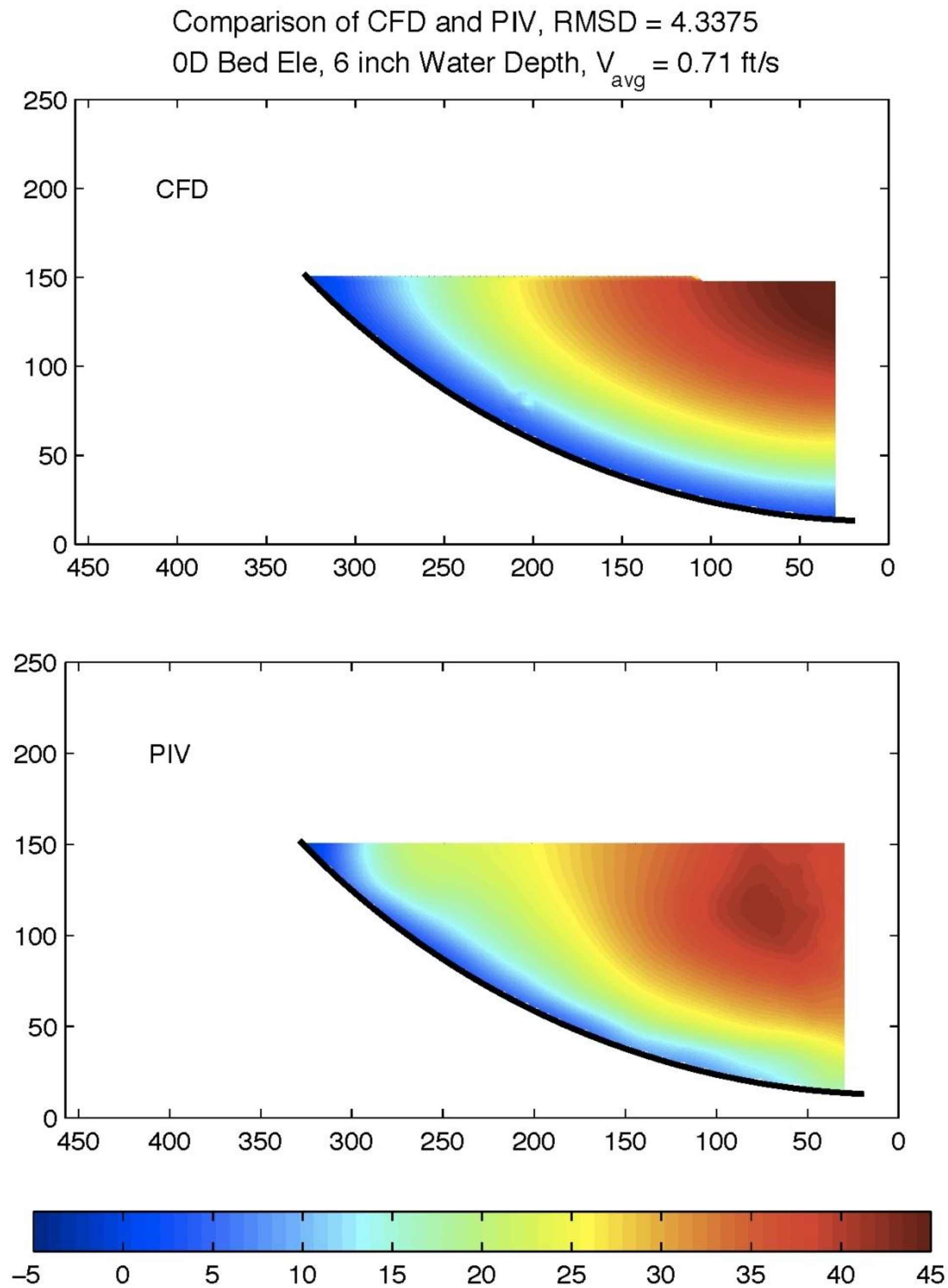


Figure 2.6 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 0.71 fps for 6 inch water depth (velocity:21.6 cm/s)

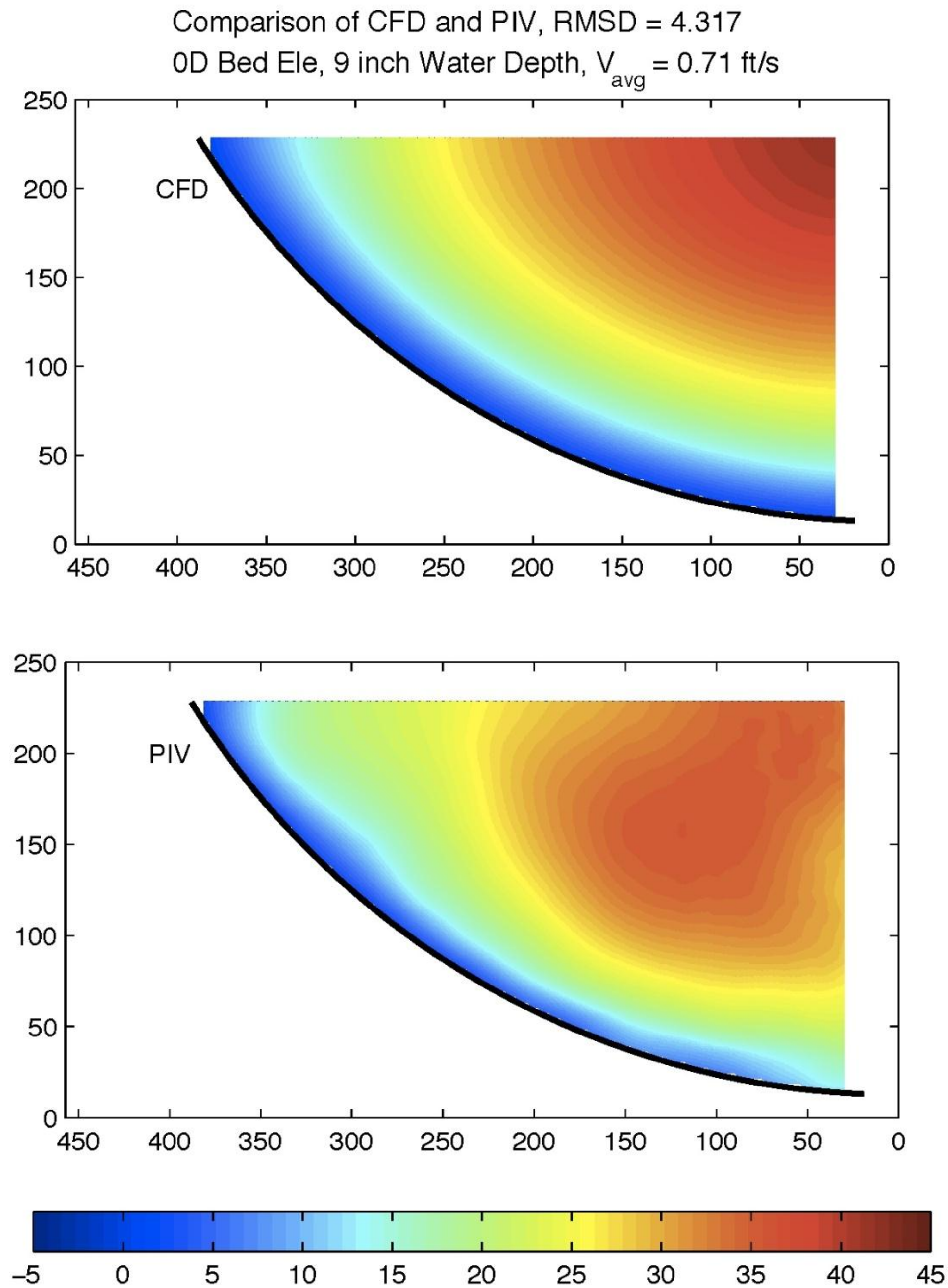


Figure 2.7 Comparison of CFD and PIV velocity contour under the condition of 0D bed elevation and 0.71 fps for 9 inch water depth (velocity: 21.6 cm/s)

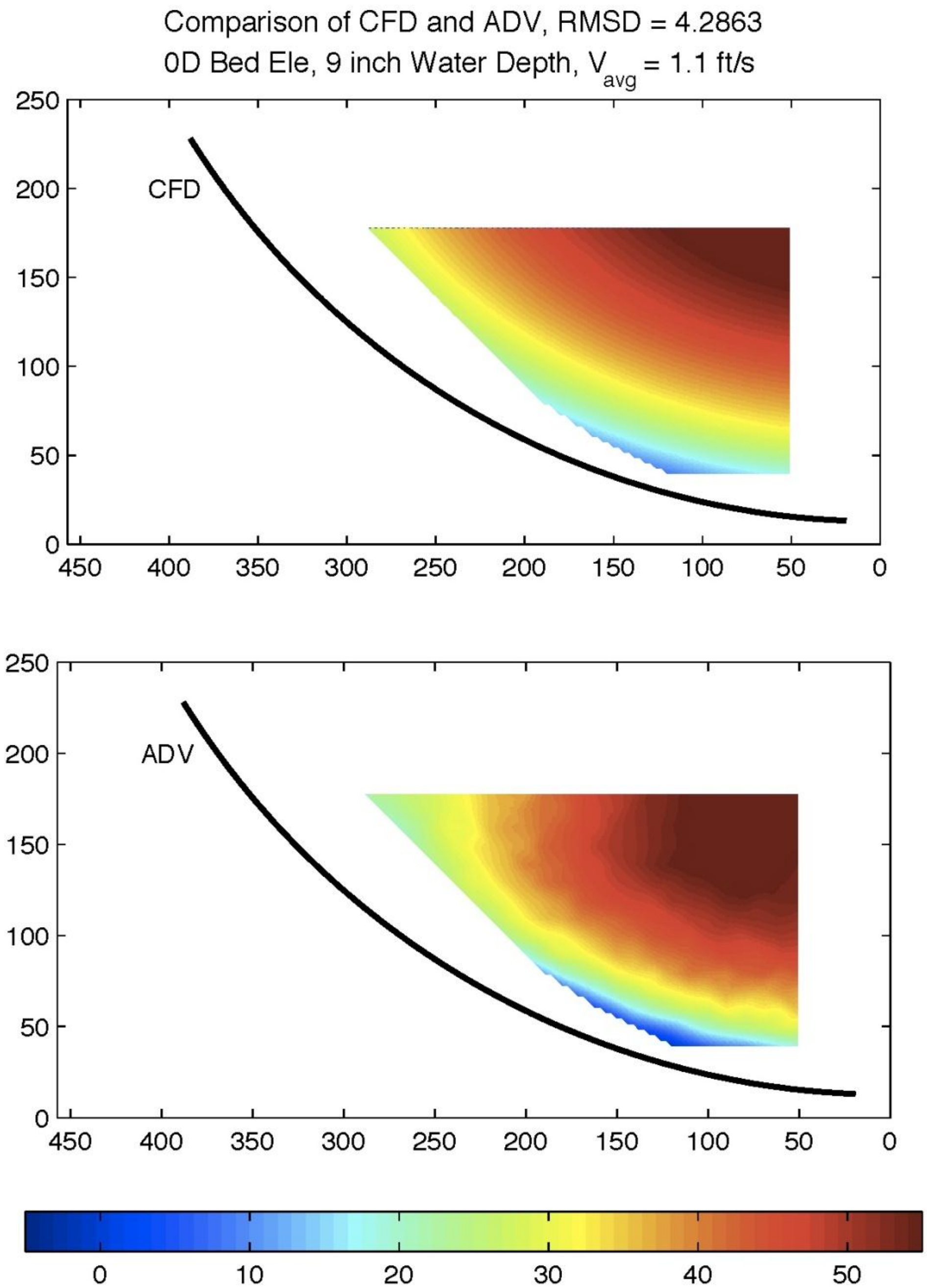


Figure 2.8 Comparison of CFD and ADV velocity contour under the condition of 0D bed elevation and 1.1 fps for 9 inch water depth (velocity: 33.5 cm/s)

(2) Bed elevation at 5.4 inch

The variation of bed elevation is an important and unique consideration in this study. Ideally, a gravel bed exhibits two special characteristics: (1) An elevated boundary that changes the geometry of the channel and roughness of the boundary. (2) A permeable material in the gravel-occupied area that allows relatively low velocity flow and significant energy dissipation. In this stage of the study, the effect of (2) is neglected. Although this does not perfectly simulate the field sediment condition, it is more consistent with the lab test setup, for which a single layer of gravel is laid onto the solid flume bed to represent the roughness of the gravel bed. Figure 2.9 shows the sketch of the experimental model (left) and the CAD model of culvert section geometry (right). The dimples shown in the CAD model were created by a 2-D periodical function that yields a similar roughness as that of natural bed with specified gravel size.

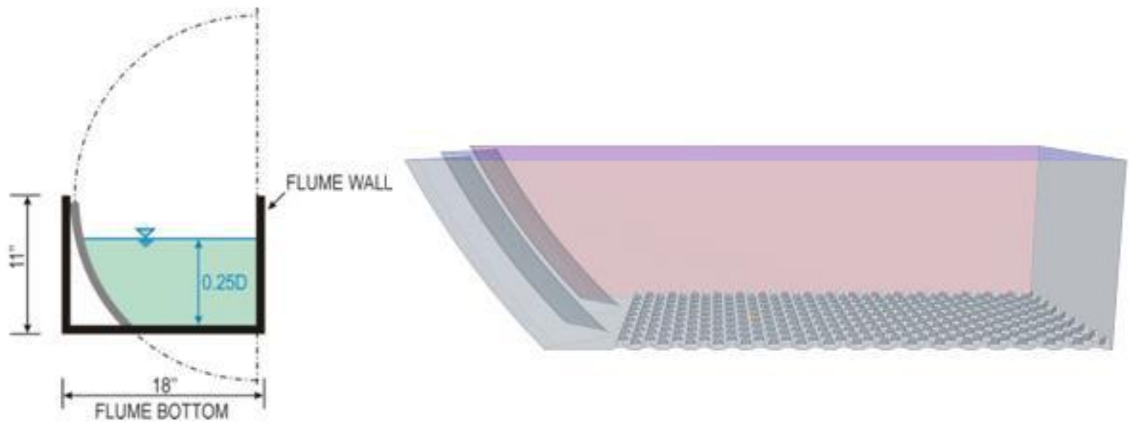


Figure 2.9 Sketch of experimental model (left) and CAD model of culvert section (right) under the situation of bed elevation at 5.4 inch

The CFD simulations for clean culvert pipes described in the previous section were repeated on the model shown in Figure 2.9. Results were compared in Figure 2.10 through Figure 2.12. Similarly, the trough cross-section (the largest cross-sectional area) was used for the comparison between CFD and PIV. Detailed parameters are given in Table 2.2.

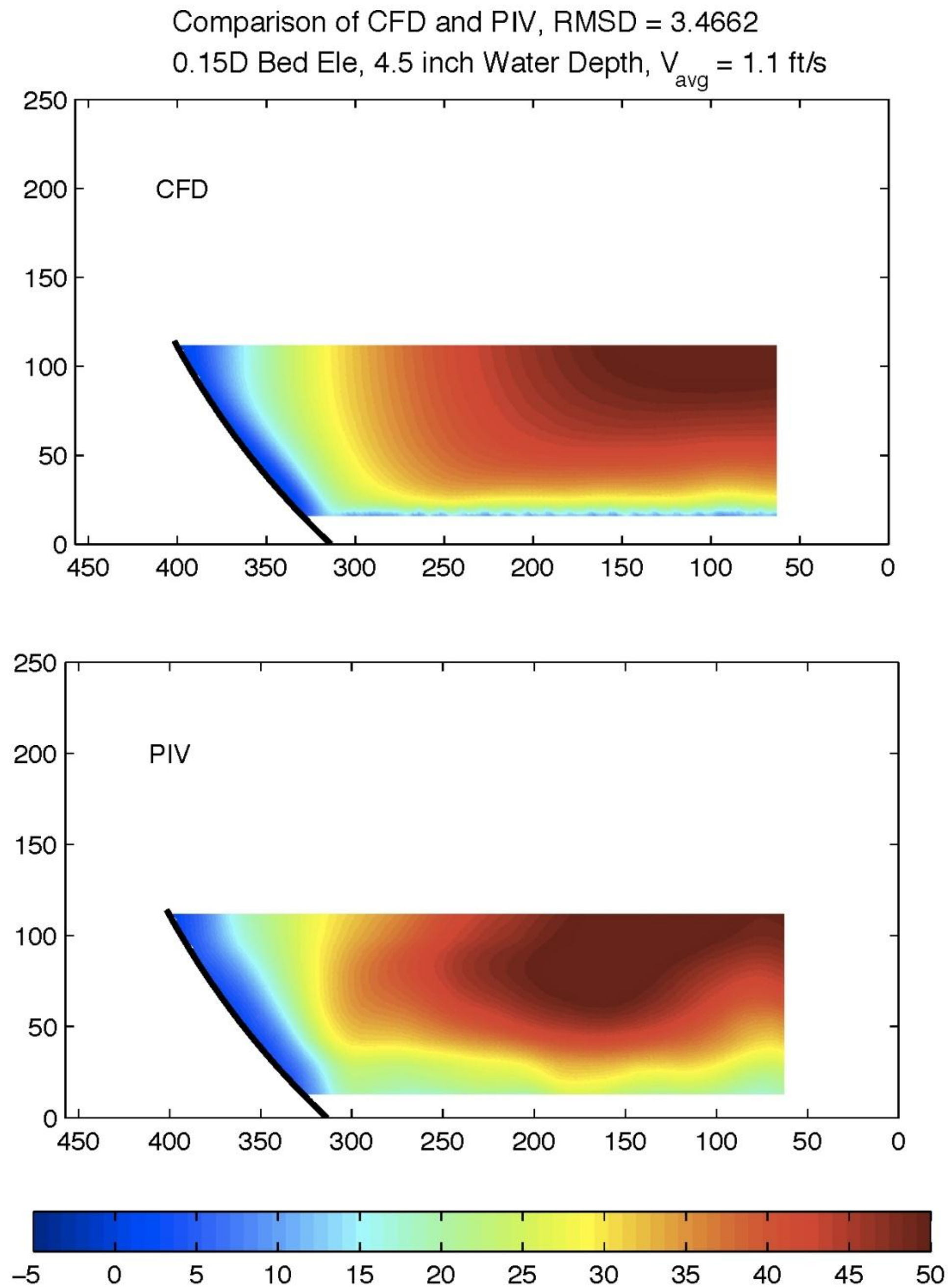


Figure 2.10 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 1.1 fps for 4.5 inch water depth (velocity: 33.5 cm/s)

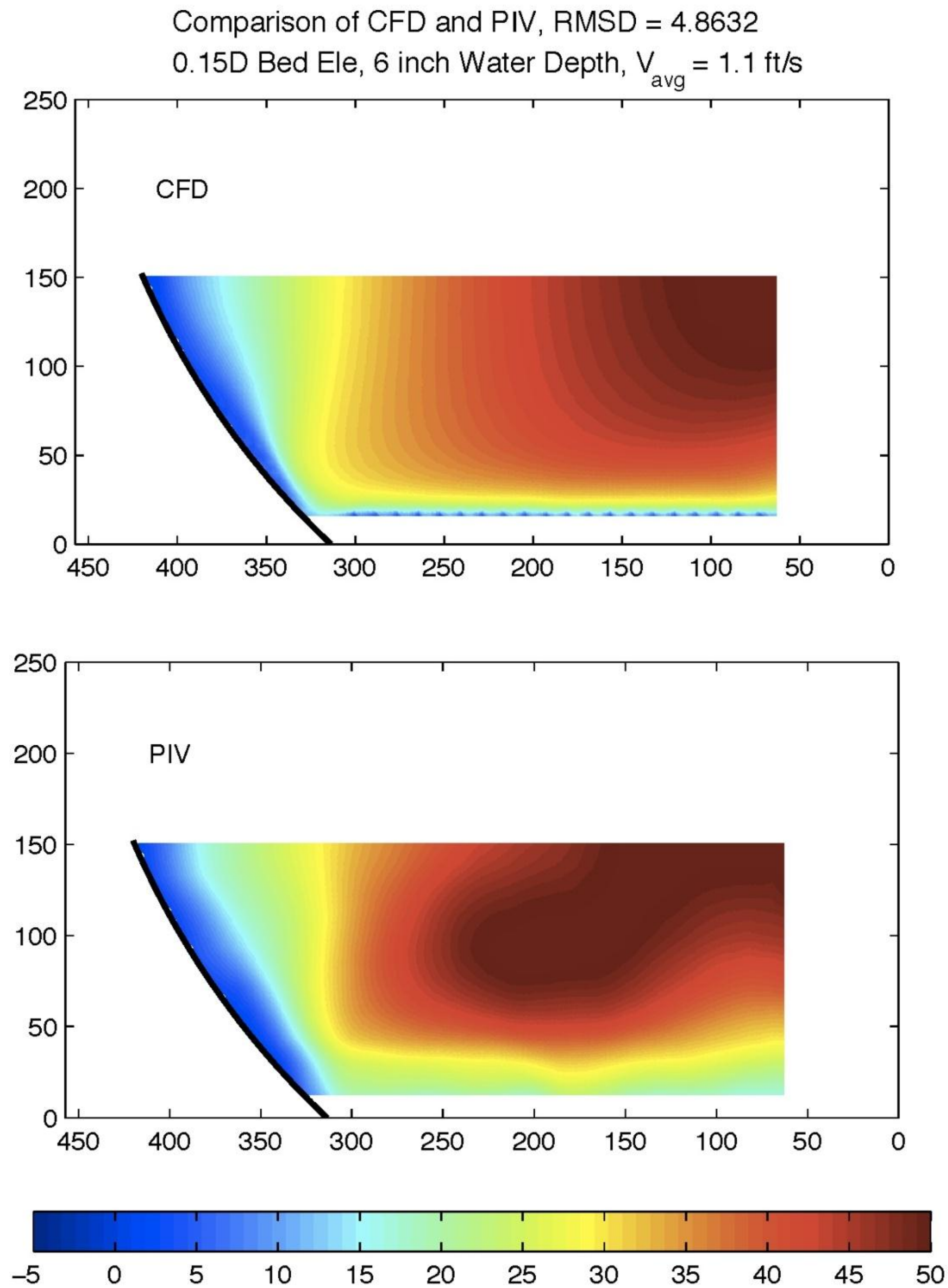


Figure 2.11 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 1.1 fps for 6 inch water depth (velocity: 33.5 cm/s)

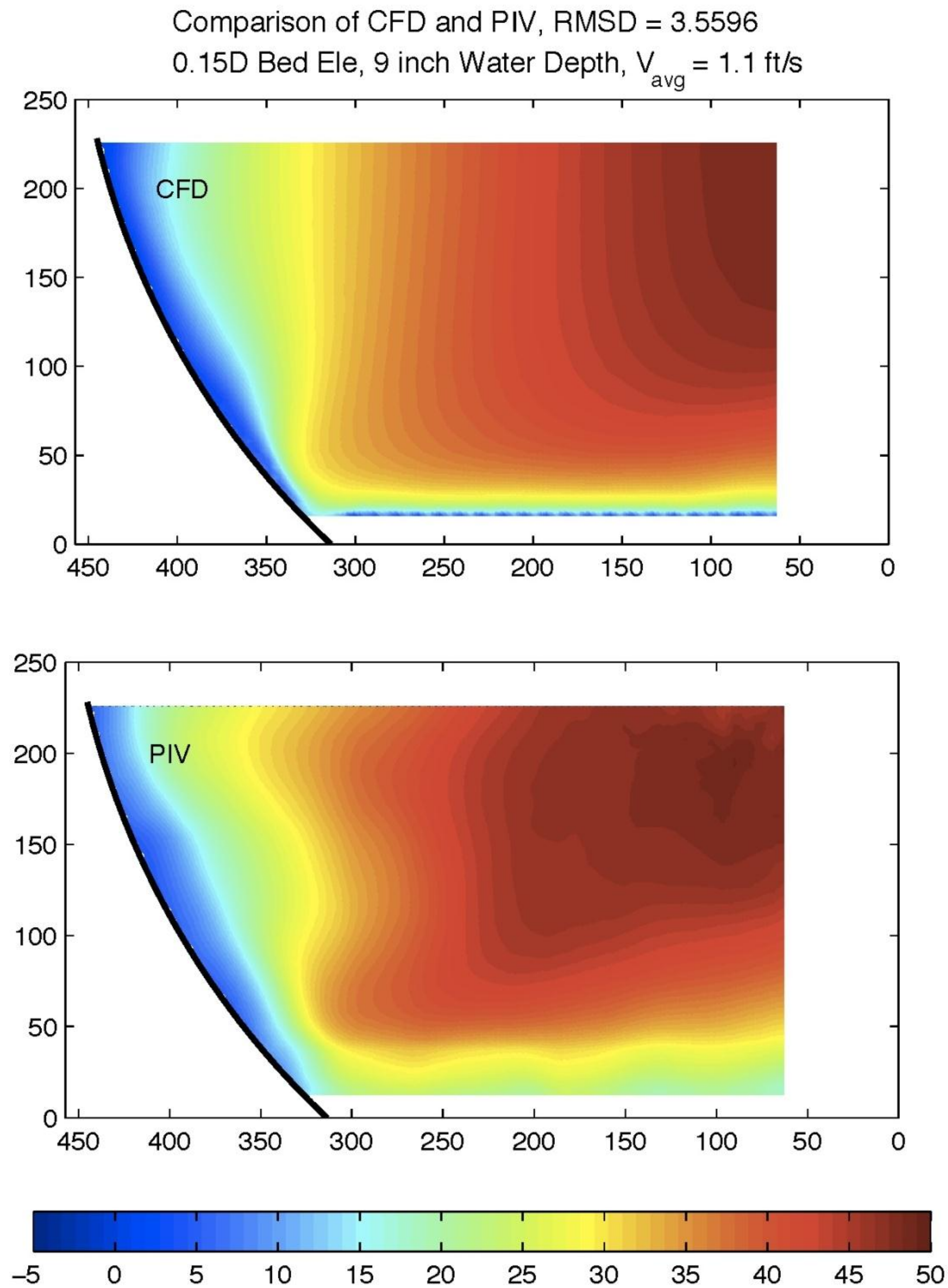


Figure 2.12 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 1.1 fps for 9 inch water depth (velocity: 33.5 cm/s)

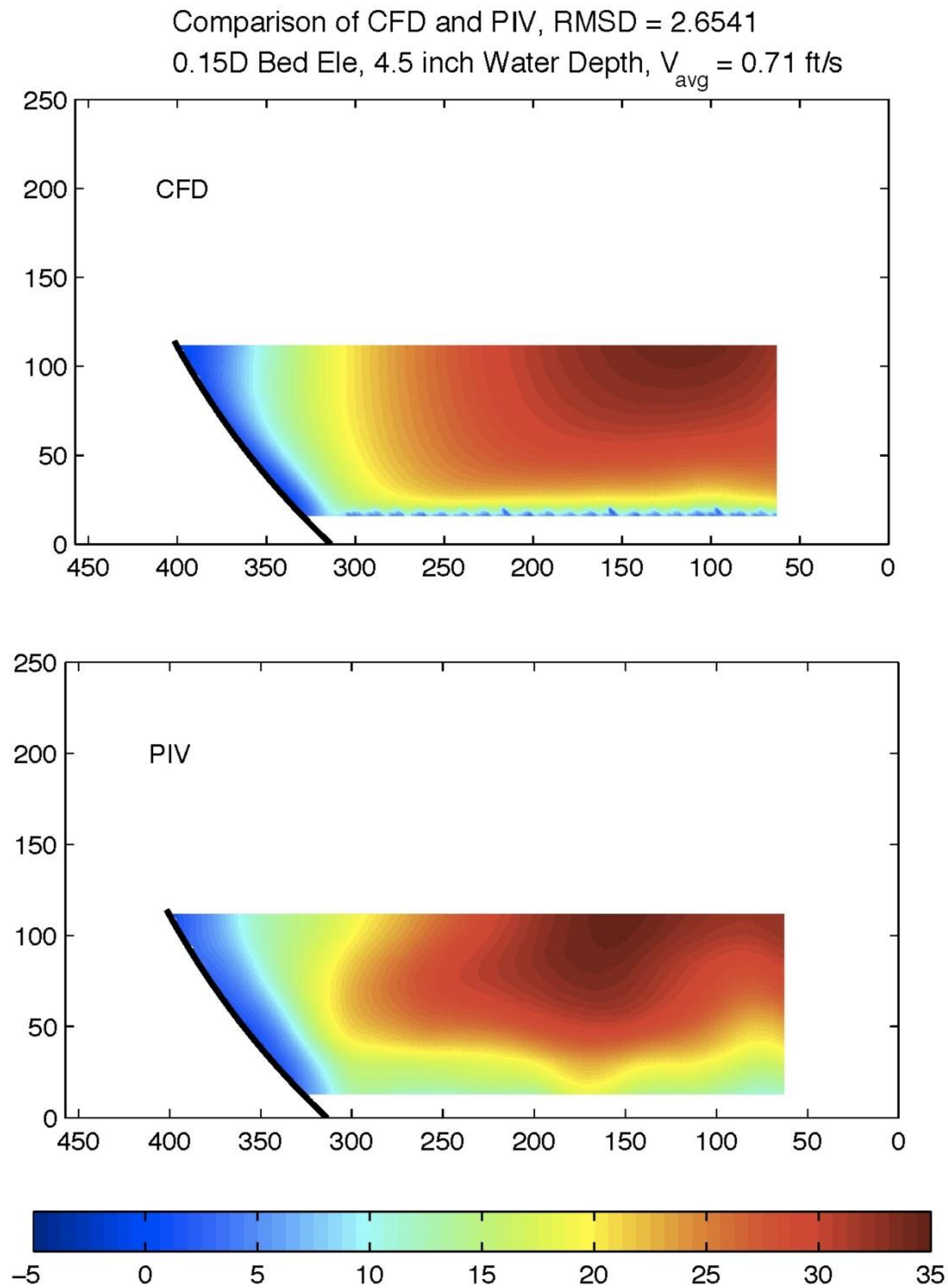


Figure 2.13 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 0.71 fps for 4.5 inch water depth (velocity: 21.6 cm/s)

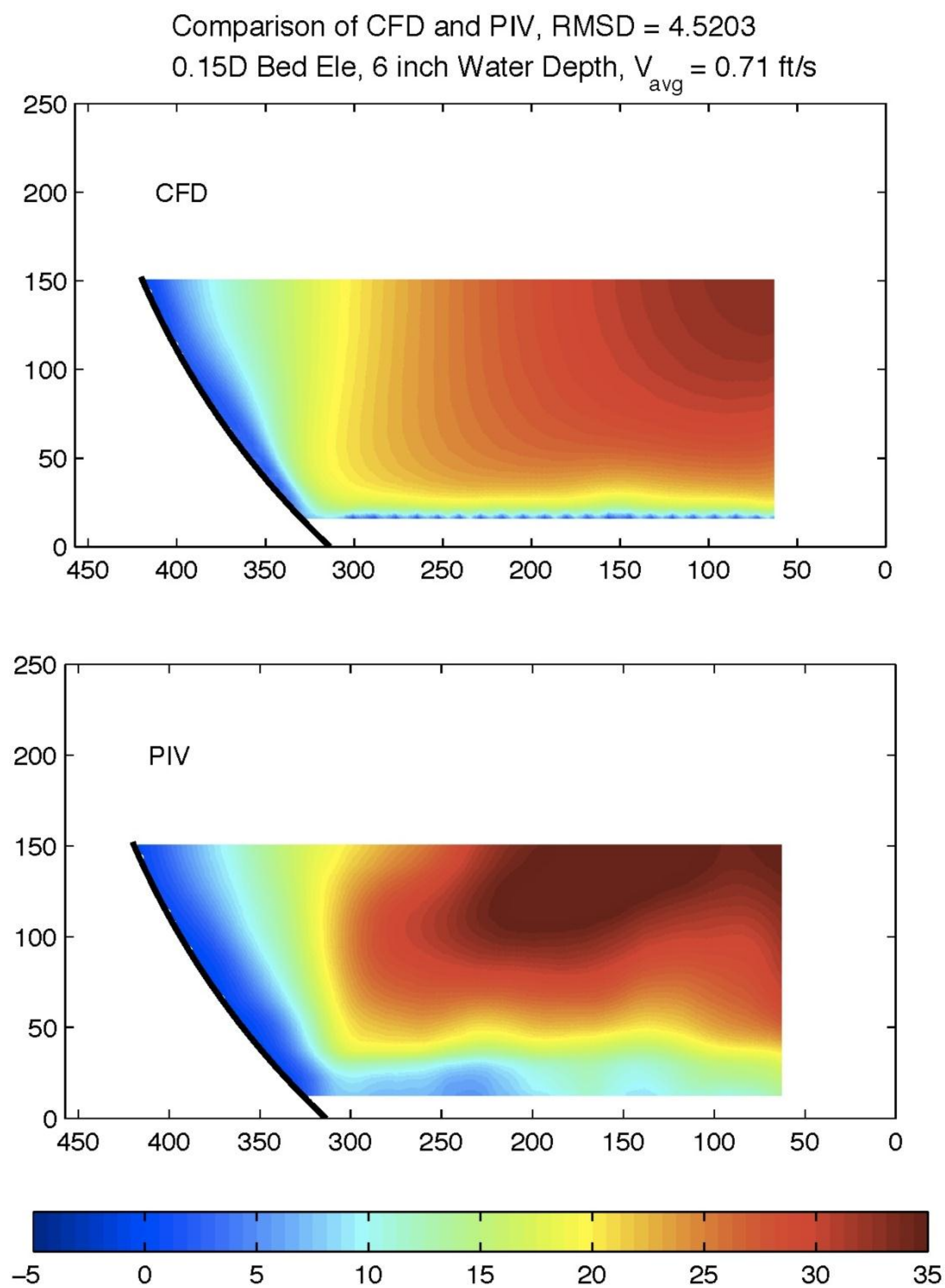


Figure 2.14 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 0.71 fps for 6 inch water depth (velocity: 21.6 cm/s)

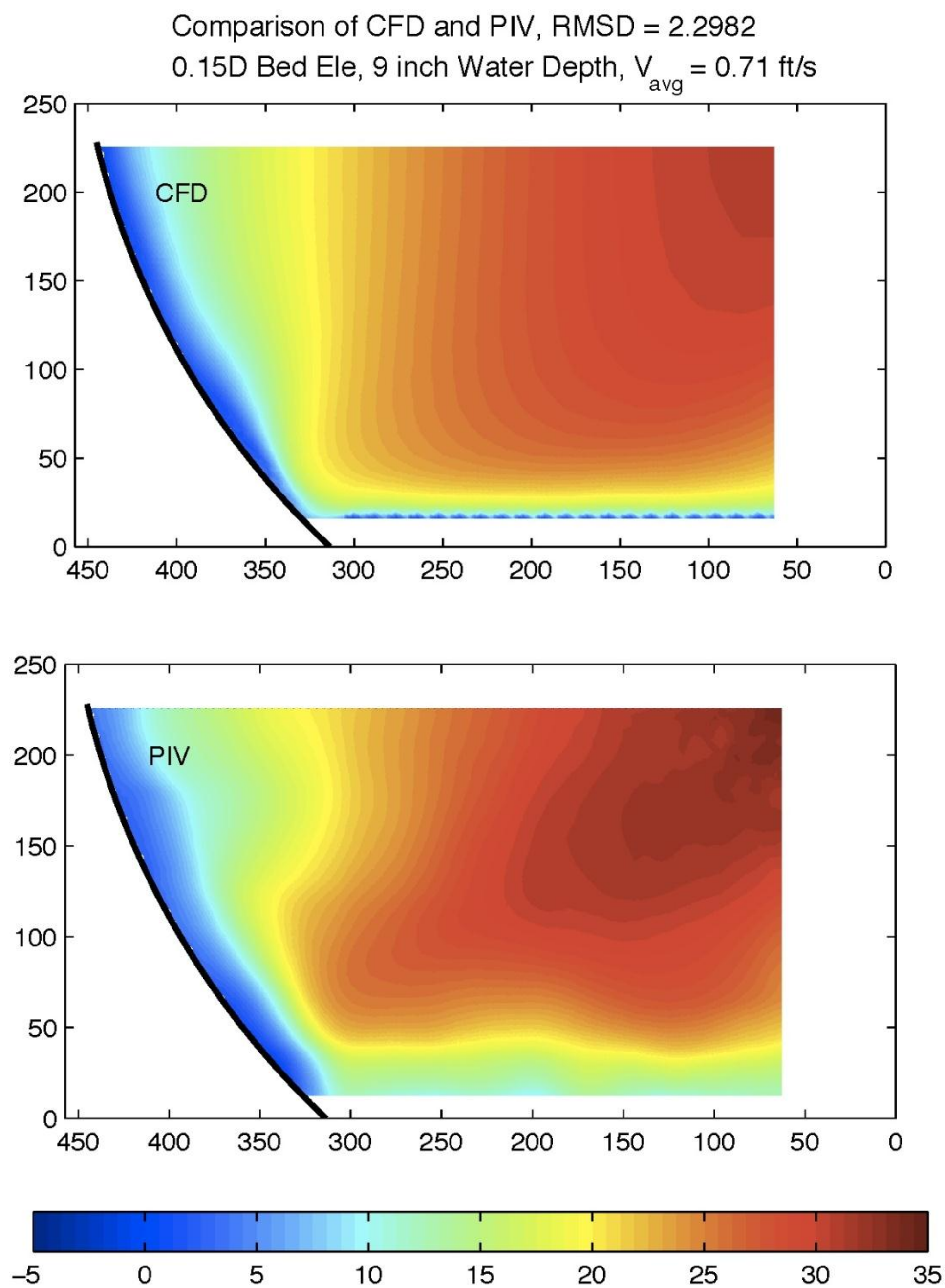


Figure 2.15 Comparison of CFD and PIV velocity contour under the condition of 0.15D bed elevation and 0.71 fps for 9 inch water depth (velocity: 21.6 cm/s)

(3) Bed elevation at 10.8 inch

The deepest sediment bed elevation in this study is 10.8 inch (0.3 D). Figure 2.16 shows the sketch of the experimental model (left) and the CAD model of culvert section geometry (right).

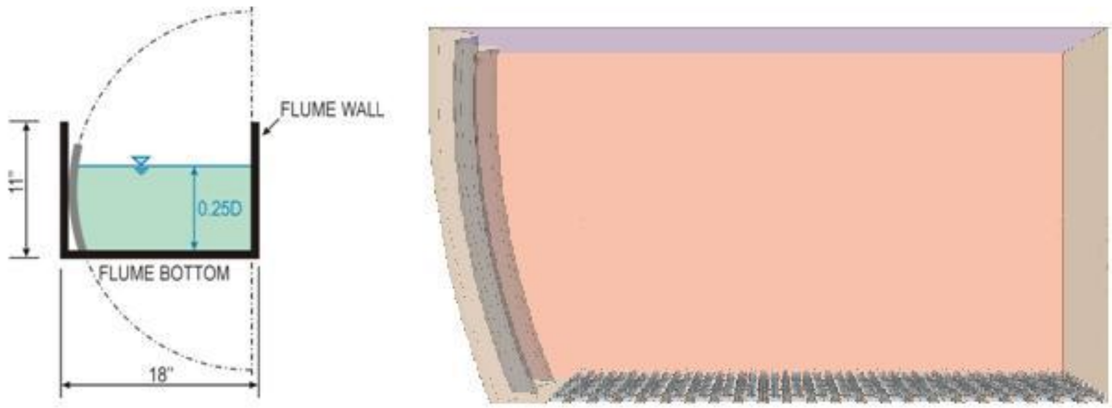


Figure 2.16 Sketch of experimental model (left) and CAD model of culvert section (right) under the situation of bed elevation at 10.8 inch

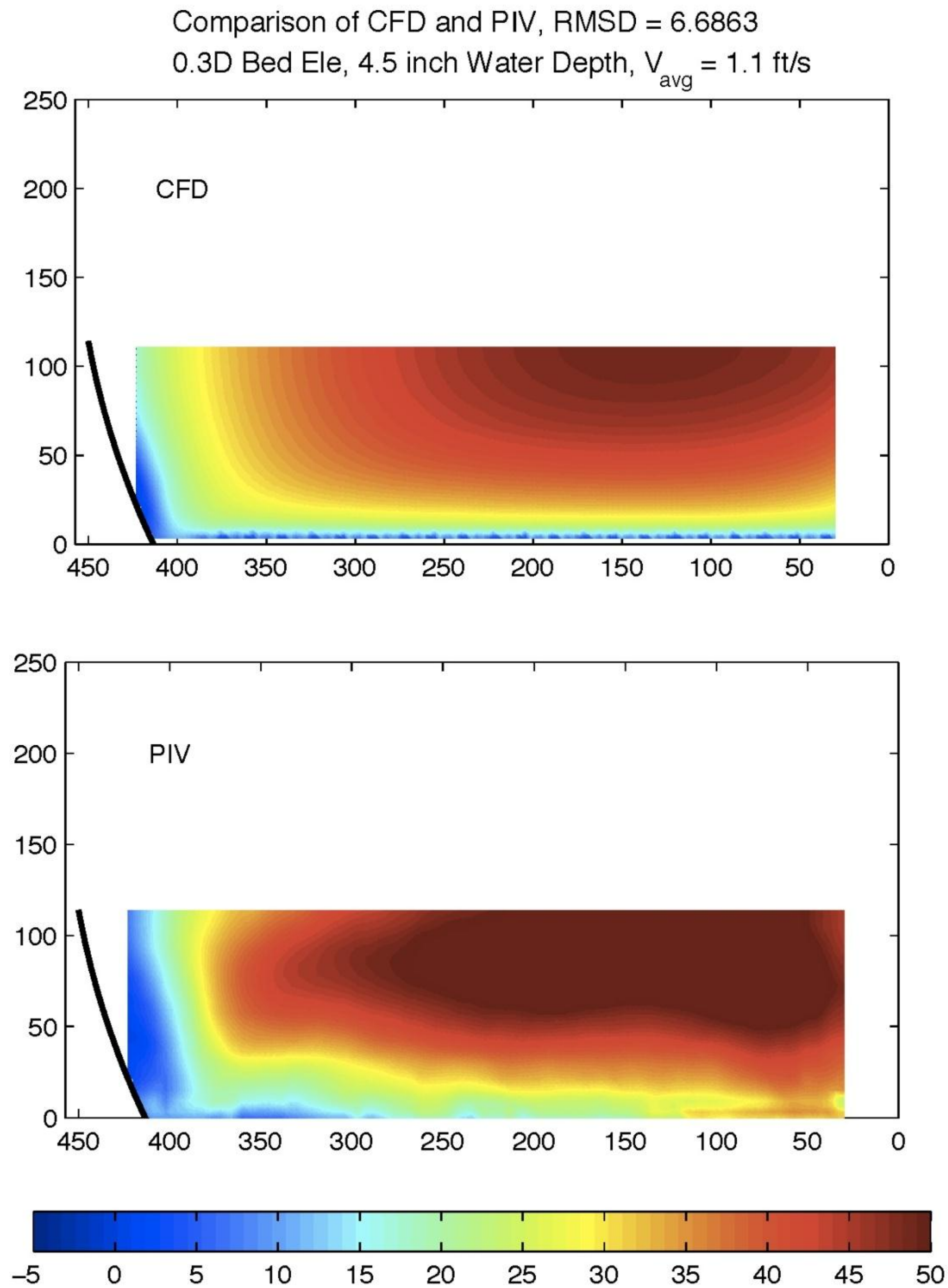


Figure 2.17 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 1.1 fps for 4.5 inch water depth (velocity: 33.5 cm/s)

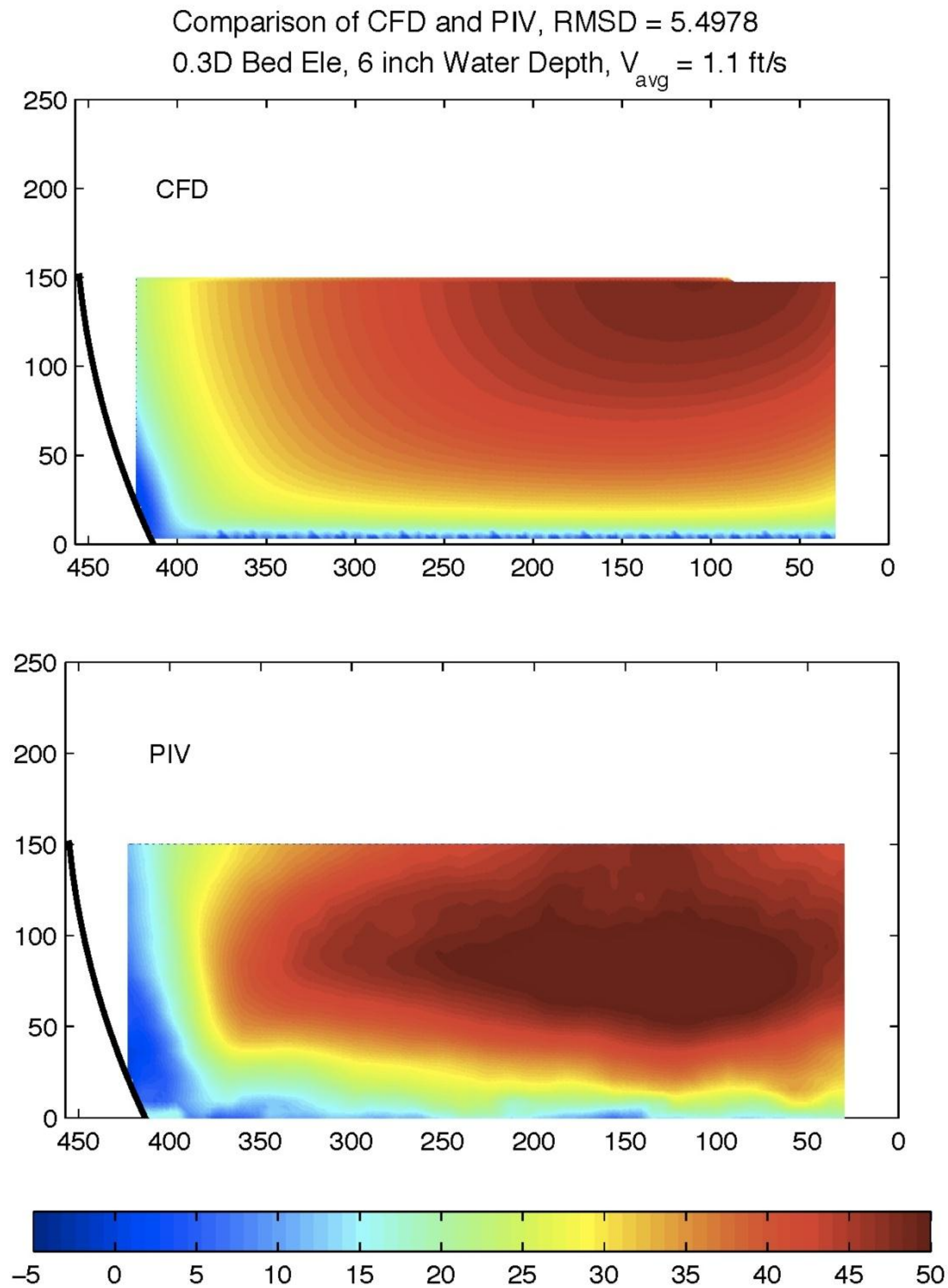


Figure 2.18 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 1.1 fps for 6 inch water depth (velocity: 33.5 cm/s)

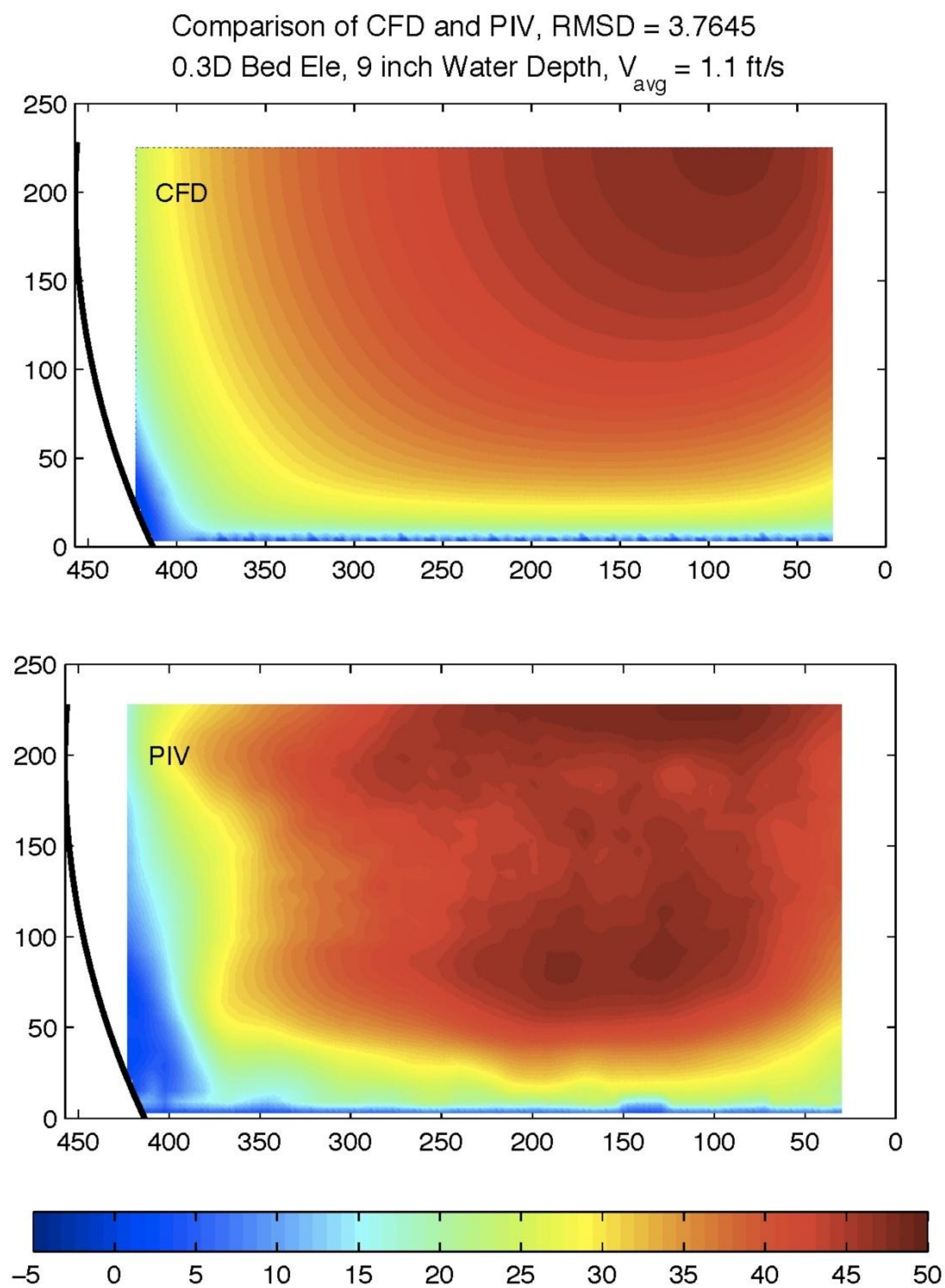


Figure 2.19 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 1.1 fps for 9 inch water depth (velocity: 33.5 cm/s)

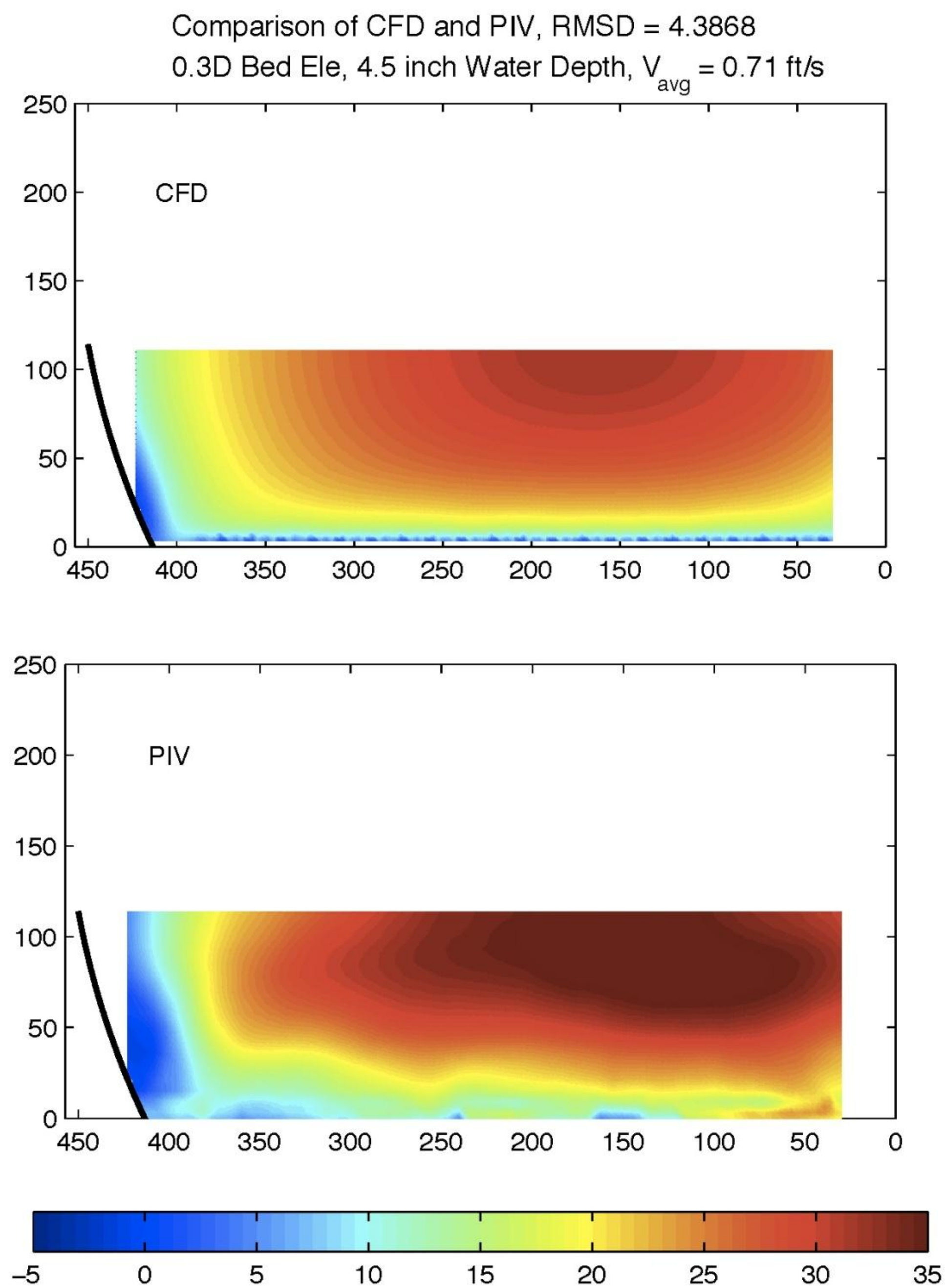


Figure 2.20 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 0.71 fps for 4.5 inch water depth (velocity: 21.6 cm/s)

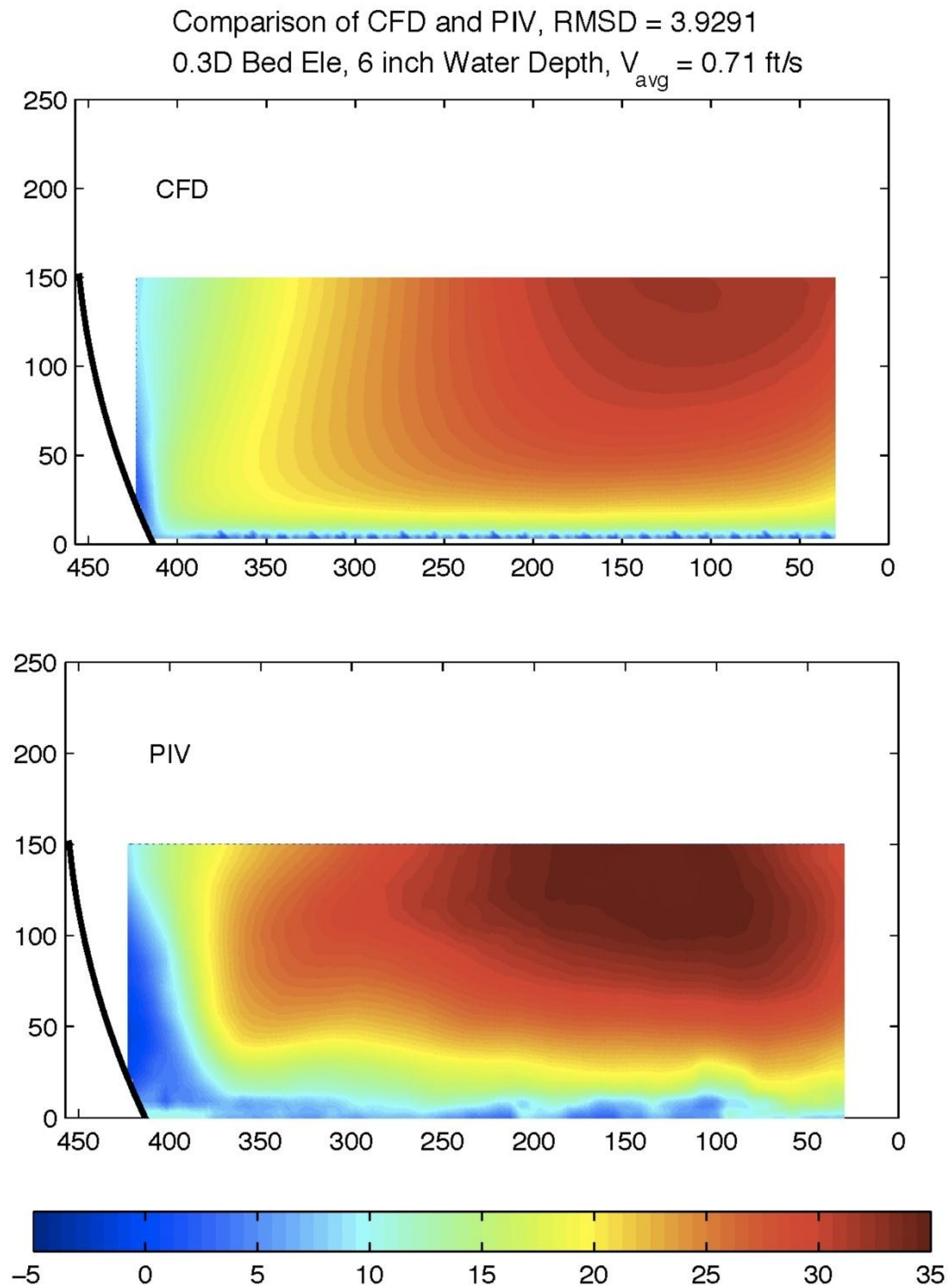


Figure 2.21 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 0.71 fps for 6 inch water depth (velocity: 21.6 cm/s)

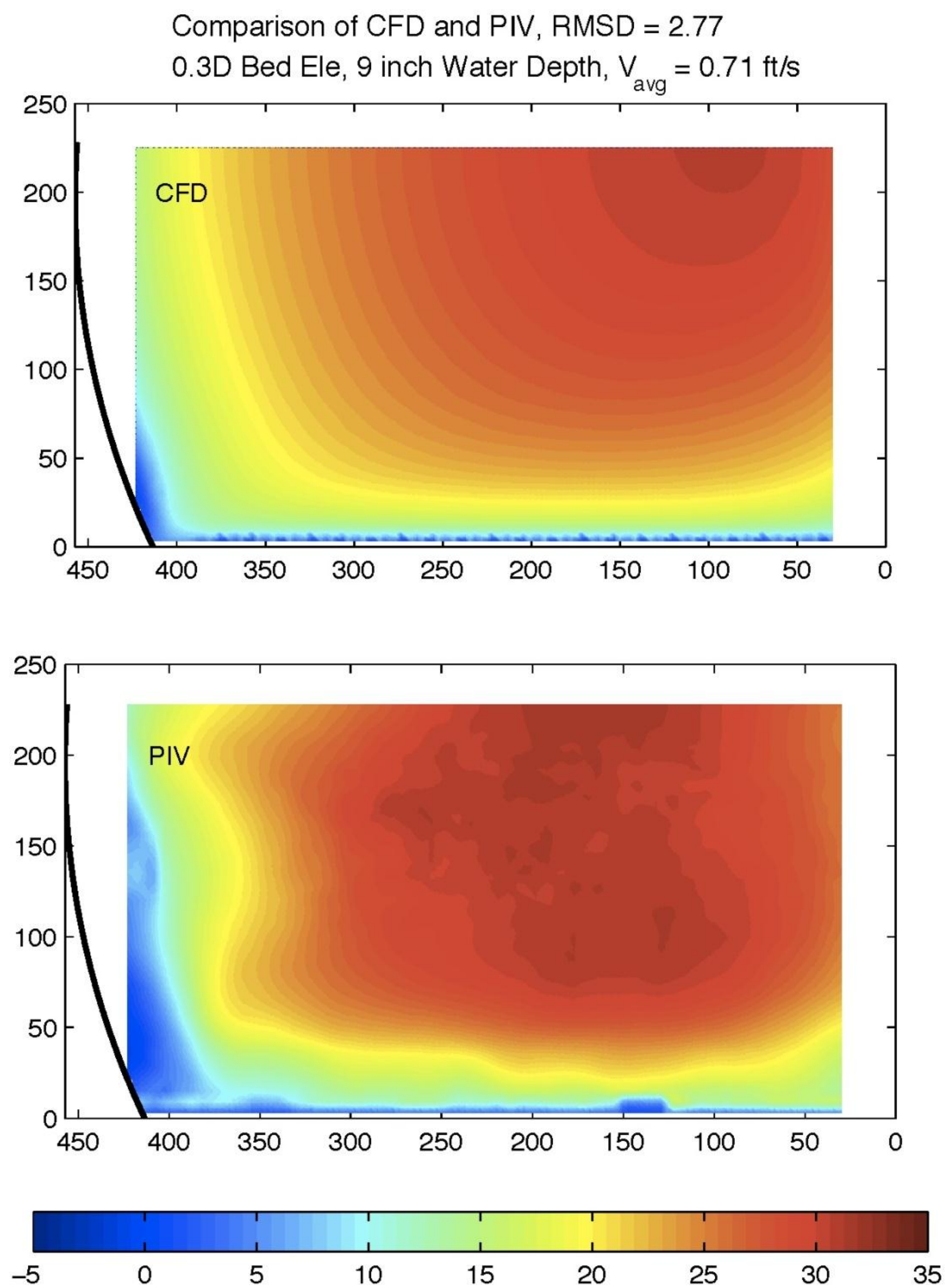


Figure 2.22 Comparison of CFD and PIV velocity contour under the condition of 0.3D bed elevation and 0.71 fps for 9 inch water depth (velocity: 21.6 cm/s)

2.1.2. Model accuracy analysis

When the data from two different methods are compared (for example, CFD and PIV), the difference can somewhat vary through the cross section. The root-mean-square-deviation (RMSD) is used in this study to provide a single measure of difference for the comparison of a large number of data points in an entire cross section. RMSD is defined as:

$$RMSD_{area} = \sqrt{\frac{\sum_{area}(V_1 - V_2)^2}{n_{area}}} \quad \cdots\cdots (2.1)$$

where V_1 and V_2 are velocity magnitudes from two different approaches. The RMSD tends to be greater for greater average velocity. A relative percentage error is used as a normalized measure of the error. It is defined as:

Relative error (%)=100 x RMSD/$V_{average}$ ⋯⋯ (2.2)

Based on the 5mm-by-5mm interpolated grid data, the RMSD are calculated for each flow and bed condition. RMSD and relative error vary from 2.182 cm/s to 9.515 cm/s and from 9.48% to 28.38%, respectively.

The RMSD number and relative error for each situation are listed in Table 2.3:

Table 2.3 RMSD and relative error between CFD and experimental results for different conditions

Bed elevation(inch)	Velocity (fps)	Water depth (inch)	PIV and CFD		ADV and CFD	
			RMSD(cm/s)	Relative error(%)	RMSD(cm/s)	Relative error(%)
0	0.71	4.5	5.185	23.96	2.545	11.76
		6	4.337	20.04	4.043	18.68
		9	4.317	19.95	4.116	19.02
	1.1	4.5	9.515	28.38	7.838	23.38
		6	6.227	18.57	3.177	9.48
		9	7.316	21.82	6.241	18.61
5.4	0.71	4.5	2.654	12.26	2.572	11.89
		6	4.520	20.89	2.327	10.75
		9	2.298	10.62	2.182	10.08
	1.1	4.5	3.466	10.34	3.828	11.42
		6	4.863	14.50	6.300	18.79
		9	3.559	10.61	3.448	10.28
10.8	0.71	4.5	4.386	20.27	5.452	25.19

Bed elevation(inch)	Velocity (fps)	Water depth (inch)	PIV and CFD		ADV and CFD	
			RMSD(cm/s)	Relative error(%)	RMSD(cm/s)	Relative error(%)
		6	3.929	18.16	3.552	16.41
		9	2.77	12.80	3.458	15.98
	1.1	4.5	6.686	19.94	7.834	23.36
		6	5.497	16.39	8.747	26.09
		9	3.764	11.23	6.435	19.19

2.1.3. Sources of the error

The CFD data are in good agreement with experimental measurement. The errors can be attributed to several reasons, which are summarized as below:

1. A trumpet-shaped inlet with honeycomb flow straightener were used in combination with tilting of the flume and the adjustment of flap gate to obtain a flow condition that is fairly close to uniform flow at the test section where PIV and ADV data were taken. Since it is neither uniform inlet nor fully developed flow (which requires a very long channel), some error was expected when it is compared to the fully developed flow from the cyclic boundary condition in CFD.
2. Some error in the discharge measured by the magnetic flow meters might contribute to a small part of the total error.
3. Explicit assumptions used in the CFD modeling and implicit assumptions embedded in the commercial CFD codes.
4. Interpolation error.
5. Collective effect of other minor experimental error.

2.2. 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