



Energy Systems Division

Computational Mechanics Research and Support for Aerodynamics and Hydraulics at TFHRC





Year 2 Quarter 2 Progress Report

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

Availability of This Report

This report is available, at no cost, at http://www.osti.gov/bridge. It is also available on paper to the U.S. Department of Energy and its contractors, for a processing fee, from: U.S. Department of Energy Office of Scientific and Technical Information P.O. Box 62 Oak Ridge, TN 37831-0062 phone (865) 576-8401 fax (865) 576-5728 reports@adonis.osti.gov

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Computational Mechanics Research and Support for Aerodynamics and Hydraulics at TFHRC, Year 2 Quarter 2 Progress Report

by

S.A. Lottes¹, C. Bojanowski¹, J. Shen², Z. Xie², and Y. Zhai² ¹ Transportation Research and Analysis Computing Center (TRACC) Energy Systems Division, Argonne National Laboratory ² Turner-Fairbank Highway Research Center

submitted to Kornel Kerenyi¹ and Harold Bosch¹ ¹ Turner-Fairbank Highway Research Center

June 2012

Table of Contents

1. Introduc	ction and Objectives	8
1.1. Com	putational Fluid Dynamics Summary	8
2. Comput	ational Fluid Dynamics for Hydraulic and Aerodynamic Research	10
2.1. Press	sure Flow Scour Formula Update for Highway Engineering Circular 18	10
2.1.1.	Testing the Theoretical Model for Pressure Flow Scour using CFD analysis, Initial Fit	12
2.1.2.	Second Fit	15
2.1.3.	Resolving Issues of the Second Fit	20
2.1.4.	References	24
2.2. Mod	eling of the Wind Tunnel Laboratory at TFHRC	24
2.3. Com	putational Modeling and Analysis of Flow through Large Culverts for Fish Passage	25
2.3.1.	Validation of the CFD models	25
	2.3.1.1. Comparison of CFD results with experimental data	26
	2.3.1.2. Model accuracy analysis	49
	2.3.1.3. Sources of the error	51
2.3.2.	References	51
2.4. Mod	eling of Truck Generated Salt Spray under Bridge with Sliding Mesh	52
2.5. New	Flume Design for Hydraulics Laboratory at TFHRC	52
2.6. Train	ning and Technology Transfer	58

List of Figures

Figure 2.1: Bridge deck geometry
Figure 2.2: Symmetric section of bridge deck from center of a railing post to halfway to the next post11
Figure 2.3 Vertical contraction and definition for geometric parameters
Figure 2.4 Initial fit based on Equations 2.2 and 2.314
Figure 2.5 Dividing streamline position for different flood heights17
Figure 2.6 Dividing streamline position for different flood heights18
Figure 2.7 CFD result for 3 different velocity levels19
Figure 2.8 Equation 2.6 at different velocity levels vs. CFD results
Figure 2.9 Equation 2.12 vs. CFD results22
Figure 2.10 Equation 2.13 at different velocity levels (K=0.95) vs. CFD results23
Figure 2.11 Comparison between predicted scour (design scour depth) and experimentally measured scour depth
Figure 2.12 Sketch of experimental model (left) and CAD model of culvert section (right) for bed elevation at 0 inch
Figure 2.13 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 2.14 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 2.15 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)31
Figure 2.16 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)32
Figure 2.17 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 2.18 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 2.19 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)
Figure 2.20 Sketch of experimental model (left) and CAD model of culvert section (right) under the situation of bed elevation at 5.4 inch
Figure 2.21 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 2.22 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 2.23 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 2.24 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)40
Figure 2.25 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 2.26 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)
Figure 2.27 Sketch of experimental model (left) and CAD model of culvert section (right) under the situation of bed elevation at 10.8 inch
Figure 2.28 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 2.29 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)45
Figure 2.30 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 2.31 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)47
Figure 2.32 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 2.33 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)49
Figure 2.34 A conceptual drawing of the preliminary inlet design53

Figure 2.35: Two different transition profiles of trumpet: streamline and multi-line
Figure 2.36: Comparison of velocity uniformity along different sections
Figure 2.37: Comparison of velocity uniformity along different sections in the test regime56
Figure 2.38: Comparison between with-honeycomb and without-honeycomb for the streamline trumpet
Figure 2.20: Appeursement for CED training course hold in March
Figure 2.40: Announcement for CFD training course held in March, topics and tutorials60

List of Tables

Table 2.1: Boundary conditions	
Table 2.2 Contour plots comparing CFD, PIV, and ADV. All figures compare CFD against F	VIV except Figure
2.19, which compares CFD against ADV.	
Table 2.3 RMSD and relative error between CFD and experimental results for different of	conditions 50

1. Introduction and Objectives

The computational fluid dynamics (CFD) and computational structural mechanics (CSM) focus areas at Argonne's Transportation Research and Analysis Computing Center (TRACC) initiated a project to support and compliment the experimental programs at the Turner-Fairbank Highway Research Center (TFHRC) with high performance computing based analysis capabilities in August 2010. The 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 TFHRC for a period of five years, beginning in October 2010. The analysis methods employ well benchmarked and supported commercial computational mechanics software. 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.

The major areas of focus of the project are wind and water effects on bridges — superstructure, deck, cables, and substructure (including soil), primarily during storms and flood events — and the risks that these loads pose to structural failure. For flood events at bridges, another major focus of the work is assessment of the risk to bridges caused by scour of stream and riverbed material away from the foundations of a bridge. Other areas of current research include modeling of flow through culverts to improve design allowing for fish passage, modeling of the salt spray transport into bridge girders to address suitability of using weathering steel in bridges, CFD analysis of the operation of the wind tunnel in the TFHRC wind engineering laboratory.

This quarterly report documents technical progress on the project tasks for the period of January through March 2012.

1.1. Computational Fluid Dynamics Summary

The primary Computational Fluid Dynamics (CFD) activities during the quarter concentrated on the development of models and methods needed to continue the ongoing work in scour modeling, culvert modeling, CFD analysis of the Turner-Fairbank wind tunnel, CFD modeling and analysis of salt spray from

large trucks passing under bridges using weathering steel, and modeling and analysis of concept testing for an in-situ scour device to measure scour related properties of sediment bed material. During this quarter, modeling and analysis of the separation of flow at the leading edge of a flooded bridge deck was continued to aid in the development of an enhanced approach for evaluating scour due to submergence of bridge decks during floods in the federal guidelines. Modeling of flow through culverts for fish passage continued with the work on a porous media model to capture the effects of large diameter gravel in the bottom of the culvert revealed some difficulties in obtaining physically realistic results in the modeling of flow parallel to porous beds. A meshed out rough bed model for large roughness elements was also tested and compared against experiment. This model appears to yield adequate engineering accuracy. Modeling of the wind tunnel in the TFHRC laboratory continued, and a new CFD model was developed to analyze flow in the wind tunnel and the room under a variety of flow conditions including with and without furniture. Work on the CFD model using the sliding mesh capabilities of STAR-CCM+ with multiphase droplet tracking was continued. Simulations for a matrix of conditions are being carried out with different droplet sizes, both a single truck and a truck followed by another truck, and two wind speeds with the wind crossing the domain in four directions with respect to the truck or trucks.

2. Computational Fluid Dynamics for Hydraulic and Aerodynamic Research

During the first quarter of 2012, modeling and analysis of the separation of flow at the leading edge of a flooded bridge deck was continued to aid in the development of an enhanced approach for evaluating scour due to submergence of bridge decks during floods. This work culminated in the completion and submission of a new procedure for calculating vertical contraction scour in HEC-18 [1] by TFHRC staff in collaboration with TRACC analysts. Modeling of flow through culverts for fish passage continued with work on using a porous media model to capture the effects of large diameter gravel in the bottom of the culvert. An alternative approach was also pursued. In this approach a uniform arrangement of large roughness elements approximately the size of the gravel were meshed out explicitly in the geometry of the model. Results of CFD simulations with the large roughness elements in the model were compared against experiment and are reported in Section 2.3.1.1. TFHRC provided geometry files for their wind tunnel laboratory and the new CFD model was used to analyze flow in the wind tunnel and the room under a variety of flow conditions including with and without furniture. The CFD model using the sliding mesh capabilities of STAR-CCM+ with multiphase droplet tracking is being used to test the effects of a variety of conditions of droplet distributions under bridges.

2.1. Pressure Flow Scour Formula Update for Highway Engineering Circular 18

An update to the submerged-flow bridge scour evaluation procedure in HEC-18 [1] was completed by TFHRC and TRACC. The approach to scour hole depth estimation assumes that the scour process will enlarge the area under the bridge until it is large enough to pass the flow with a condition of critical shear stress at the bed. The bridge deck is a bluff body in the flow and flow separation will normally occur at the upstream bottom edge of the submerged bridge deck. The separation zone under the bridge restricts the area open to flow under the deck and is therefore an important parameter in reliably predicting the depth of the scour hole. CFD simulations were performed to investigate the relation between the initial opening height under the submerged deck before scour and the thickness of the separation zone, and one test was done to see if the thickness of the separation zone changed during the scour process.



Figure 2.1: Bridge deck geometry

The bridge deck with superstructure geometry is shown in Figure 2.1. To reduce computer time and eliminate the effects of the flume side walls, the simulations were performed using a section of the bridge deck cut through the middle of a post and running half the distance to the center of the next post. This geometry is shown in Figure 2.2.



Figure 2.2: Symmetric section of bridge deck from center of a railing post to halfway to the next post

Symmetric boundary conditions were used on the stream wise sides of the domain. The simulations were done as single phase flow with a flat water surface using a symmetry boundary condition at the surface. Previous tests have been done using the multiphase VOF model for free surface flow, and the

flat surface assumption is good except in the case with the bridge deck very close to the surface but still overtopped. The inlet boundary was taken to be a uniform velocity located just at the outlet of the honeycomb in the TFHRC scour flume. The honeycomb is a flow straightener that also strips off boundary layers, and therefore the uniform inlet velocity is a reasonably good assumption applied at this position.

2.1.1. Testing the Theoretical Model for Pressure Flow Scour using CFD analysis, Initial Fit

Testing of the theoretical model was conducted using data from Arneson [2], Umbrell [4], and TFHRC [3] and flow visualization using PIV and CFD modeling. The bulk of the material presented in this section pertains to the results of the CFD modeling and final tuning of the formula for separation zone thickness, *t*, used in calculation of scour depth, defined as:

$$y_{2} = h_{b} + y_{s} = \left(\frac{V_{ue} \cdot h_{ue}}{K_{U} \cdot D_{50}^{1/3}}\right)^{6/7} + t$$
 2.1

Where:

- y_s is the contracted section between the bottom of the separation zone and the lowest point of the scour hole. This is the depth effective for the flow through the bridge opening after scour.
- V_{ue} is the average upstream velocity within the area from which all streamlines go under the bridge deck.
- h_{ue} is the depth below the stagnation point upstream, below which all streamlines go under the bridge superstructure. For a fully inundated bridge, this depth is conservatively taken to be $h_{ue}=h_b+T$. For partially submerged bridge, $h_{ue}=h_b+a$, where a is the amount of submergence.
- K_{U} is 6.19 for SI units, or 11.17 for U.S. customary units.
- D_{50} is sediment size (m or ft).
- *t* is the thickness of the separation zone at the location above maximum scour.

In Equation 2.1 the value of the separation zone thickness, *t*, is not available from experimental measurements. For practical design, it is necessary that *t* does not become one of the design parameters to be obtained. It was therefore necessary to develop an analytical model that ties *t* to other measured parameters and then use the experimental data to calibrate this model. This calibrated model, along with the fitting parameters computed in the calibration process, is used in the final scour evaluation.



Figure 2.3 Vertical contraction and definition for geometric parameters.

In the approach taken by TFHRC, a dimensional analysis was used to obtain a format of the equation for the thickness of the separation zone. Experimental data was then used to find the best parameter fit using the least-squares method. A safety margin was also intentionally added to this formula to allow sufficient reliability in design. The initial calibration process yielded the following two curves defining separation zone thickness for the cases after and before the deck's overtopping:

$$\frac{t}{h_t} = 1.7 \left(\frac{V}{\sqrt{gh_t}} \frac{1.17}{h_t} \right)^{-0.3} \left(\frac{\rho V h_t}{\mu} \right)^{-0.15} \left(\frac{h_b}{h_u} \right)^{1.2} \left(\frac{h_t}{h_u} \right)^{-1}$$
2.2

$$\frac{t}{a} = 1.7 \left(\frac{V}{\sqrt{ga}}\right)^{-0.3} \left(\frac{\rho Va}{\mu}\right)^{-0.15} \left(\frac{h_b}{h_u}\right)^{1.2} \left(\frac{a}{h_u}\right)^{-1}$$
2.3

Where:

g – acceleration due to gravity ρ – density of water μ – viscosity of water a – h_{μ} - h_{b}

The geometrical parameters used in these equations are defined in Figure 2.3. It is important to note that the thickness of separation zone from Equations 2.2 and 2.3 is not the theoretical separation zone thickness defined by the boundary streamline. It is obtained from experimental data and produces the best scour depth prediction when used with Equation 2.1. It includes compensation for errors/biases of a number of other factors as well as needed conservatism for design. To visualize these curves the

conditions from Example 1 defined in the new HEC-18 update was adopted here with the following given parameters:

Upstream channel width and bridge opening width (W)= 40 ft (12.2 m) Total discharge (Q) = 2800 ft³/s (79.3 m³/s) Upstream channel discharge (Q₁) = 2000 ft³/s (56.6 m³/s) Upstream floodplain discharge = 800 ft³/s (22.7 m³/s) Upstream channel flow depth (h_u)= 10.0 ft (3.0 m) Bridge opening height (h_b) = 8.0 ft (2.4 m) Deck thickness (T) = 3 ft (0.91 m) Bed material D₅₀ = 15 mm (V_c = 6.0 ft/s, 1.8 m/s, V_c is the critical velocity for the sediment size) Upstream channel velocity (V=Q₁/(W_{hu})) = 2000/(40 x 10) = 5.0 ft/s (1.5 m/s)

Figure 2.4 presents Equations 2.2 and 2.3 for several different upstream velocities and other quantities defined as above. It was noted that the initial fit allows for separation zone thickness significantly greater than zero for the cases where the water level was just slightly above the bottom line of the bridge superstructure ($t \rightarrow 0$ as $t_h \rightarrow 0$). Although Equations 2.2 and 2.3 fit to the experimental scour data well, creation of a significant separation zone when the water is at a level that just touches the superstructure is counterintuitive. Also there was a discontinuity introduced between the two regions described by the curves – before overtopping and after overtopping.



Figure 2.4 Initial fit based on Equations 2.2 and 2.3

2.1.2. Second Fit

In order to address these issues a new fit was proposed that would also meet the reliability criteria for the experimental data prediction as follows

$$\frac{t}{h_b} = 1.7 \left(\frac{V}{\sqrt{gh_u}} \frac{T}{h_t} \right)^{-0.3} \left(\frac{\rho V h_u}{\mu} \right)^{-0.15} \left(\frac{h_b}{h_u} \right)^{0.2} \left(\frac{h_t}{h_u} \right)^{0.2}$$
2.4

$$\frac{t}{h_b} = 1.7 \left(\frac{V}{\sqrt{gh_u}}\right)^{-0.3} \left(\frac{\rho V h_u}{\mu}\right)^{-0.15} \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{a}{h_u}\right)^{0.2}$$
 2.5

This set of equations was obtained in the following way, based on Eq.2.2 and Eq. 2.3. ht in the square root of the first term and the second term are of equal and opposite sign and can therefore be replaced by another parameter without changing the value of the right hand side. Choosing the upstream depth, h_u, yields the upstream Froude number as the first factor in the first term and the upstream Reynolds number as the second term. These non-dimensional groups appear to be a more natural choice. The same replacement can be done for "a" in Eq. 2.3. The factor of 1.1 in Eq. 2.2 arose from the conservative determination of the upstream position of the stagnation streamline in simulation cases when a railing was present that is not shown in Figure 2.3. The term with 1.1 factor has a negative exponent, and therefore dropping the 1.1 factor in the term makes it more conservative, physically realistic and removes the discontinuity in t when the water level is at the overtopping point, $h_t = T$. The exponent of the last term in Eq. 2.2 and Eqn. 2.3 was rounded down to -1 when it could have been rounded up to -0.8 without substantially affecting the quality of the fit. Rounding it up to -0.8 and multiplying the equations through by h_t/h_b to normalize by the bridge opening height, h_b , and simplifying yields Eqs. 2.4 and 2.5. This form of the equations addresses both issues noted for the initial data fit. Due to the fact that the discontinuity was removed and noting that $h_t = h_u - h_b$ and that $a = h_u - h_b$ the two equations could be now combined into one equation:

$$\frac{t}{h_b} = 1.7 \left(\frac{V}{\sqrt{gh_u}}\right)^{-0.3} \left(\frac{\rho V h_u}{\mu}\right)^{-0.15} \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{h_t}{h_u}\right)^{0.2} \left(1 - \frac{h_w}{h_t}\right)^{-0.3}$$
2.6

where h_w is defined as a wier flow height defined as $h_w = h_t - T$ for $h_t > T$, and $h_w = 0$ otherwise.

Subsequently a series of parametric studies and CFD simulations were conducted to scrutinize the latest fit to the experimental data. One part of the parametric study uses the bridge site specified in Example 4 of the updated section of HEC-18 – clear water condition. Multiple cases with different water level (main variable), different upstream water velocities (secondary variable) and different clearance between the river bed and the bridge superstructure were analyzed.

Separation zone streamline plots for different water levels in the case with h_b =2.4 m are shown in Figure 2.5 and Figure 2.6. The plotted cases are for an approach velocity of 1.5 m/s, however, further simulations for other velocities showed that the results are nearly independent of velocity (as is apparent in Figure 2.7). The cases are also for the pre-scoured flat bed, and consequently the thickest point of the separation zone is close to the upstream opening of the flooded bridge. Consequently, the

TRACC/TFHRC Y2Q2

scour rate is initially highest near the bridge opening but as the scour hole develops, the deepest point moves close to the downstream end of the bridge opening, and the thickest point in the separation zone also moves back near downstream exit of the opening. This behavior is not covered in CFD results presented here. For comparison the second fit based on Equation 2.6 is plotted against this data in Figure 2.8. The dip in t in the CFD results occurs when the water level of the overhang of the bridge deck surface beyond the first beam, a geometry feature that may be present on most but not all real bridges, and geometry of real bridge superstructures may be more complex than that used in the analysis and result in additional non-monotonic variations of the value of t as a function of h_t .



Figure 2.5 Dividing streamline position for different flood heights





Figure 2.6 Dividing streamline position for different flood heights



Figure 2.8 Equation 2.6 at different velocity levels vs. CFD results

TRACC/TFHRC Y2Q2

2.1.3. Resolving Issues of the Second Fit

Attention was given to three areas of the results in which the previous fit showed potential issues. These areas were:

- 1. When the submergence just starts— the amount of submergence, h_t , is small. It is expected that the separation zone thickness would approach zero when h_t approaches zero. The relationship between t and h_t are shown by CFD result to be nearly linear up to the point the deck overhang starts to be inundated.
- 2. When a significant weir flow occurs—the amount of submergence, h_t , is large. It is observed in previous physical and CFD simulations that t decreases with h_t when the superstructure is fully inundated. While recent CFD simulations find t remaining nearly constant after full inundation starts, the simulations were based on an unscoured stream bed. It is expected that if scour hole develops, t would increase because of the greater size of the bridge opening and would also show some increasing trend with h_t . However, the t given by the proposed equation is not bounded for large h_t . It approaches infinity when h_t approaches infinity.
- 3. When water velocity is higher—for example, when velocity is 2.5 m/s, the predicted value of t from the fitted equation falls well below the CFD prediction. The proposed equation has a strong inverse dependence on velocity, with a total power of -0.45. Since the Reynolds number of the orifice flow is fairly high even for relatively minor flooding and the bridge cross section consists of sharp edges, it is expected that the velocity dependence is very low. CFD simulations also show minimal changes in *t* for different upstream velocities.

The proposed equations of the second fit give zero t when h_t is near zero, and are somewhat conservative for small h_t . Problem Area 1 is therefore less of a concern for the second fit.

The increase of *t* with significant weir flow is associated with the last term of the fit Equation:

$$\left(1-\frac{h_w}{h_t}\right)^{-0.3}$$
 2.7

This term is a result of the Froude number term of the fully inundated scour equation and is never datafitted individually. It is therefore reasonable to make changes to it if there is a good basis. By observation from CFD and engineering judgment, it is likely that the increasing trend of t with h_t is not as strong as that shown in Figure 2.8. A smaller exponent appears warranted to reduce the concern of Problem Area 2.

Problem Area 3 needs to be discussed from two different aspects:

1. Format of equation: The Froude number, F_r , and Reynolds number, R_e , terms both contain factors V and h_u . When the exponent of F_r is twice as that of R_e , h_u is eliminated. Whereas the velocity terms in F_r and Re are multiplied and their influence increases. If the exponent of F_r and R_e are the same in magnitude and have opposite sign, then the velocity terms in them are eliminated, while the h_u terms become more pronounced. The implication of V terms having a large exponent is a strong inverse velocity dependence. The result of h_u terms having a large

exponent is that *t* does not scale with bridge dimensions and flow depths. Both possibilities are against CFD findings and engineering experience.

 Experimental procedure: The fit equations have parameters that are fitted to experimental data. Values of the exponents have produced a very good fitting to lab data. If the current exponents indicate a dependence on velocity, it is likely that there will be some dependence within the range of the sets of experiments, however, the dependence may be small.

Three potential approaches were proposed in discussions between TFHRC and TRACC to address these issues:

Approach (1): Maintain scaling without distortion. The current equations provide t that scales with linear dimensions and fits data well. The weak point of this fit is that it has strong dependency on the velocity and t decreases significantly when at higher velocity. It is believed that the source of velocity dependence is a result of the data analysis scheme that leads to the result. t is calculated by subtracting the effective orifice flow depth, y_2 , from the total depth, h_b+y_s . y_2 is obtained from discharge, $V_{ue} h_{ue}$ or Q, and critical velocity, V_c . The equation for critical velocity is based on uniform, fully developed flow in an open channel. y_2 from this method may contain a certain amount of error that is related to velocity. This error may be compensated in the fitted equations that lead to t. However, engineering judgment supports the proposition that the resultant scour depth, y_s , should be free of unrealistic influence from velocity, and leaving the dependence in the fitted equation, could under some real case full scale conditions lead to under prediction of scour depth.

Approach (2): Remove the velocity dependence. With a significant velocity dependence, current design equations are not consistent with CFD results over the fairly narrow range of velocities tested (which is nearly completely independent from velocity), but the fitted equation is very close to the CFD results at one specific velocity. This velocity can be used as a reference that anchors the design equation to the CFD-predicted values. If V = 0.5 m/s is chosen, the Froude number and Reynolds number terms are:

$$\left(\frac{V}{\sqrt{gh_u}}\right)^{-0.3} \left(\frac{\rho V h_u}{\mu}\right)^{-0.15} = \left(\frac{0.5}{\sqrt{g}}\right)^{-0.3} \left(\frac{\rho 0.5}{\mu}\right)^{-0.15} \approx \frac{1}{4.13}$$
 2.8

Note that the h_u factor vanishes because -0.3 is twice -0.15. The equations for t become:

$$\frac{t}{h_b} = \frac{1.7}{4.13} \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{h_t}{h_u}\right)^{0.2} \left(\frac{T}{h_t}\right)^{-0.1}$$
 2.9

$$\frac{t}{h_b} = \frac{1.7}{4.13} \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{a}{h_u}\right)^{0.2}$$
 2.10

Or in a single equation form

$$\frac{t}{h_b} = \frac{1.7}{4.13} \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{h_t}{h_u}\right)^{0.2} \left(1 - \frac{h_w}{h_t}\right)^{-0.1}$$
2.11

These equations maintain consistency with CFD results (flat bed). To increase the level of resemblance with CFD results and engineering experience, the exponent of T/h_t or $1-h_w/h_t$ is changed to -0.1 to

TRACC/TFHRC Y2Q2

reduce the rate of increase for y when h_t gets larger. In order to increase reliability of the design the scaling factor was subsequently modified from 1.7/4.13 to 0.5 yielding:

$$\frac{t}{h_b} = 0.5 \left(\frac{h_b}{h_u}\right)^{0.2} \left(\frac{h_t}{h_u}\right)^{0.2} \left(1 - \frac{h_w}{h_t}\right)^{-0.1}$$
2.12

This fit has the advantage of a simple form and no dependency on the velocity. With the safety factor it over-predicts the CFD results for the whole range of applicable values of h_t , as is required to ensure that scour is not under predicted. The quality of the fit to the experimental data was also good.



Figure 2.9 Equation 2.12 vs. CFD results

Approach (3): Balanced consistency between velocity and length scale. A power of -0.45 produces a considerable effect from velocity, known to be physically unrealistic. To reduce this effect, one can reduce the exponent of F_r and R_e proportionally (keeping the power of h_u zero so there is no scaling issue). If they are cut in half, the equations become:

$$\frac{t}{h_b} = K \left(\frac{V}{\sqrt{gh_u}} \right)^{-0.15} \left(\frac{\rho V h_u}{\mu} \right)^{-0.075} \left(\frac{h_b}{h_u} \right)^{0.2} \left(\frac{h_t}{h_u} \right)^{0.2} \left(1 - \frac{h_w}{h_t} \right)^{-0.1}$$
2.13

This expression has much less velocity dependence, but does not fit experimental data very well.

TRACC/TFHRC Y2Q2



Figure 2.10 Equation 2.13 at different velocity levels (K=0.95) vs. CFD results

Thorough analysis of the possible solutions led to the conclusion that the best choice is the equation having no dependence on the upstream velocity. If it comes to the quality of the fits to the experimental data all the presented choices gave similar fit and reliability indicators. Figure 2.11 shows design equation of approach 2 compared to the experimental data sets. Obtaining good fits of experimental data sets using any of the design equations in the various approaches could be due to the fact that the experimental data did not cover the whole possible range of applicable cases and were all done at laboratory scale. The direction of future experiments to further improve the design equation should be aimed to cover cases outside of the already tested domain. Such tests would likely greatly improve the reliability of the fitted formula. The equation without the velocity dependence provided the simplest form, which makes it easy to apply for the engineers. Also locking it at one specific level assures that the unverified inverse dependency on velocity would not lead to unsafe prediction of *t* for some unverified cases. Equation 2.12 was thus incorporated to the new HEC-18.

Figure 2.11 shows the comparison between scour depths predicted by Equation 2.12 ("Design Scour Depth") with the measured scour depth from experiments. The predicted value is safely greater than observed scour without excessive conservatism.



Figure 2.11 Comparison between predicted scour (design scour depth) and experimentally measured scour depth

2.1.4. References

- 1. *"Evaluating Scour at Bridges,"* Hydraulic Engineering Circular No. 18, FHWA-HIF-12-003, April, 2012.
- 2. Arneson, L. A., and Abt, S. R., "Vertical contraction scour at bridges with water flowing under pressure conditions." Transportation Research Record. 1647, 10-17, 1998.
- 3. Shan, H., Xie, Z., Bojanowski, C., Suaznabar, O., Shen, J., Lottes, S., and Kerenyi, K., *"Submerged-Flow Bridge Scour under Clear-Water Conditions,"* FHWA Report, DTFH61-11-D-00010, 2011.
- 4. Umbrell, E. R., Young, G. K., Stein, S. M., and Jones, J. S., "*Clear-water contraction scour under bridges in pressure flow*". J. Hydraul. Engrg. 124(2), 236-240, 1998.

2.2. Modeling of the Wind Tunnel Laboratory at TFHRC

The geometry files provided for the initial development of the wind tunnel and room model indicated the pulley on the fan drive shaft was a solid disk. A visit to TFHRC in January, 2012, included a tour of the wind tunnel laboratory, and during that visit, it was noted that the pulley was not a solid disk but was open with six spokes. The model is currently being modified to have a pulley that matches the 6 spoke wheel of the laboratory, and the model is being moved from version 6.04 to 7.02 of STAR-CCM+. The open, spoked pulley on the side of the wind tunnel with the least resistence for return air flow is expected to increase the asymmetry in the test section in the simulations by a small amount. Results will be reported in the next quarter report.

2.3. 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.

2.3.1. 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.

2.3.1.1. 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:

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

Table 2.1: Boundary conditions

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 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 xdirection and v-direction. The results were plotted in color-coded contour. Figure 2.13 through Figure 2.33 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 2.19, 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.

Velocity	1.1′/s			0.71′/s		
Flow Depth	4.5″	6″	9″	4.5″	6″	9″
Sediment						
elevation						
0 D	Figure 2.13	Figure 2.14	Figure 2.15	Figure 2.16	Figure 2.17	Figure 2.18
			Figure 2.19			
0.15 D	Figure 2.21	Figure 2.22	Figure 2.23	Figure 2.24	Figure 2.25	Figure 2.26
0.3 D	Figure 2.28	Figure 2.29	Figure 2.30	Figure 2.31	Figure 2.32	Figure 2.33

Table 2.2 Contour plots comparing CFD, PIV, and ADV. All figures compare CFD against PIV except Figure 2.19, which compares CFD against ADV.

(1) Bed elevation at 0 inch

Figure 2.12 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 2.13 through Figure 2.18 are taken from a trough section, i.e. the largest cross section.



Figure 2.12 Sketch of experimental model (left) and CAD model of culvert section (right) for bed elevation at 0 inch



Figure 2.13 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 2.14 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 2.15 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 2.16 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 2.17 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 2.18 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 2.19 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 2.20 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 2.20 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 2.20. Results were compared in Figure 2.21 through Figure 2.23. 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 2.21 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 2.22 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 2.23 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 2.24 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 2.25 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 2.26 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 2.27 shows the sketch of the experimental model (left) and the CAD model of culvert section geometry (right).



Figure 2.27 Sketch of experimental model (left) and CAD model of culvert section (right) under the situation of bed elevation at 10.8 inch



Figure 2.28 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 2.29 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 2.30 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 2.31 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 2.32 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 2.33 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)

2.3.1.2. 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:

TRACC/TFHRC Y2Q2

$$RMSD_{area} = \sqrt{\frac{\sum_{area} (V_1 - V_2)^2}{n_{area}}}$$
 2.14

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:

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:

Bed	Velocity	Water	PIV and CFD		ADV and CFD	
elevation(inch)	(fps)	depth	RMSD(cm/s)	Relative	RMSD(cm/s)	Relative
		(inch)		error(%)		error(%)
		4.5	5.185	23.96	2.545	11.76
	0.71	6	4.337	20.04	4.043	18.68
0		9	4.317	19.95	4.116	19.02
		4.5	9.515	28.38	7.838	23.38
	1.1	6	6.227	18.57	3.177	9.48
		9	7.316	21.82	6.241	18.61
		4.5	2.654	12.26	2.572	11.89
	0.71	6	4.520	20.89	2.327	10.75
5.4		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

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

Bed	Velocity	Water	PIV and CFD		ADV and CFD	
elevation(inch)	(fps)	depth	RMSD(cm/s)	Relative	RMSD(cm/s)	Relative
		(inch)		error(%)		error(%)
		9	3.559	10.61	3.448	10.28
	0.71	4.5	4.386	20.27	5.452	25.19
		6	3.929	18.16	3.552	16.41
10.8		9	2.77	12.80	3.458	15.98
		4.5	6.686	19.94	7.834	23.36
	1.1	6	5.497	16.39	8.747	26.09
		9	3.764	11.23	6.435	19.19

2.3.1.3. 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.

2.3.2. References

 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 assessment 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

2.4. Modeling of Truck Generated Salt Spray under Bridge with Sliding Mesh

The truck model is currently being updated with mud flaps and tested in STAR-CCM+ version 7.02. A number of refinements to the mesh models and particle parcel tracking models are also being investigated to improve accuracy and computational efficiency. Updates to the models done in this quarter:

- The domain of the model was extended each direction
- The mesh around the truck was restructured more efficient use of small cells through multiple definitions of the volumetric controls
- Adding mudflaps to the model
- The simulations have been speeded up through redefining the interaction of water droplets with boundaries: rebound from mudflaps, stick to the bridge superstructure but escape for the other wall boundaries. That way hundreds of thousands trajectories of particles on boundaries are not computed.

2.5. New Flume Design for Hydraulics Laboratory at TFHRC

The large tilting flume at TFHRC will be replaced with a high-speed flume with a live-bed scour testing capability. The flume consists of several sections. The test section is a straight channel with a depression

that can be filled with bed material. A long stretch of straight channel is provided before the test section to allow the development of a proper boundary layer for the test section. An inlet that may include a main pipe reception, a diffuser, and a trumpet is connected to the long straight section to feed a wellconditioned flow into the channel. The inlet needs to be designed to provide a sufficiently uniform velocity profile at the entrance of the long straight section. CFD simulation is employed to optimize the geometry to accomplish the desired flow condition under high discharge rate. During this quarter, the study focuses on the trumpet section that serves as a transition between the enlarged area after the diffuser and the long straight section that leads to the test section.



Figure 2.34 A conceptual drawing of the preliminary inlet design

When the ease of construction is a primary consideration, the transitional section of the trumpet may be built with a number of straight panels to form an approximated curve. This may or may not create a detrimental effect on the flow. A more physically sound but potentially expensive option is to build the trumpet with continuously curved panels. Such a "streamlined" construction is compared with the multi-panel option by CFD modeling. The geometrical configurations of the models for the two options are shown in Figure 2.35 (a) and (b). These models consist of one half of the flume using symmetrical boundary conditions at the centerline.



Figure 2.35: Two different transition profiles of trumpet: streamline and multi-line

The first image in Figure 2.36 shows the velocity magnitude in a few cross-sections in the trumpet with smoothly curved transition. The second image in Figure 2.36 shows the velocity magnitude in the trumpet built with multi-linear panels. The smooth curve produces a fairly uniform velocity throughout the cross-section at the end of the trumpet. The trumpet with multi-linear transition produces a velocity distribution that is similar to that by a smoothly curved trumpet in general. However, some local high velocity near the corner is observed. The local high velocity is especially clear for the last three sections shown in Figure 2.36.



Figure 2.36: Comparison of velocity uniformity along different sections

Figure 2.37 shows the cross-sectional velocity contour at the potential testing section in the main channel. Figure 2.37(a) is the result from a smoothly curved trumpet, while Figure 2.37(b) is from a multi-linear trumpet. It illustrates that the "streamlined" version of the trumpet design produces a more uniform velocity distribution than the multi-linear one does along a long stretch of the straight channel downstream.



(a) Streamline trumpet transition case



(b) Multi-linear trumpet transition case



The configuration of honeycomb located between the diffuser and the trumpet plays a significant role in the orientation of the water streamlines. The simulation with and without the honeycomb for streamlined trumpet case is shown in Figure 2.38. The flow streamlines with the configured honeycomb in Figure 2.38(a) obviously have more uniform velocity and stay more coherent than the other case that does not have a honeycomb. Figure 2.38(b) shows a potential recirculation zone that consists of high variation in velocity and significantly warped streamlines. This may reduce the efficiency of the inlet and contribute to the increase of non-uniformity of the flow predicted in the main channel.



(b) Without honeycomb

Figure 2.38: Comparison between with-honeycomb and without-honeycomb for the streamline trumpet

CFD work in this quarter investigated the hydraulic performance of the trumpet in the preliminary inlet design for the new Sediment Recirculation Flume at TFHRC. It concluded that the streamlined trumpet profile created a more uniform flow condition than the multi-linear panel option did. This might justify the use of the more expensive smoothly curved trumpet design. A honeycomb between the diffuser and the trumpet also offered significant benefit in the flow condition in the trumpet area. The honeycomb property used in this study would be translated to physical specifications and used to assist the acquisition of the necessary parts.

The trumpet study focuses on the half of the flume inlet that feeds water to the main channel. In the next quarter, the diffuser and the geometry of the front end of the inlet will be studied.

2.6. Training and Technology Transfer

Technology transfer of high performance computational analysis techniques is an important part of Argonne's work to support and advance engineering and research programs at TFHRC. The technology transfer is accomplished by publishing the work done and techniques developed in reports and papers, presentations at conferences, and training courses in CFD offered by TRACC. The second training course in the application of CD-adapco's STAR-CCM+ CFD software transportation related problems was conducted on March 21-22, 2012. The course has evolved to be nearly entirely hands on tutorials that focus on the latest techniques for solving problems in hydraulics and wind engineering. Two new tutorials were developed for the March training, and the mesh morphing was expanded to include a scour model. A number of the problems of interest include motion of objects in the flow domain and the need to have a moving and possibly deforming mesh. These types of problems are some of the most challenging to set up and solve. The two new tutorials focused on these techniques. One included a propeller in a water reservoir with a free surface. The other uses sliding meshes to move a truck model under a bridge at 60 mph to analyze the distribution of droplet spray coming off of the tires on salted wet roads. These tutorials provided training in newly developed models that are a focus of ongoing research to the attendees.

Participants were provided with trial licenses and software by CD-adapco to work on the tutorials during the training days and experiment with the software for about two weeks after the course. Course attendees included TFHRC research staff, state DOT researchers, professors, and students from several universities. About fifteen people attended at the TRACC facility and about an equal number participated online via Adobe Connect. The training announcement is shown in Figure 2.39 and Figure 2.40.



Figure 2.39: Announcement for CFD training course held in March

STAR-CCM+ Training Sessions

Wednesday-Thursday March 21-22, 2012 9:30 AM-4:30 PM (CST)

General Topics

Introductions and Agenda **Basics of Computational Fluid Dynamics** STAR-CCM+ CFD Software Basics **STAR-CCM+** features Graphical User Interface Workflow Meshing with STAR-CCM+ **Post-Processing Basics Daily Schedule** Becoming a TRACC User **Useful Software** Sessions will start at 9:30 AM (CST) File Transfer (WinSCP) and end at 4:30 PM. **Desktop Virtualization (NoMachine)** Lunch will be from 12 PM till 1:15 PM. Others There will be a 10 minute break in How to use STAR-CCM+ on the each half day session. **TRACC Cluster Training Location Map** Hydraulics and Aerodynamics **Tutorials** Trial licenses and STAR-CCM+ software will be provided to R participants to carry out a number of tutorials under the guidance of the instructor. Tutorials will include: **Gravity Driven Flow** Forces on a Flooded Bridge Deck Wind Loading on Roadside Signs Flow Through a Lab Scale Culvert **River Bed Erosion / Mesh Morphing Propeller with Free Surface Model** Truck Tire Spray under Bridge using Sliding Mesh Map of Argonne campus Contact > 630-252-5290 | CFD_TRACC@anl.gov | www.tracc.anl.gov Argonne ENERGY

