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

October, 2009

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.

This fourth quarter project report for Year 3 of the project (Y3Q4) 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 3 Statement of Work (SOW) the activities and objectives for the third 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.

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 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 and pressure scour under flooded bridge decks. 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.

CFD Modeling of Flow through Culverts for Fish Passage:

A culvert is a conduit used to enclose a flowing body of water. They are often corrugated for strength. They 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.

The culvert model considered in this study is based on an initial set of culvert experiments in a flume at TFHRC. A quarter portion of the circular cross section of the culvert having spiral corrugations on it was used as shown in Figure 1. Simulation of flow through the culvert with different discharge rates and with different water level depths as shown in Table 1 is planned.

A CAD model was created based on the dimensional details provided by TFHRC and the simulations were performed for different conditions to compare with the experimental data to be provided by TFHRC. Pro-ENGINEER was used for creating the CAD model of culvert and was imported into STAR-CCM+ in IGES (*Initial Graphics Exchange Specification*) file format. The CAD model consists of three parts: the *intake* (also called the inlet), the *barrel* (or corrugated portion) and the *diffuser* (also called outlet). Simulations were performed for pipe-2 (i.e., water depth of 116 mm and discharge of 3.7 *L/s*) with spiral corrugations.

The geometry is 8.0 m long and a curvature of 0.45 m for the corrugations.

Table 1. Configurations of measurements the fish passage flume.

Pipe	Water Depth [*mm*]	Discharge [*L/s*]
1	115	3.2
2	116	3.7
3	111	2.5
4	108	1.8
5	117	4.8
6	119	5.9

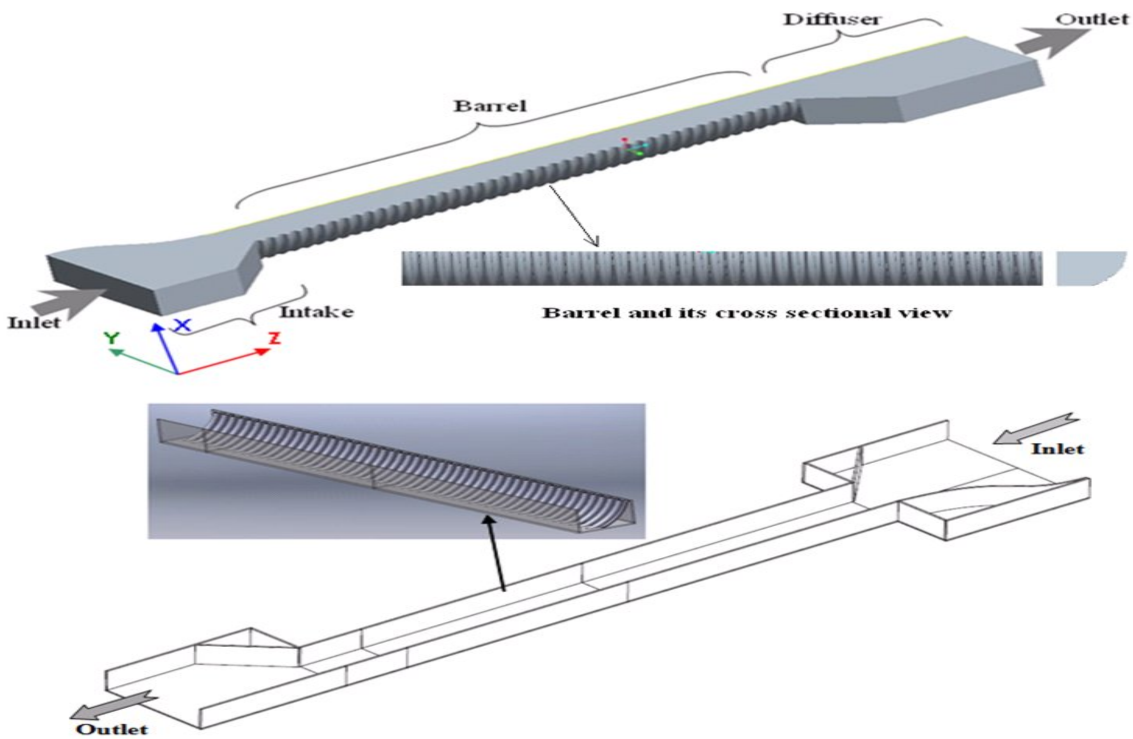
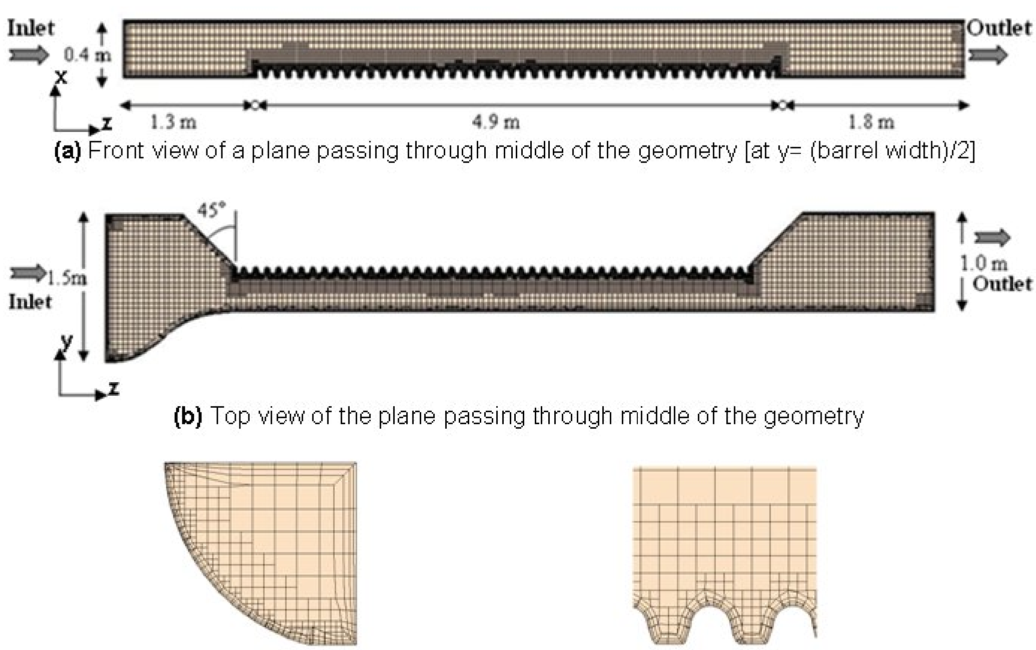


Figure 1: Different views of the fish passage flume (Intake, Barrel and Diffuser)

The Volume of Fluid (VOF) technique was used to simulate the free surface flow. The computational mesh for the domain is shown in Figure 2. For the two geometries the maximum cell size is 5.0 cm (core size) and the prism layer thickness near the wall is approximately 6.0 mm.



(a) Front view of a plane passing through middle of the geometry [at y= (barrel width)/2]

(b) Top view of the plane passing through middle of the geometry

(c) Cross sectional view of barrel

(d) Enlarged view of corrugations

Figure 2: Culvert flume dimensions and computational mesh

The analysis was set up as a three-dimensional transient using a k-epsilon turbulence model with wall function treatment. Boundary conditions are:

- Inlet water level: 0.116 m
- Inlet velocity magnitude: 0.022 m/s
- Volume fraction: composite (water and air) used to set inlet water level
- Pressure outlet boundary condition
- All other surfaces are defined as hydro-dynamically smooth no-slip walls.

Simulation results for the base case:

The volume fraction of water is shown in Figure 3. The initial condition for water level was not intended to be physically realistic and was set to uniform 0.116 m height equal to the inlet height, Figure 3 (a). Figure 3 (b) and (c) are the volume fraction of water after 25 seconds of simulation time. The flow depth drops through the corrugations, and after the corrugations there is a sudden drop in flow depth where flow expands into the outlet flow zone.

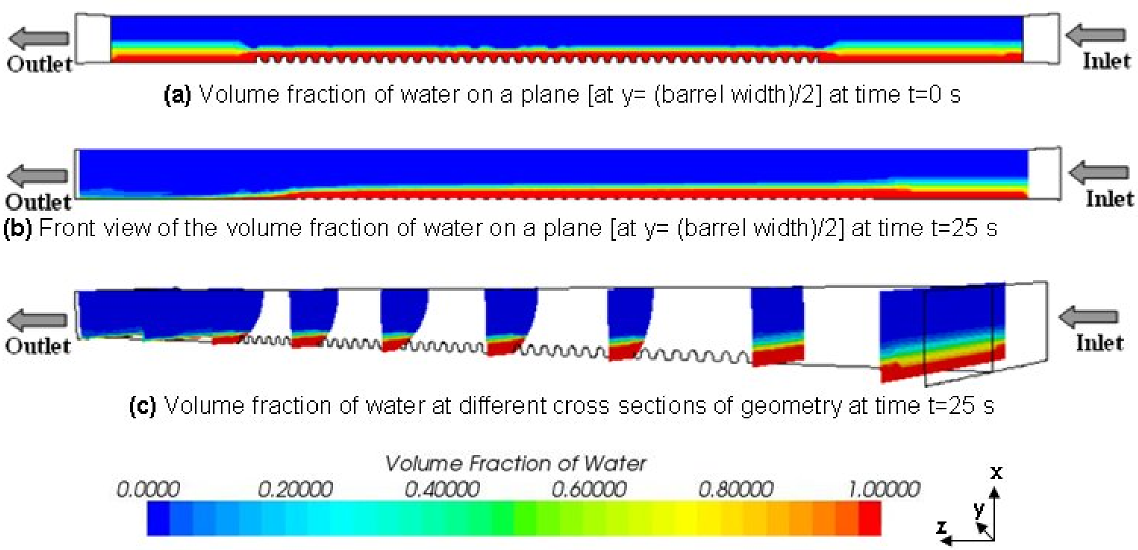
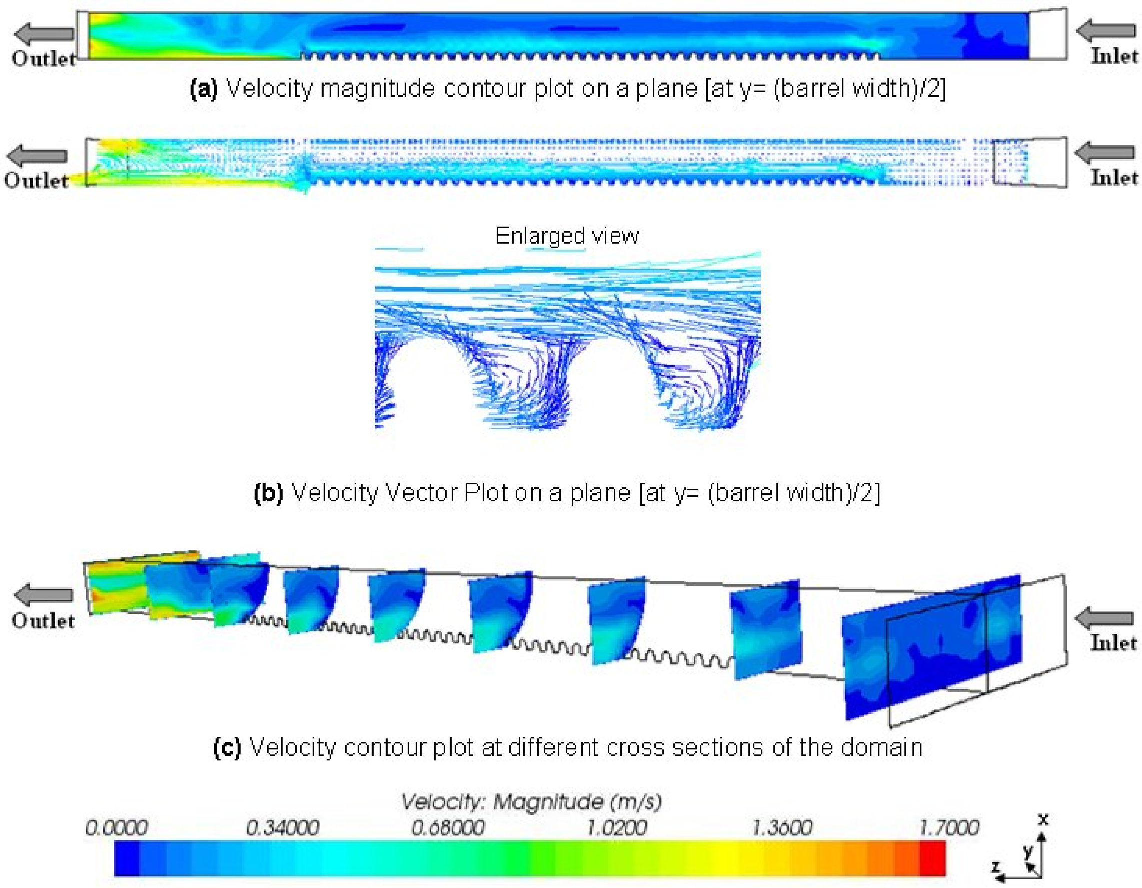


Figure 3: Volume fraction of water in culvert flume

Velocity vector and contour plots are shown in the Figure 4. Figure 4 (b) shows recirculation zones within the corrugations.



(a) Velocity magnitude contour plot on a plane [at y= (barrel width)/2]

(b) Velocity Vector Plot on a plane [at y= (barrel width)/2]

(c) Velocity contour plot at different cross sections of the domain

Figure 4. Velocity vector and contour plots in culvert flume