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

Section on CFD Modeling of Flow through Culverts

Principal Investigator:
Hubert Ley, 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

June 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 third quarter project report for Year 4 of the project (Y4Q3) 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 CFD-based simulation methodologies 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. When the development of a CFD analysis methodology is successfully completed, training in its use is added to the CFD training courses offered periodically by TRACC.

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 that 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 new infrastructure may overshadow concerns over 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. When water runoff volume is high, streams actively erode streambeds changing course and bathymetry of waterways and may interrupt natural fish migration. Culverts do not adapt to changing streams and can 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 for fish passage. 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+, and the two-phase VOF model with water and air. Simulation results are compared with the experimental data obtained from TFHRC. The culvert in this study is half of the cross section of a culvert barrel 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 Figure 2.

Based on the dimensional details provided by TFHRC, a CAD model, as depicted in Figure 1, has been created in Pro-ENGINEER and 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.

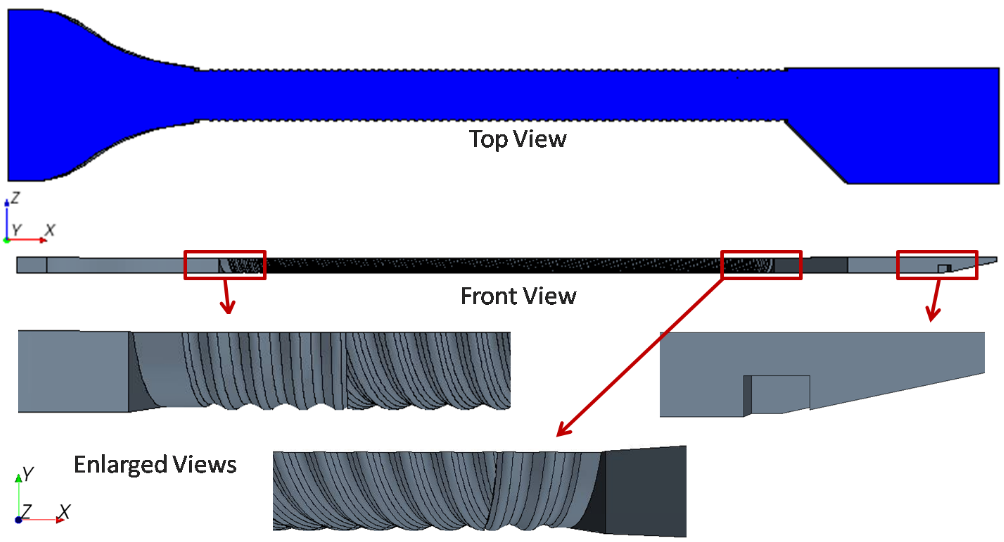

Figure 1. CAD model of culvert flume

In order to conserve computer resources and use finer mesh size in the water flow region, the computational domain was truncated in the air region and a new domain was created. 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 a computational cell and is derived by solving an additional transport equation along with the governing equations for conservation of mass and momentum. All properties and field variables are characterized as volume averaged values. Computational cells away from the free surface interface have a water volume fraction of either zero or one and the fluid material properties of the fluid in the cell are either those of air or water. The free surface may cut through computational cells and then those cells contain a mixture of the heavy fluid, water, and the light fluid, air, with properties that are a volume fraction weighted average of air and water properties.

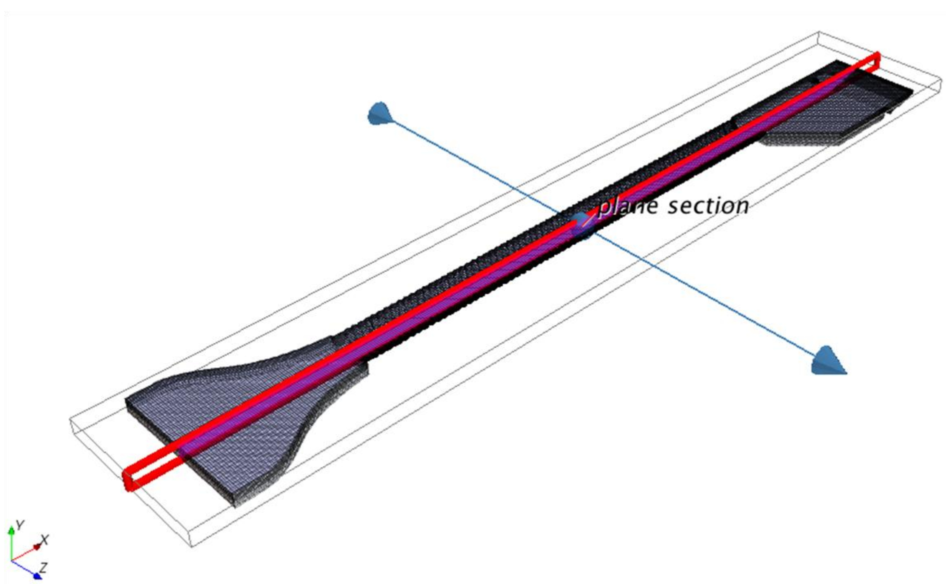

Figure 2. Isometric view of the mesh scene with a plane section passing through the center line.

In this reporting period, simulations of two cases were performed, one with a discharge of 8.6 L/s and a zero degree angle for the flap gate at the exit and the other with a discharge of 4.65 L/s and an angle of 11.006 degrees for the flap gate at the exit.

Case 1: discharge 8.6 L/s and zero degree angle for the flap gate at the exit

A stream wise plane section was created for visualization as shown in Figure 2 passing through the center line of the barrel and the corresponding mesh for this plane is as shown in Figure 3. Different blocks were created to refine the mesh in the water-air interface region to resolve the interface accurately.

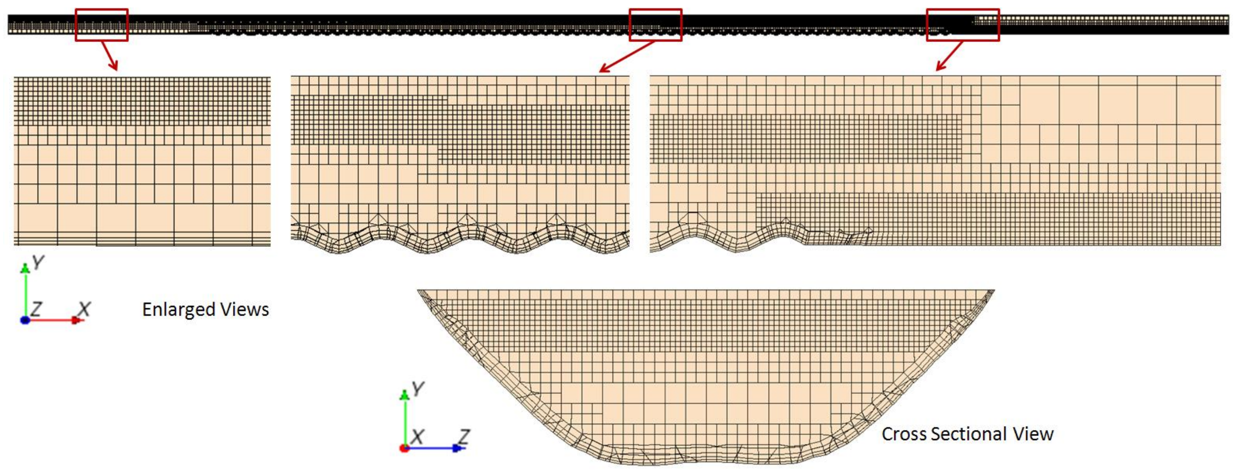

Figure 3. Mesh scene shown on a section plane passing through center plane

A plot of the volume fraction of water shown on the stream wise section plane is shown in Figure 4.

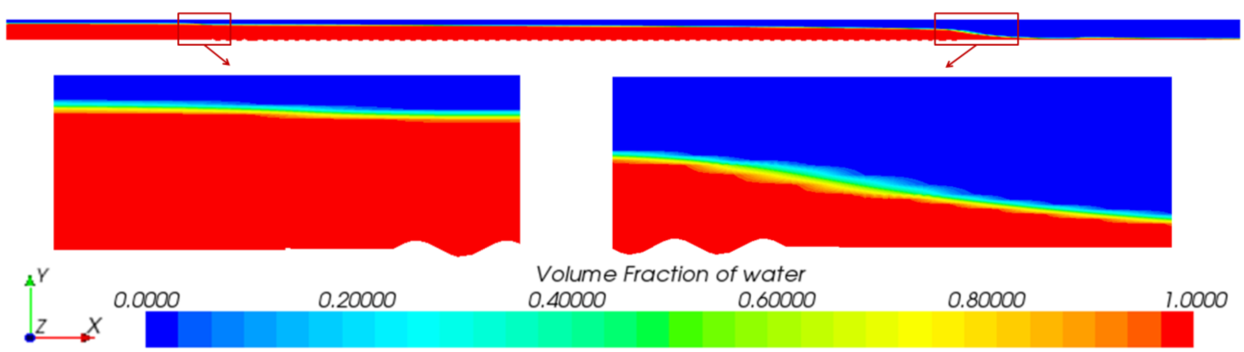

Figure 4. Volume fraction of water shown on a section plane

A velocity contour plot on the section plane is shown in Figure 5. The narrowing of the channel through the intake causes the flow to accelerate into the barrel. There is a transition to supercritical flow just beyond the end of the barrel as the flow accelerates through the diffuser toward a waterfall at the exit plane modeled with an atmospheric pressure boundary condition at the exit plane.

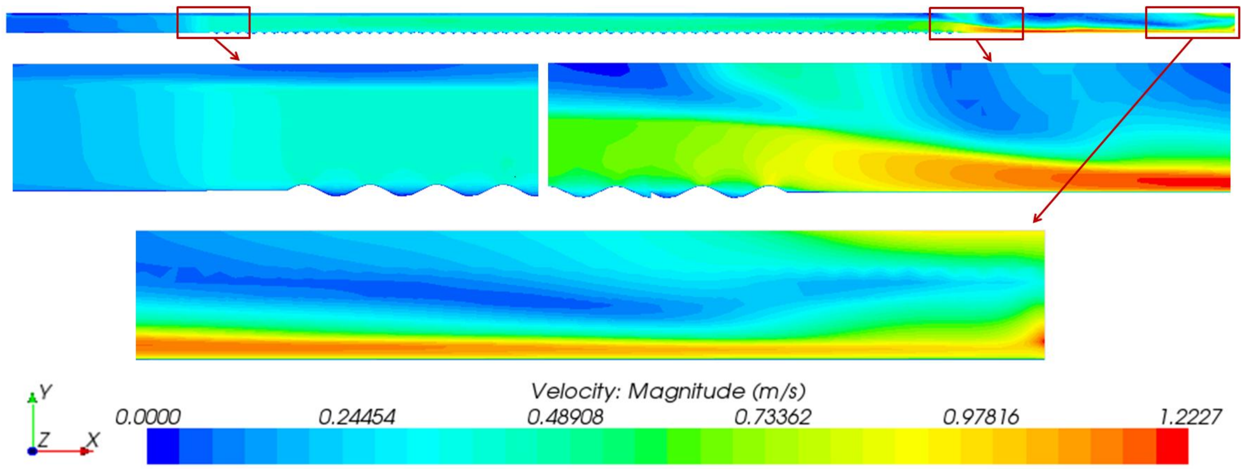

Figure 5. Velocity contour plot shown on a section plane

Volume fractions of water and velocity contours are plotted on different cross sectional planes as shown in Figure 6. As the flow moves through the barrel the zone of maximum velocity in the cross section shifts toward the right as viewed from the upstream. When the flow enters the diffuser, the channel expands toward the right and the flow accelerates through a turn towards the right wall. This asymmetric diffuser geometry appears to cause the shift towards the right of the zone of maximum velocity in the cross section of the barrel as shown in Figure 6.

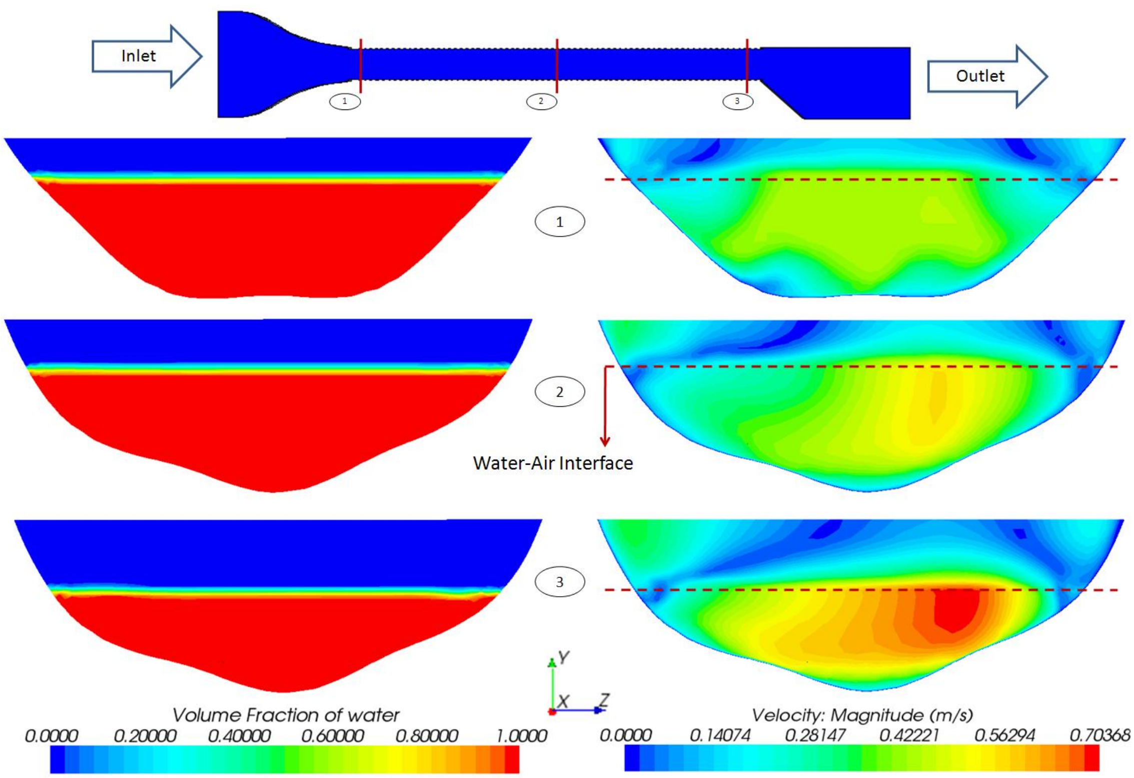

Figure 6. Volume fraction of water and velocity contour plots shown on three different cross sectional planes

Several average flow parameters and the Froude number were calculated at different sections of the flume as shown in Figure 7. At the exit of the barrel region the Froude Number is within less than a percent of critical and it is supercritical at the end of the angled section of the diffuser and exit of the flume. Water enters as a tranquil flow with a Froude number of 0.0526 and leaves as a supercritical flow with a Froude number of 2.4.

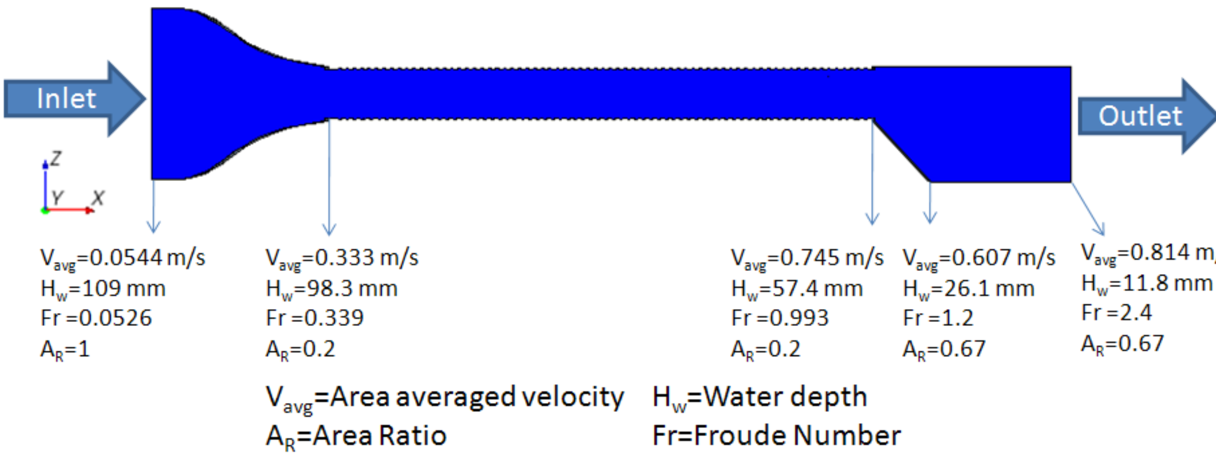

Figure 7. Flow parameters calculated at different cross sections

A velocity streamline plot is shown in Figure 8. This plot also shows the effects of the asymmetric diffuser that extend back into the barrel.

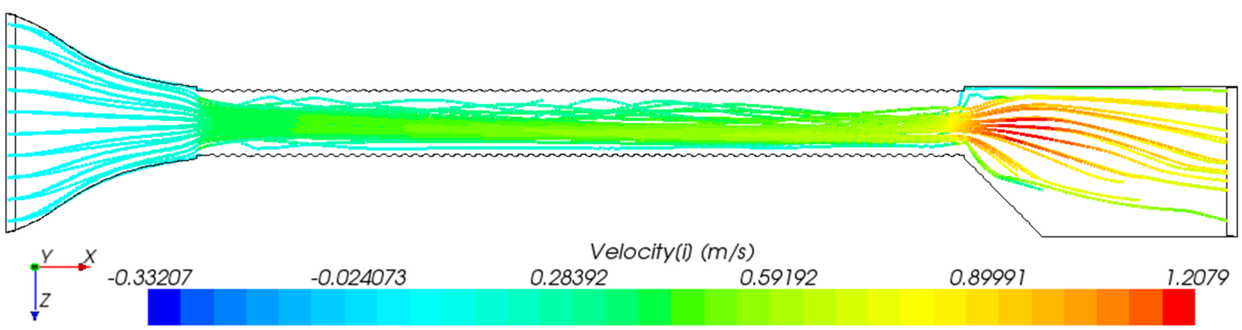

Figure 8. Streamline velocity plot of water in x-direction shown on a section plane in the top view

The water depth calculated based on the position of a VOF=0.5 iso-line down the barrel center plane compared to experimental data is plotted in Figure 9. The computed and experimental depths are within about 5 mm of each other, except for the last experimental point in the diffuser. Most of the difference is a consequence of an immediate small drop at the inlet that is probably a consequence of downstream control where the flow transitions to supercritical, and those conditions are not easy to change by adjusting the exit boundary condition. The slope depth curve of both the computed and experimental data over the zone of the barrel, however, appears to be very close.

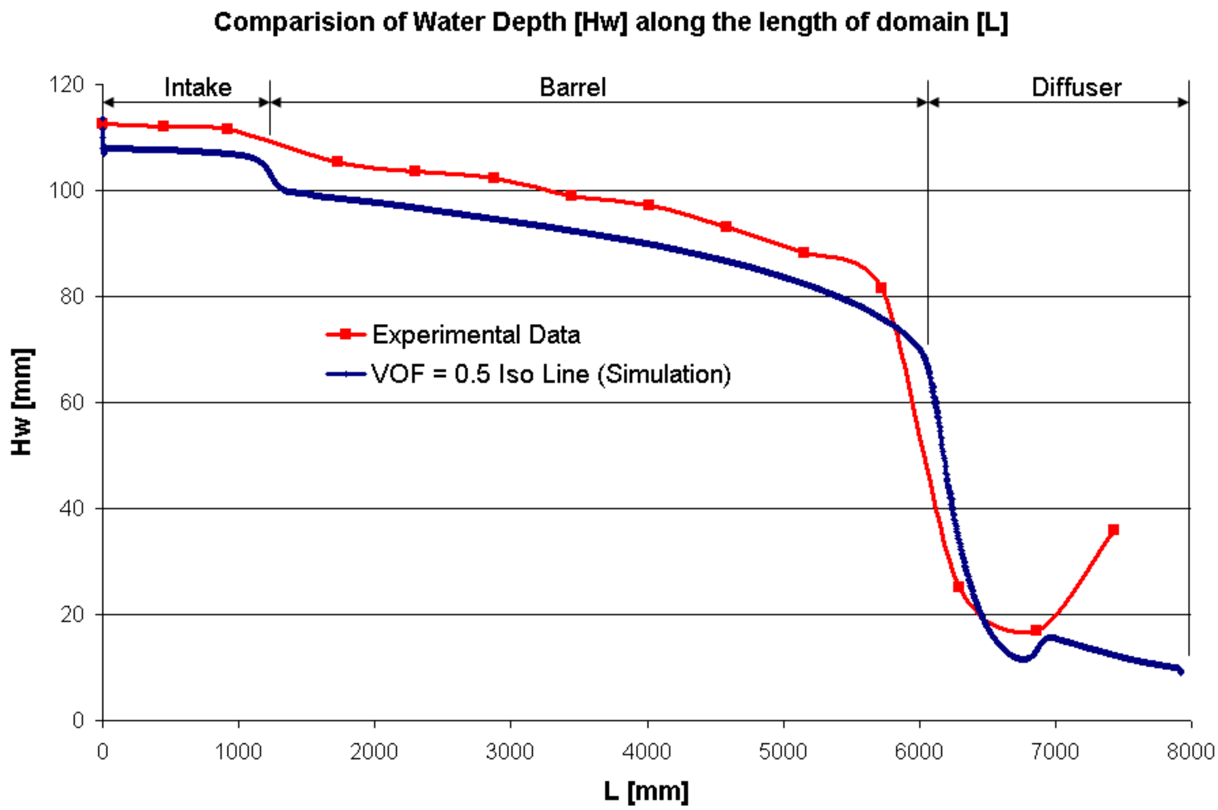

Figure 9. Comparison of simulation results for water level depth with the experimental data.

Case 2: discharge of 4.65 L/s and angle of 11.006 degrees for the flap gate at the exit

The mesh plotted on a length wise barrel center plain cut and a cross section in the barrel is shown in Figure 10 below. The highly refined zones in the grid are positioned where the water-air interface is expected in order to accurately resolve the interface.

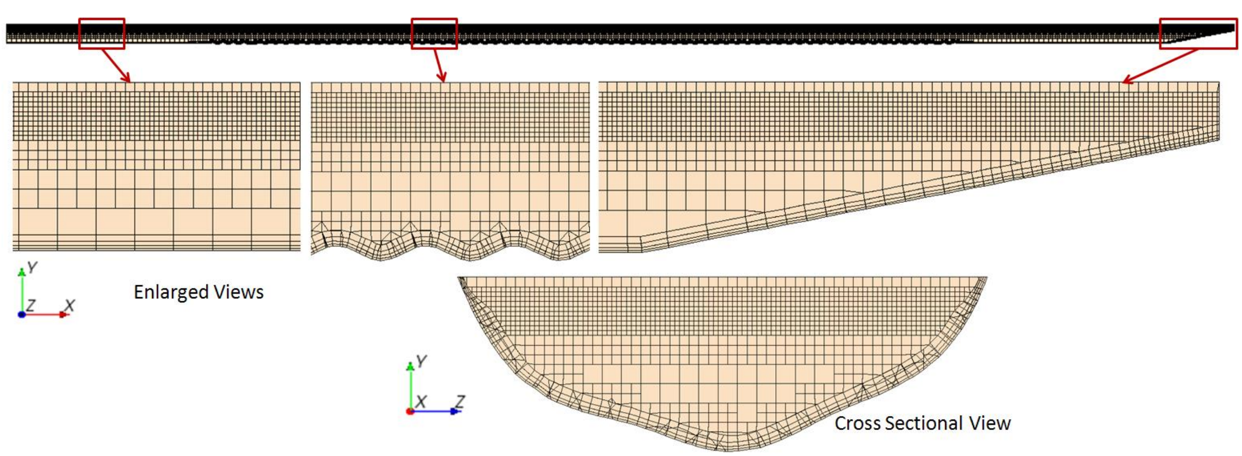

Figure 10. Mesh scene shown on a section plane passing through center plane

A plot of the volume fraction of water shown on the center section plane is presented in Figure 11.

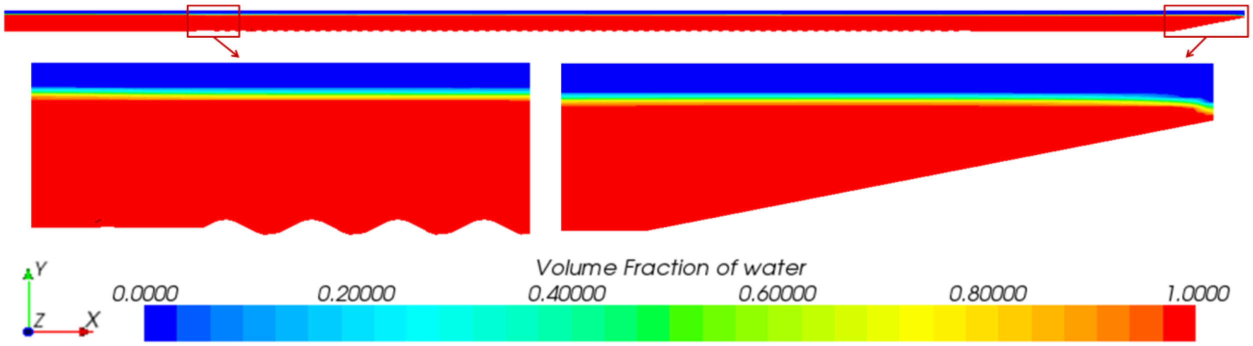

Figure 11. Volume fraction of water shown on a section plane

A velocity contour plot on the cut plane down the center of the barrel is shown in Figure 12. Flow accelerates through the converging inlet into the barrel and decelerates where it spreads out in the diffuser, flows up the flap gate, and then accelerates again as it approaches the exit fall off at the end of the flap gate. The flow separates and does not follow the asymmetric widening of the channel along the right diffuser wall. This behavior can be seen in the streamline plot in Figure 13. Streamlines do not extend over the width of the channel until the flow is passing over the flap gate near the exit plane.

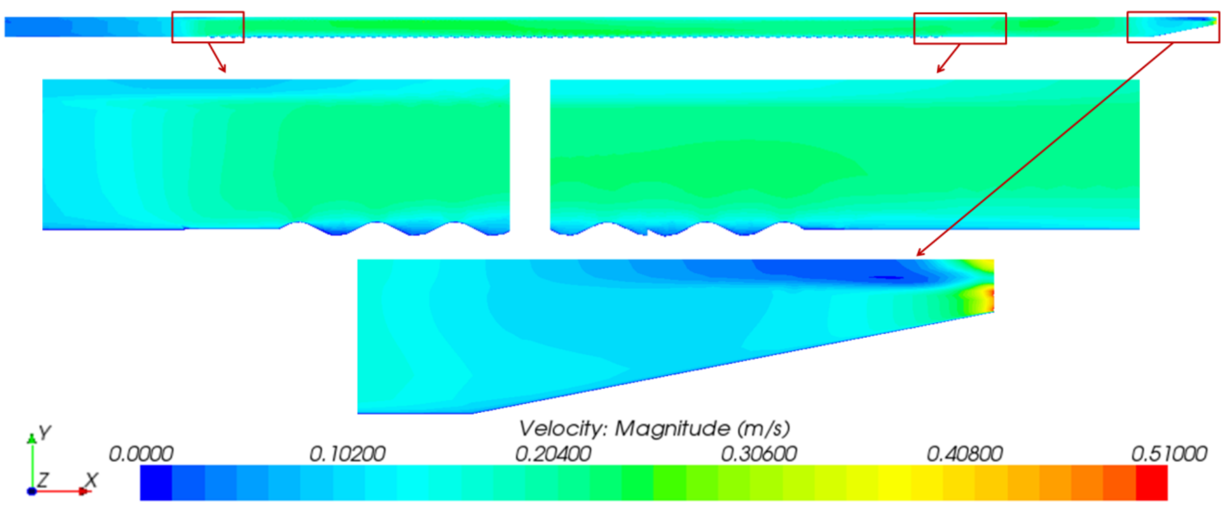

Figure 12. Velocity contour plot shown on a section plane

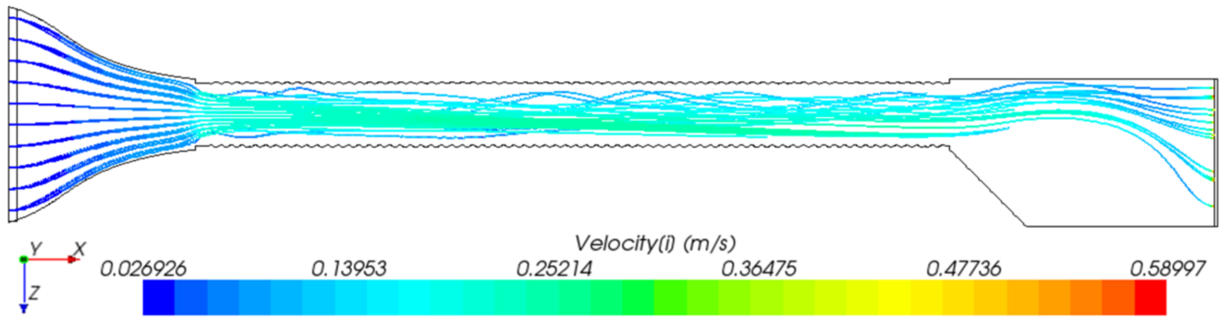

Figure 13. Streamline velocity plot of water in x-direction shown on a section plane in the top view

Volume fraction of water and velocity contours are plotted on different cross sectional planes as shown in Figure 14. Again in this case, the asymmetric diffuser and exit channel with respect to the barrel center line appear to cause the high velocity zone created as flow develops in the barrel to shift toward the right as shown in Figure 14.

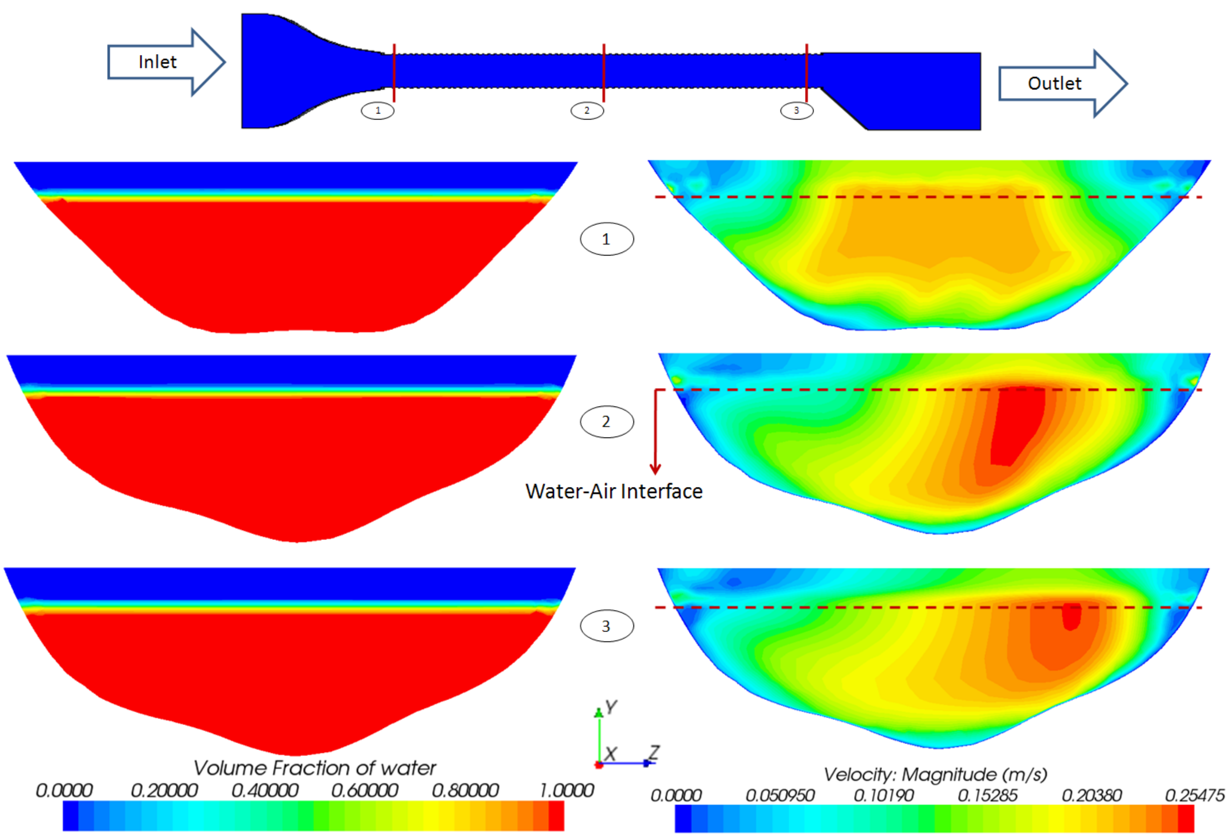

Figure 14. Volume fraction of water and velocity contour plots shown on three different cross sectional planes

The water depth calculated based on the position of a VOF=0.5 iso-line down the barrel center plane compared to experimental data is plotted in Figure 15. The computed and experimental depths are within about 1 to 2 mm of each other over the barrel and diffuser section. This difference is likely within the range of both experimental and computational uncertainty. The slope of the depth curve of the simulation in the barrel, which is a consequence of losses in the barrel, is very close to that of the experimental data.

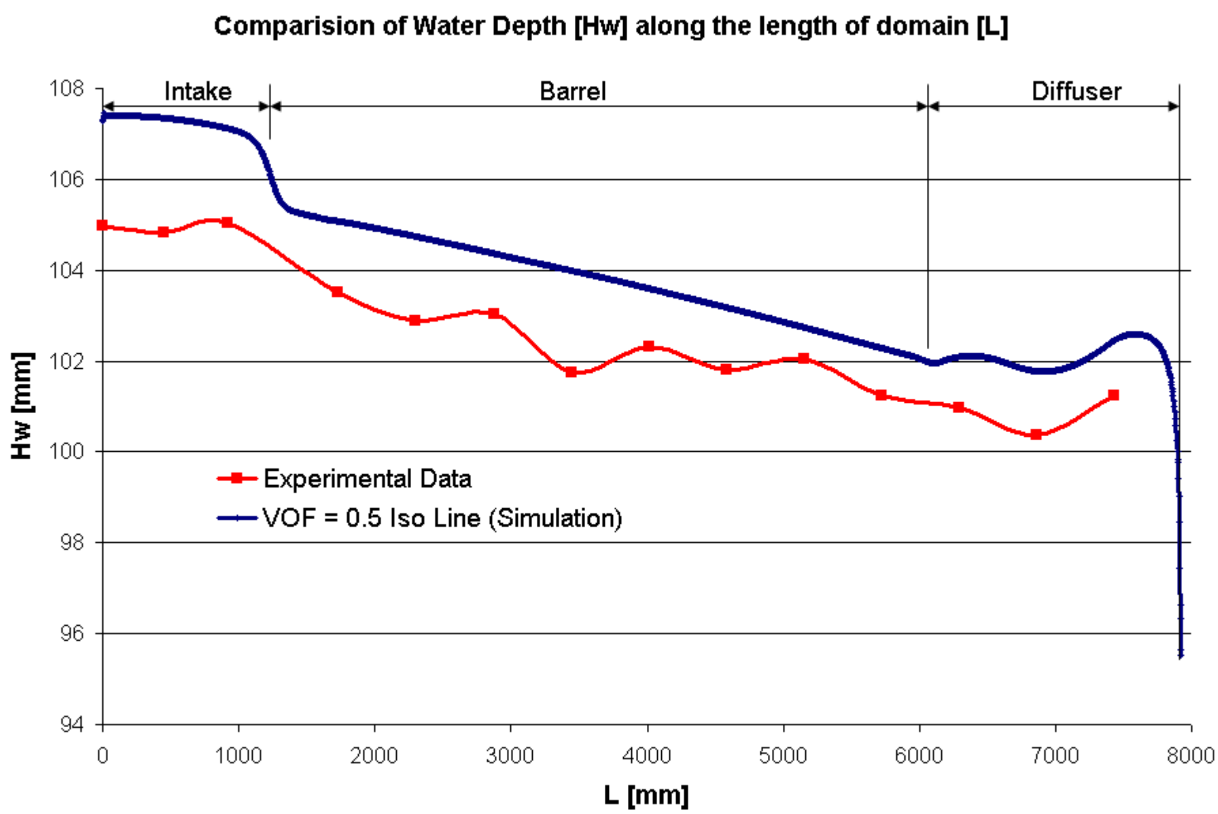

Figure 15. Comparison of simulation results for water level depth with the experimental data.

Current work involves development of a methodology to adjust angle of tilt of the flume by adjusting the orientation of the gravity vector until the free surface of the water in the barrel section is parallel to the bottom of the flume for given flow conditions. This angle corresponds to the losses in the flume through the barrel section. It gives the slope needed to provide enough energy to the flow to overcome the resistance inside the barrel region.