Transportation Research and
Analysis Computing Center (TRACC)
Year 3 Quarter 4 Progress Report

Section on CFD Modeling of Flow through Culverts

Principal Investigator:
David P. Weber, Ph.D.

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

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

CFD Investigators:
Vishnu Vardhan Reddy Pati
Milivoje Kostic, Ph.D.
Pradip Majumdar, Ph.D.

Northern Illinois University

Submitted to:

Ms. Dawn Tucker-Thomas
Office of Research Development & Technology
Research and innovative Technology Administration
1200 New Jersey Avenue, SE, Building E 33-464
Washington, D.C. 20590

January, 2010

Introduction

Argonne National Laboratory initiated a FY2006-FY2009 multi-year program with the US Department of Transportation (USDOT) on October 1, 2006, to establish the Transportation Research and Analysis Computing Center (TRACC). As part of the TRACC project, a national high performance computer user facility has been established, with full operations initiated in March 2008. The technical objectives of the TRACC project include the establishment of a high performance computing center for use by USDOT research teams, including those from Argonne and their university partners, and the use of advanced computing and visualization facilities for the performance of focused computer research and development programs in areas of interest for USDOT.

These objectives are being met by establishing a high-performance computing facility, known as the Transportation Research and Analysis Computing Center (TRACC), and providing technical support for its use by USDOT staff and their university and industry contractors. In addition to facilities for advanced computing, visualization, and high-speed networking in the TRACC facility, advanced modeling and simulation applications research is being conducted by the TRACC facility scientific applications staff in coordination and collaboration with USDOT researchers.

The first quarter project report for Year 4 of the project (Y4Q1) summarizes progress on the principal activities associated with the operation of the computing center and in the performance of the computational research in the four key application areas identified by USDOT as its highest priorities. As defined by the Year 4 Statement of Work (SOW) the activities and objectives for the fourth year of the project are: (1) traffic modeling and simulation and emergency transportation planning; (2) computational fluid dynamics for hydraulics and aerodynamics research; (3) multi-dimensional data visualization; and (4) computational structural mechanics applications. This section of the report summarizes the progress on computational fluid dynamics modeling and analysis of flow through culverts.

The establishment of the high performance computing center based on a massively parallel computer system and the transportation research and demonstration projects associated with key focus areas include the use of computing facilities as well as the exchange of research results with the private sector and collaboration with universities to foster and encourage technology transfer at the DuPage National Technology Park (DNTP). Argonne university partners include the University of Illinois and Northern Illinois University.

Computational Fluid Dynamics for Hydraulic and Aerodynamic Research

Scaled experiments conducted at the Turner-Fairbank Highway Research Center (TFHRC) hydraulics laboratory are being used to establish the foundations of a CFD-based simulation methodology in hydraulics analysis of bridges and other structures, including the assessment of lift and drag forces on bridge decks, pressure scour under flooded bridge decks, and analysis of flow through culverts. Scour modeling includes analysis of bed stresses and their influence on scouring, and evaluation of active or passive scour countermeasures. Addressing environmental issues such as fish passage through culverts is also a part of the program.

Modeling and Analysis of Flow through Culverts

A culvert is a conduit used to enclose a flowing body of water. They are often corrugated for strength and may be used to allow water to pass underneath a road, railway, or embankment. They may carry flood waters, drainage flows, and natural streams below earth fill and rock fill structures. From a hydraulic perspective, a dominant feature of a culvert is whether it runs full or not. Culverts come in many shapes and sizes, including round, elliptical, flat-bottomed, pear-shaped, and box. They vary from the small drainage culverts found on highways and driveways to large diameter structures on significant waterways or supporting large water control works. The Federal Highway Administration (FHWA) is conducting experiments on culverts to provide designers with better information to allow for fish passage in the design. Several key parameters considered are: design approach, culvert slope, culvert geometry, stream width, and passage performance. CFD analysis of FHWA culvert experiments is being done to help experimentalists in experimental design and in understanding of experimental results. In the future determination of culvert design parameters using CFD methods validated against experiment may allow CFD determination of parameters to replace experimental determination.

The culvert model considered in this study is a quarter portion of the circular cross section of a culvert having spiral corrugations as shown in Figure 1. This configuration was used in an initial set of experiments conducted at the Turner-Fairbank Highway Research Center (TFHRC). Flow through the culvert with different discharge rates and with different water level depths are considered. Figure 2 shows the computational mesh along the mid plane.

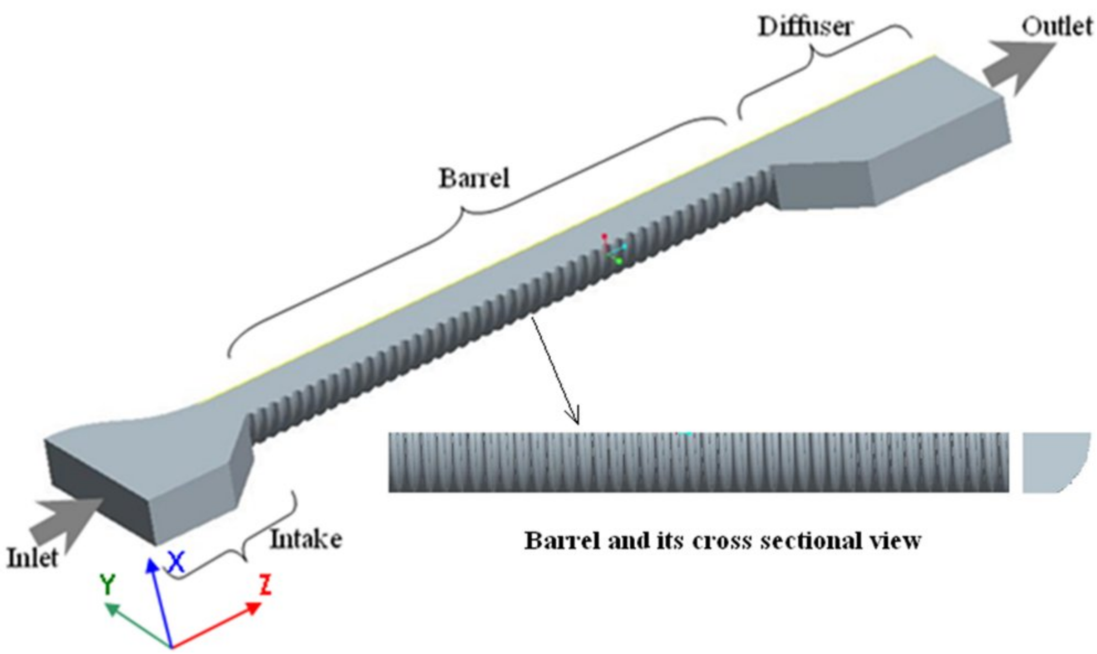

Figure 1. CAD Model of Flume

In order to economize and use finer mesh size in the water region, the computational domain was truncated in the air region and a new domain was created. Simulations were performed for two different mesh types. One domain had a finer mesh (around 2 million cells) and the other had a coarser mesh (around 0.5 million cells).

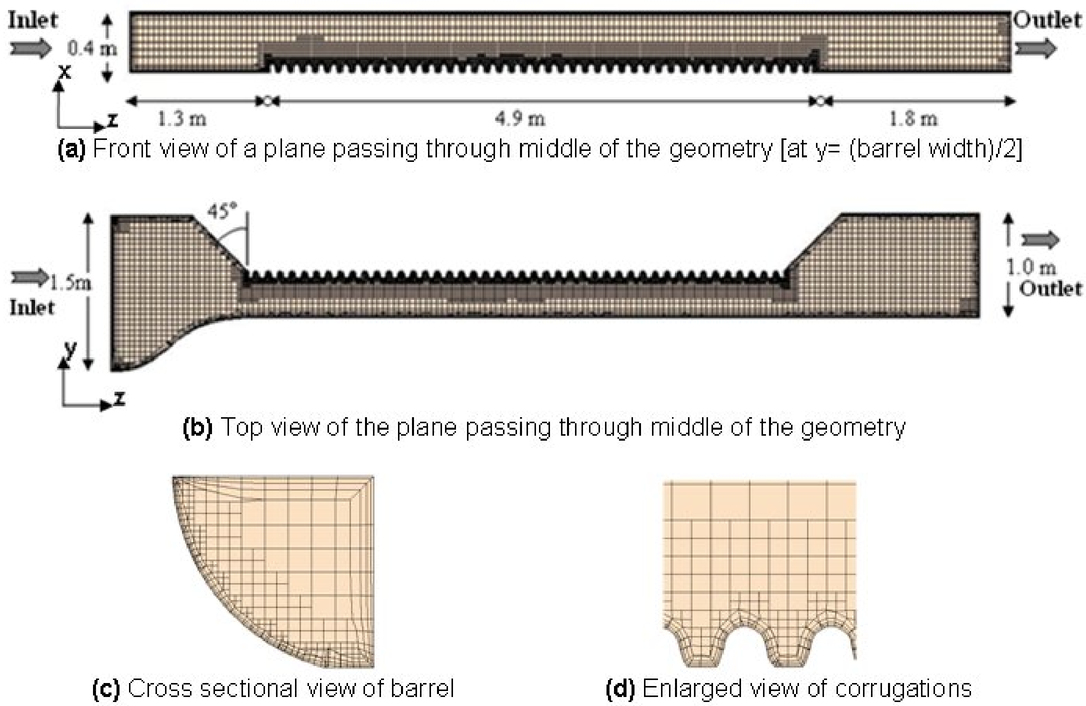

Figure 2. Computational mesh along a plane passing through mid-section of the barrel

Table 1 shows six different discharge rates and different water level depths that were analyzed in this study. Exit water level depth from the simulation results is also listed. The water/air interface is taken to be a volume fraction of water equal to 0.8.

Figures 3 and 4 show variation in the water volume fraction and velocity contours at different sections along the length of the flume for a discharge flow rate of 3.7 L/s. An enlarged view of these plots is also presented in Figure 5. Figures 6 and 7 show details of the free surface and water depth with corresponding velocity variation along a longitudinal plane at the middle section of the flume. A free surface profile was created for all configurations based on the condition of 0.8 VOF value, and velocity contour plots are plotted on this free surface. The interface between air and water in these plots appears to be smeared out because the grid is not sufficiently fine to resolve this interface sharply. In 3D simulations using a sufficiently fine grid to improve the resolution of the water depth would require greatly increased computational resources. These additional resources can be applied to the CFD analysis of the next set of culvert analysis experiments at TFHRC.

Table 1. Flow conditions and configurations

Configuration No.	Discharge Q [L/s]	Inlet velocity V_{in}[m/s]	Initial water depth h_{in} [mm]	Exit water depth h_{es} [m]Simulation
1	1.8	0.0115	108	0.00495
2	2.5	0.0156	111	0.00588
3	3.2	0.01924	115	0.00678
4	3.7	0.02205	116	0.00734
5	4.8	0.0284	117	0.00814
6	5.9	0.0343	119	0.00910

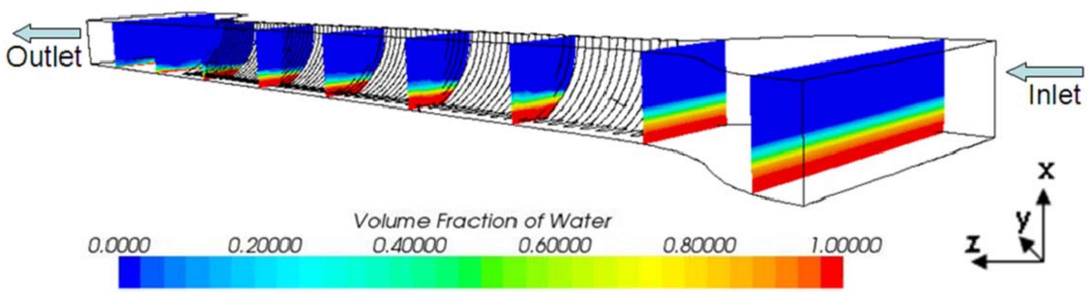

Figure 3. Volume fraction of water at different cross sectional planes along the direction of flow (for discharge of 3.7 L/s)

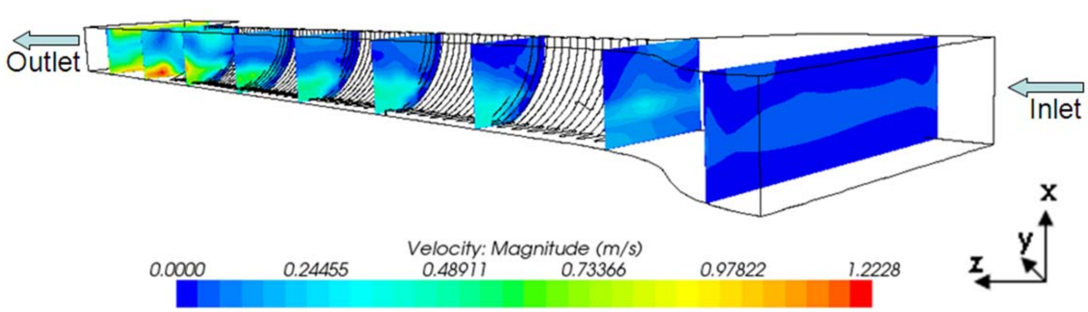

Figure 4. Velocity contour plots at different cross sectional planes along the direction of flow (for discharge of 3.7 L/s)

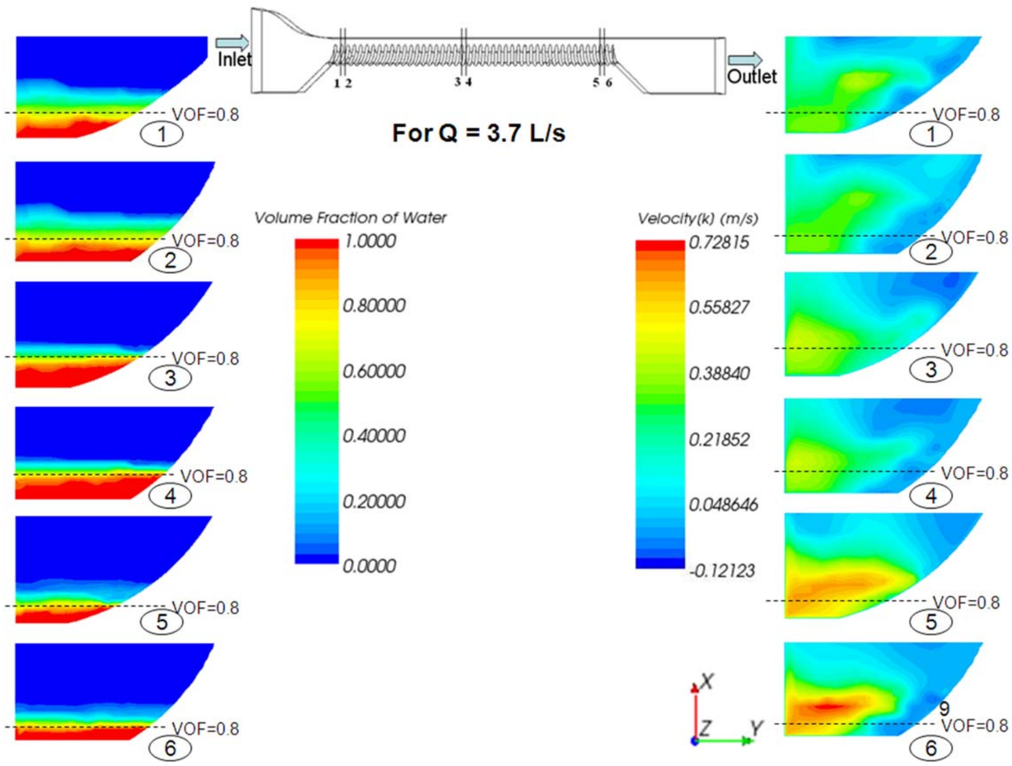

Figure 5. Volume fraction of water and velocity contour plots at different cross sections (for discharge of 3.7 L/s)

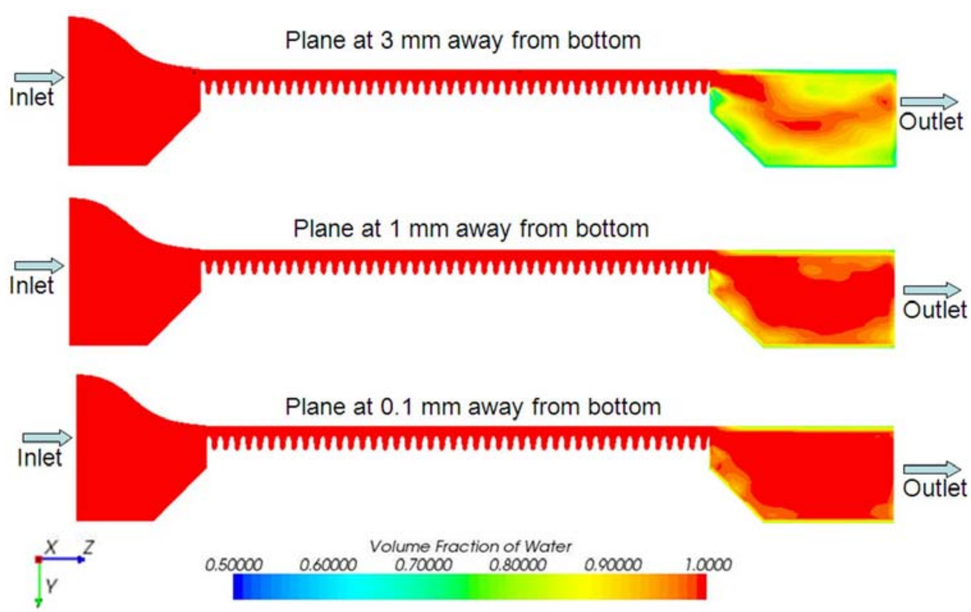

Figure 6. Volume fraction of water (for discharge of 3.7 L/s)

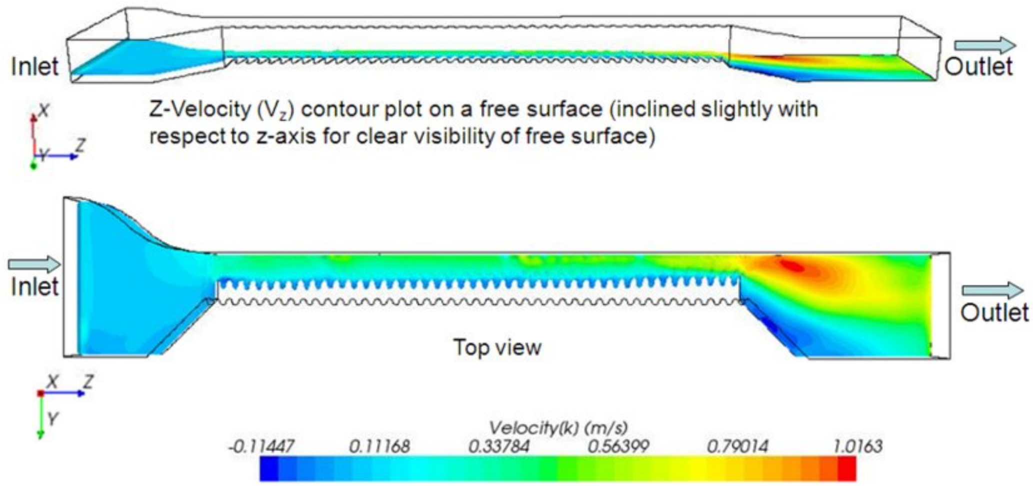

Figure 7. Velocity scalar contour plot on free surface (for discharge of 3.7 L/s)

Similar results are also obtained for flow rates of 1.8 L/s , 2.5 L/s, 3.2 L/s, 4.8 L/s, and 5.9 L/s. A total pressure plot at the culvert bottom for all of the different discharge rates was plotted and compared in Figure 8. Results show variation in pressure drop in the flume with different flow rates. The bottom pressure is primarily hydrostatic and therefore also corresponds to the flow depth. The trend shown appears to be reasonable.

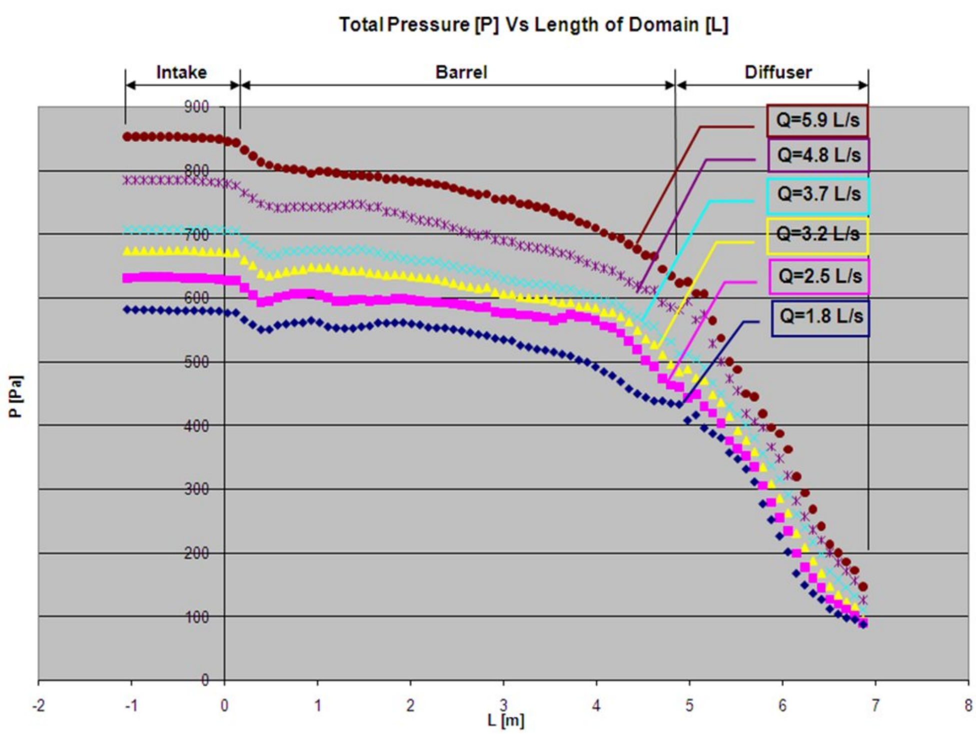

Figure 8. Total pressure vs. length of domain for different configurations