Transportation Research and
Analysis Computing Center (TRACC)
Year 4 Quarter 2 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

March 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 second quarter project report for Year 4 of the project (Y4Q2) 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. It may be used to allow water to pass underneath a road, railway, or embankment. It is a hydraulic structure and it may carry flood waters, drainage flows, and natural streams below earth fill and rock fill structures. From a hydraulic viewpoint, a dominant feature of a culvert is whether the flow through it runs with a full or partial cross-section. Culverts come in many shapes and sizes, including round, elliptical, flat-bottomed, pear-shaped, and boxed. 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. Culverts tend to be preferred over bridges because they cost less to build and maintain.

In periods of rapid development the need for a new infrastructure often overshadows concerns of potential environmental impacts. Streams have been straightened and channeled through pipes, and culverts have been sized without considering future impacts on fish migration. As a result there has been a deterioration of freshwater habitat, and the endangerment of many fish species. In recent years a movement towards restoring freshwater ecosystems previously impacted by human activity has intensified. As water runoff volume increases, the streams actively degrade waterways and may interrupt natural fish migration. Culverts are unable to adapt to the degrading streams and instead become barriers to fish movement. The most common reasons culverts become barriers are excessive outlet drops, high water velocity within the culvert, turbulence within the culvert, accumulation of sediment and debris, and an inadequate water depth within the culvert. In general, the optimum design for peak flow conveyance will not meet fish passage criteria at all discharges. Fish size appears to have little influence on ability to negotiate a culvert despite its effect on swimming performance. One theory is that smaller fish utilize regions of low velocity near the culvert wall.

The Turner Fairbank Highway Research Center (TFHRC) is conducting experiments on culverts to provide designers with better information to improve culvert design. Major hydraulic criteria influencing fish passage are: flow rates during fish migration periods, fish species, roughness, and the length and slope of the culvert. In this study a simulation model is developed using the commercial CFD software, STAR-CCM+, with the two-phase VOF model (water and air) and the results are validated with the experimental data obtained from TFHRC. The Culvert model considered in this study is half of the cross section of the culvert having spiral corrugations as shown in Figure 1. This configuration is used in experimental evaluation of the culvert at TFHRC. The experimental setup of the flume is also shown in the figure.

Based on the dimensional details provided by TFHRC, a CAD model, as depicted in Figure 1, has been created in Pro-ENGINEER and is imported to STAR-CCM+ in IGES (Initial Graphics Exchange Specification) file format. This CAD model consists of three parts: the intake (also called the inlet), the barrel (or the throat or the corrugated portion) and the diffuser (also called outlet). The barrel consists of spiral corrugations throughout its length.

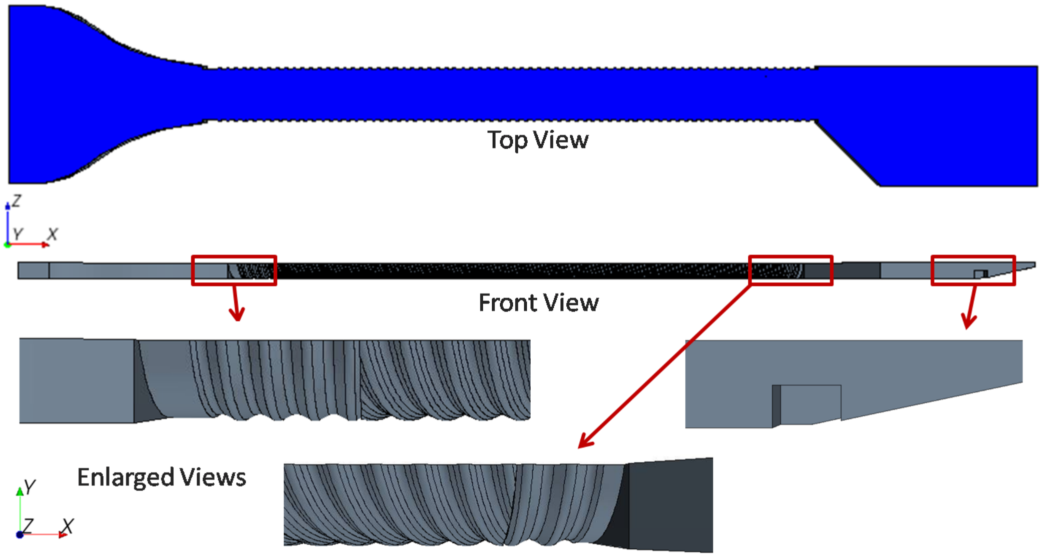

Figure 1. CAD model of culvert flume

In order to economize and use finer mesh size in the water flow region, a relatively small air region was used.

The VOF method captures the free surface profile through use of the variable known as the volume of fluid, which is defined as the ratio of the heavy fluid phase volume to total volume of the computational cell and is derived by solving an additional transport equation along with the conservation of mass and momentum. All properties and field variables are characterized as volume averaged values. Thus every computational cell signifies variables and properties of purely one phase or a mixture of heavy and light (water and air) phases depending upon the volume fraction.

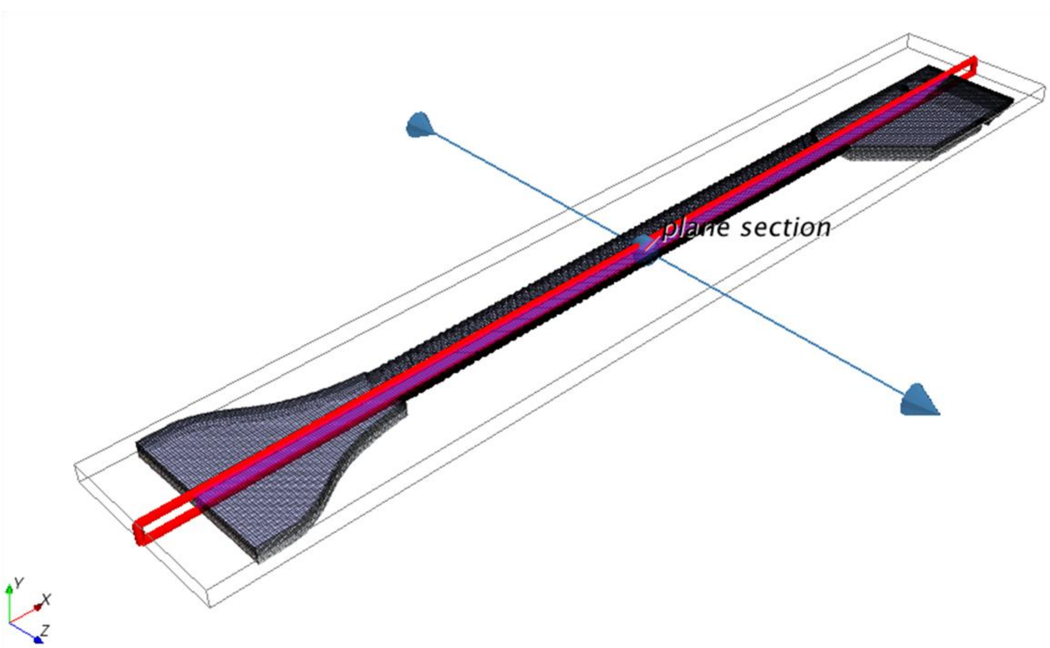

Figure 2. Isometric view of the mesh scene with a plane section passing through the center line

A plane section was created as shown in Figure 2 above passing through the center line of the barrel and the mesh on that plane as shown in Figure 3 below.

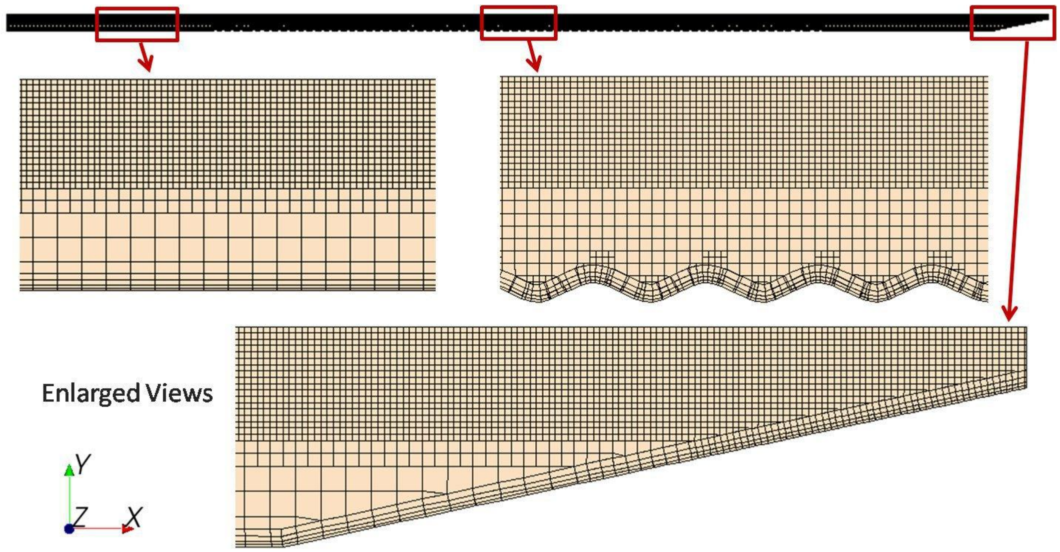

Figure 3. Mesh scene shown on a section plane passing through center plane

A simulation was done with a discharge of 2.5 L/s and an angle of 12.34 degrees for the flap gate at the exit. A plot of the volume fraction of water shown on a section plane is presented in Figure 4. Water level depth along the center plane was calculated and compared with the experimental data obtained from TFHRC. Simulation results are within 10% of experimental data as shown in Figure 3.2.18.

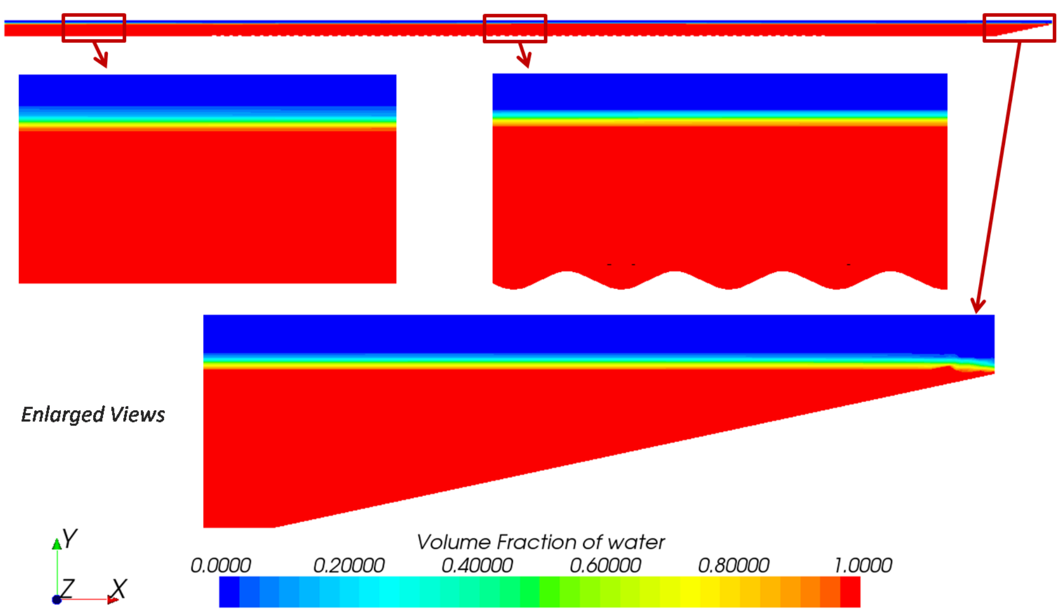

Figure 4. Volume fraction of water shown on a section plane

The axial velocity contour plot shown in Figure 5 shows that the axial velocity in the barrel (corrugations region) is higher because flow cross section is small there.

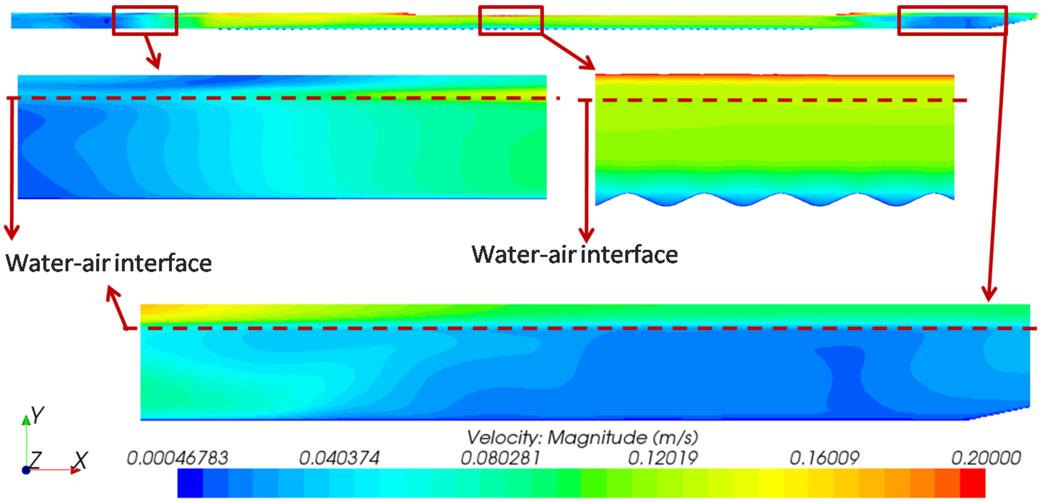

Figure 5. Velocity contour plot shown on a section plane

The volume fraction of water and velocity contour plots are plotted on different cross sectional planes as shown in Figure 6. In this relatively tranquil flow, there is not a lot of variation in stream wise velocity over the cross section.

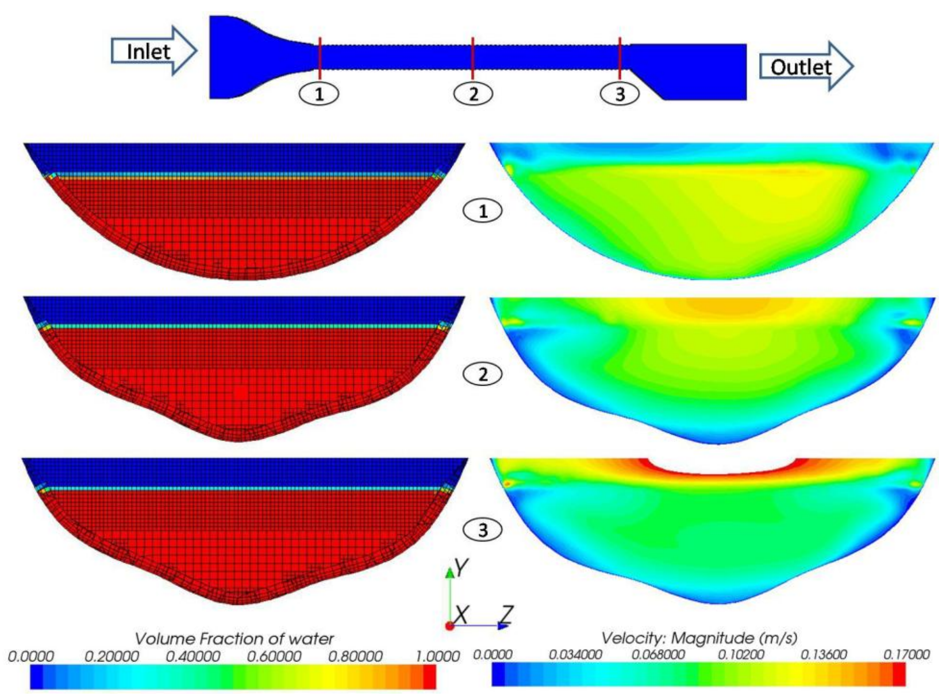

Figure 6. Volume fraction of water and velocity contour plots shown on three different cross sectional planes

Water level depth is calculated based on the pressure probe data presented in Figure 7. The water depth is calculated from static pressure obtained from a STAR-CCM+ line probe of pressure at points along the bottom of the flume and barrel. The fluctuations are a consequence of STAR-CCM+ not interpolating pressure to the coordinates of a probe point, but rather assigning the cell centroid value to it. The plot shows close agreement with the experimental data obtained from TFHRC.

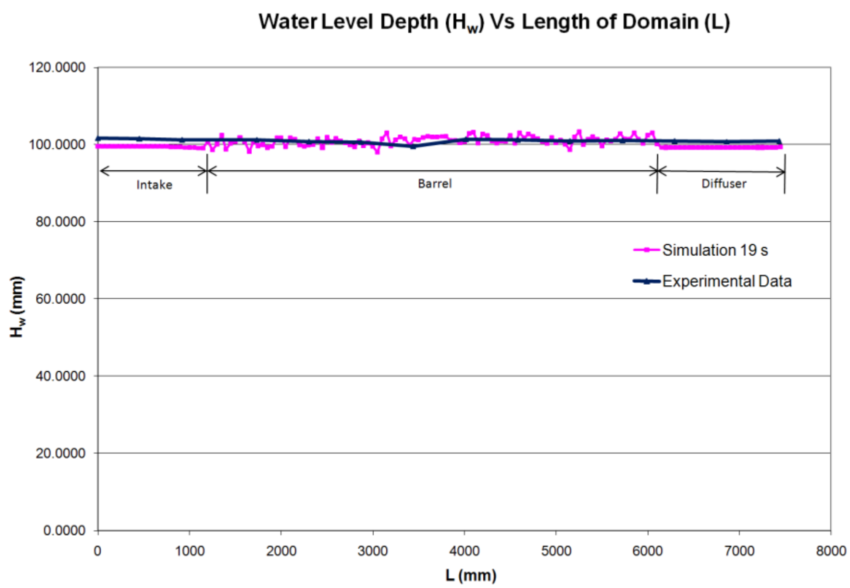

Figure 7. Comparison of simulation results for water level depth with the experimental data.

Work is continuing on cases with an intermediate flap gate angle and zero degree flap gate angle. These cases will not have a nearly uniform surface level.