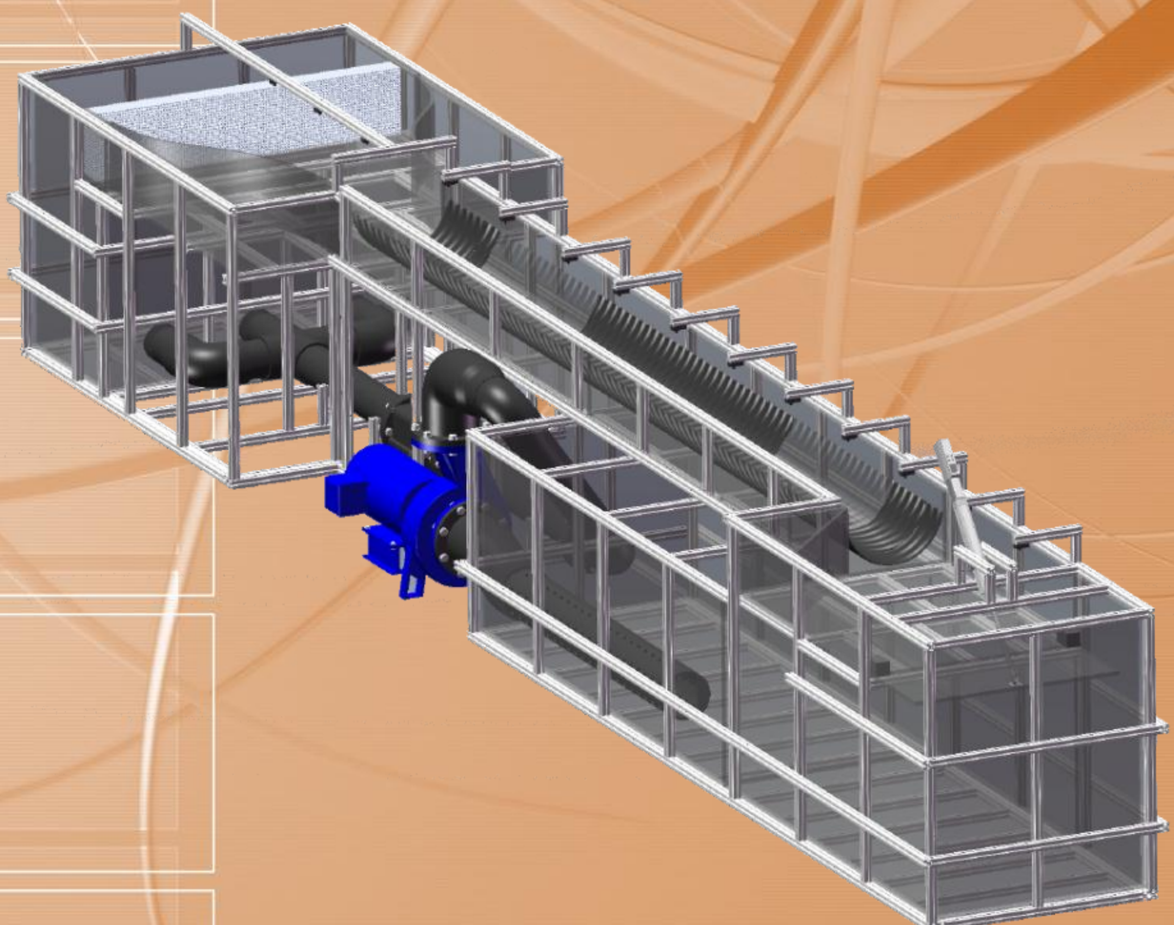
Argonne NATIONAL LABORATORY

# Computational Mechanics Research and Support for Aerodynamics and Hydraulics at **TFHRC**



## Culvert Analysis Quarterly Report

July through September 2011

**Computational Fluid Dynamics Modeling of Flow through Culverts**
**2011 Quarter 4 Progress Report**

**Transportation Research and Analysis Computing Center (TRACC)**
**Energy Systems Division (ES)**
**Argonne National Laboratory (ANL)**

**Principal Investigator:**
**Steven A. Lottes, Ph.D.**

**Contributing CFD Investigators:**

**Sudhir Lanka Venkata**
**Pradip Majumdar, Ph.D.**
**Northern Illinois University**

**Yuan Zhai**
**University of Nebraska**

**Submitted to:**
**Federal Highway Administration**

**Kornel Kerenyi, Ph.D.**
**Turner-Fairbank Highway Research Center**
**Federal Highway Administration**
**6300 Georgetown Pike**
**McLean, VA 22101**

**October, 2011**

# Table of Contents


1. Introduction and Objectives ..... 7
2. Computational Modeling and Analysis of Flow through Large Culverts for Fish Passage ..... 7
   2.1. Velocity Curves for Culverts ..... 9
      2.1.1. Local Depth-Averaged Velocity Curve Development ..... 9
      2.1.2. Average Velocity Curve Development ..... 11
   2.2. Porous Media Modeling ..... 14
      2.2.1. Setting Porous Region Values ..... 14
      2.2.2. Isotropic Resistance Using the Ergun Equation (Forchheimer for Packed Beds) ..... 15
      2.2.3. Implementation of Porous Media in STAR-CCM+ ..... 16
      2.2.4. Description of the Physical Model ..... 17
      2.2.5. Boundary Conditions for the Porous and the Fluid Regions ..... 17
      2.2.6. Meshing Methodology ..... 17
      2.2.7. Comparison of the Reduced Section with an Increased Section for a Reduced Culvert Section ..... 19
      2.2.8. Results and Discussion ..... 20
3. References ..... 26

# List of Figures


Figure 2.1: Variation of flow velocity and depth in a cross-section of a corrugated metal pipe ................ 9

Figure 2.2: Depth-averaged velocity curve development in a cross-section of 6 inch water depth with symmetry boundary ........................................................................................................................ 10

Figure 2.3: Flow path for the selected fish design criteria of velocity and depth .................................... 11

Figure 2.4: Cumultive area from the wall to the centerline .................................................................... 12

Figure 2.5: Cumulative discharge from the wall to the centerline .......................................................... 12

Figure 2.6: Cumulative discharge from the wall to the centerline in a cross-section of 6 inch water depth with symmetry boundary ................................................................................................................ 13

Figure 2.7: Cumulative area from the wall to the centerline in a cross-section of 6 inch water depth with symmetry boundary ........................................................................................................................ 13

Figure 2.8: average velocity curve in a cross-section of 6 inch water depth with symmetry boundary ... 14

Figure 2.9 CAD model representing the computational domain used for porous media modeling .......... 16

Figure 2.10 Volumetric controls created around regions of major interest for mesh refinement ........... 18

Figure 2.11 Cross sectional view of the mesh scenes ................................................................................. 19

Figure 2.12 Increased section of the culvert (nearly three times bigger than the reduced section) used for porous media modeling ........................................................................................................................ 20

Figure 2.13 Cross sectional view of mesh scene along on a plane taken along the length of the increased section ........................................................................................................................................ 20

Figure 2.14 Image depicting the line probes created at a trough and a crest for a reduced section ........ 21

Figure 2.15 Image depicting the line probes created at a trough and a crest for an increased section .... 21

Figure 2.16 Velocity profiles for both reduced and increased sections at a crest ..................................... 22

Figure 2.17 Velocity profiles for both reduced and increased sections at a trough .................................. 23

Figure 2.18 Cross sectional plane created at a crest in the porous region for a reduced section ............. 24

Figure 2.19 Representation of odd numbered Uniform strips of 1 cm width created along the fluid section ........................................................................................................................................ 24

Figure 2.20 Surface-averaged velocity variation along the uniform strips plotted using “ Threholds” ..... 25

Figure 2.21: Velocity distribution over cross section at a crest showing the variation above the porous media gravel bed........................................................................................................................ 26

# List of Tables

Table 2.1 Boundary conditions .......................................................................................................... 17

# 1. Introduction and Objectives

This 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 the Turner-Fairbank Highway Research Center (TFHRC) for a period of five years, beginning in October 2010. The analysis methods employ well-benchmarked and supported commercial computational mechanics software and also include user subroutines, functions, and external software programs and scripts written to automate the analysis procedures. 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.

This quarterly report documents technical progress on the CFD modeling and analysis of flow through culverts for the period of July through September 2011. The focus of effort for the work this year is on improving methods to assess culvert flows for fish passage.

# 2. 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 to investigate methods to model gravel in the culvert. The test matrix in the TFHRC work plan [6] includes tests with the bed height at 15% and 30% of the culvert diameter. For these cases, the culvert bed material is coarse gravel with a mean diameter, $D_{50}$ = 12 mm. At this gravel size, the gravel bed boundary cannot be treated as a rough wall using wall functions because the centroid of the near wall computational cell must be at a position that is greater than the roughness height. For 12 mm gravel, the near wall mesh would be far too large for the analysis results to be mesh independent. Two options to model flow parallel to a porous gravel bed are (1) to treat the bed as a porous media, with a flat interface dividing the two flow zones, and (2) to mesh out the rough bed contour created by the top layer of gravel.

determine the local velocities and flow distributions through culverts for the fish passage with no gravel in the culvert. In order to more accurately evaluate the ability of fish to traverse culverts, it is desirable to look at the changes in the local average velocity of the flow adjacent to the culvert wall under low flow conditions. CFD runs using the cyclic boundary conditions to obtain the fully developed flow on a reduced 3D section of the culvert (symmetric quarter of the culvert section with corrugations from trough to another trough) have been conducted using CD-adapco's STAR-CCM+ software. Use of the cyclic boundary condition requires an assumption of a nearly flat water surface that can be modeled with a symmetric boundary condition that allows a free slip water velocity at that boundary. The cyclic boundary approach shortens the simulation time required to establish a fully developed flow with a known mass flow rate (with this approach several test cases can be completed per day). The periodic fully developed condition is achieved by creating a cyclic boundary condition, where all outlet variables are mapped back to the inlet interface, except for the pressure because there is a pressure drop corresponding to the energy losses in the culvert section. The pressure jump needed to balance the pressure drop for the specified mass flow is iteratively computed by the CFD solver. The runs were conducted with various mesh sizes, to take a closer look at how the velocity distribution and other flow parameters vary at different locations of the flow field by varying the base size of the mesh to obtain solutions that are effectively mesh independent. The mesh refinement study is also used to identify meshes that are computationally efficient while yielding good mesh independent results. An investigation of how the flow field varies for different cases such as reduced culvert section when considered from a trough to trough versus crest to crest. The computational model is based on the three-dimensional transient RANS k-epsilon turbulence model with wall function treatment.

The modeling work was done in collaboration with staff at TFHRC conducting physical experiments of culvert flows for the fish passage project in the Federal Highway Administration (FHWA). A preliminary comparison of the velocity distribution on the trough section between CFD model results and laboratory observation data was conducted. The 3D CFD model solves the Reynolds averaged Navier-Stokes (RANS) equations with k-epsilon turbulence model with wall function. The VOF method, which captures the free surface profile through use of the variable known as the volume of fluid was used in the multi-phase CFD model. The verification of the CFD model for engineering application using the laboratory observation data is a key step for the further work. A comparison of the multi-phase model and full scale flume single phase model was also done.

## 2.1. Velocity Curves for Culverts

### 2.1.1. Local Depth-Averaged Velocity Curve Development

Flow velocity and flow depth are two important factors influencing fish activity in a culvert. Figure 2.1 illustrates the information that was measured in the flume and was calculated by the numerical modeling methods described below. At a given culvert cross-section, flow depth, and flow discharge, the local depth-averaged velocities $V_1$, $V_2$, $V_3$, etc. are measured at regular offsets from the culvert wall. Typically the local depth-averaged velocity will approach zero at the culvert wall and will be at a maximum near the center of the culvert.

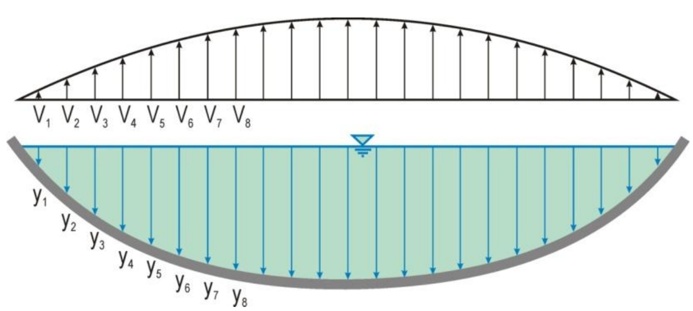



Figure 2.1: Variation of flow velocity and depth in a cross-section of a corrugated metal pipe

In the numerical modeling, the culvert cross-section is divided into evenly spaced strips, and then the grid's discharge and area in each strip are determined by integrating over the strip. The ratio of the integrated discharge and integrated area is the depth-averaged velocity. In this study, a trimmed cell mesh was used to generate an extremely high quality hexahedral-based mesh for the culvert geometry. This kind of mesh model gives a structured mesh across the culvert section that is well suited for computing strip averages because the grid can be built to align cell faces with strip boundaries. When these boundaries are not aligned with cell faces, velocity data in uniform vertical strips in the cross-section are generated by interpolating the original grid velocity data from STAR-CCM+. The exported data is then used to get the depth averaged velocity distribution over the strips by averaging the velocities falling in each strip with MATLAB. The results are shown in Figure 2.2.

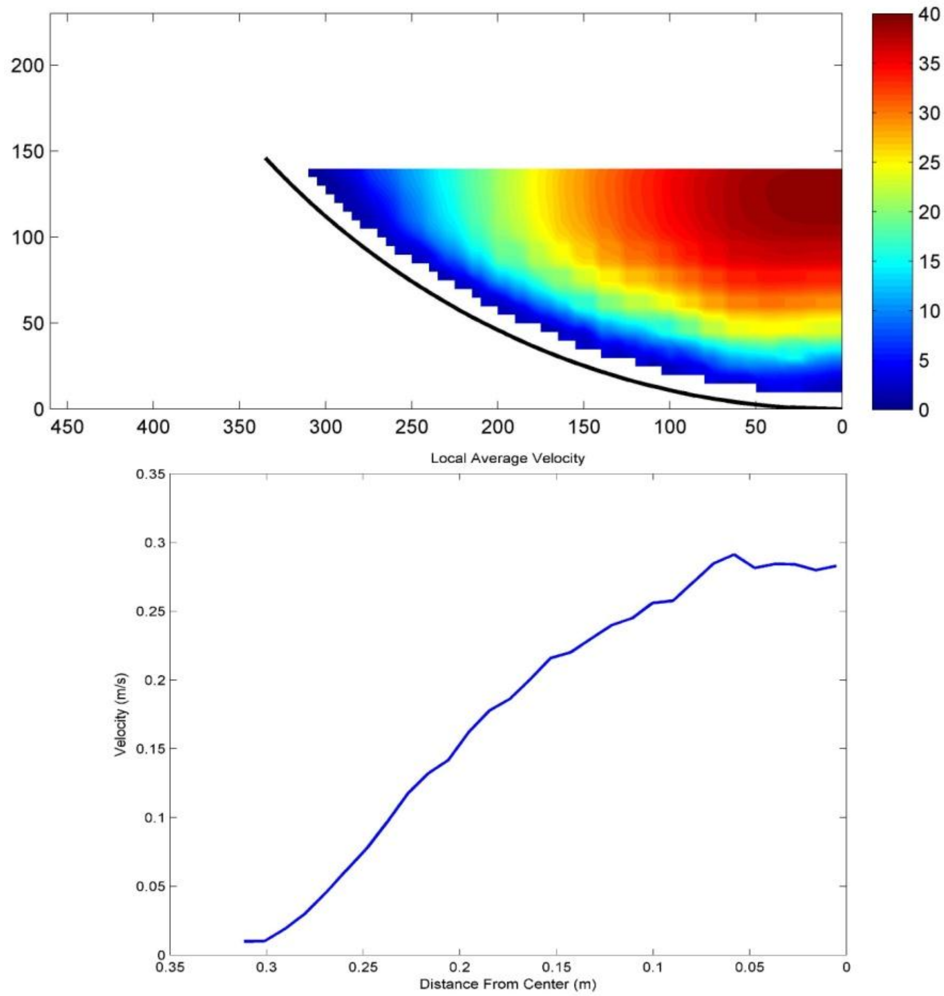



**Figure 2.2: Depth-averaged velocity curve development in a cross-section of 6 inch water depth with symmetry boundary**

In a similar manner, the local flow depths y1, y2, y3 can be measured at the same offsets from the wall that are used for the velocity readings. These depths can be related to the maximum flow depth $y_{max}$

through the culvert geometry. The relationship between the local depth-averaged velocity V and the local depth y of the flow at any point in the culvert cross-section is the important information needed for the Aquatic Organism Passage design of the culvert. The fish design criteria, can be provided by the appropriate environmental agency as noted in the discussion of Tables 1A and B in the work plan [6]. The next step is to predict velocity distribution and depth variations through culverts under various conditions, including flow depths, inlet velocities, bed elevations, and roughness. A path through a culvert suitable for fish passage can be defined for the given conditions from strip depth average velocities. The fish path in the culvert is shown in Figure 2.3.

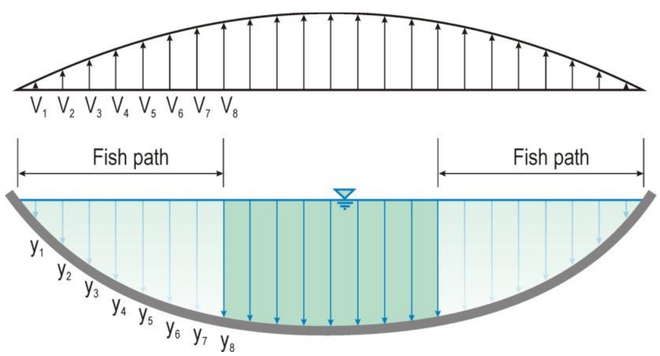



Figure 2.3: Flow path for the selected fish design criteria of velocity and depth

In Figure 2.3, one end of the path is defined by the culvert wall. The other end of the path, towards the center of the culvert, is defined by the point where the local depth-averaged flow velocity V is equal to the maximum fish swimming velocity, $V_F$ , as defined by the appropriate standard. In this illustration example $V_8 = V_F$. Note that another limit to the fish path can be the water depth when the depth in a strip is less than that required for larger species of fish, such as trout or salmon.

### 2.1.2. Average Velocity Curve Development

The area (A) and volume flow rate (Q) of the flow within the limits of the fish path also can be computed, and the average velocity of the flow within the fish path can be computed as illustrated below in Figure 2.4 and Figure 2.5.

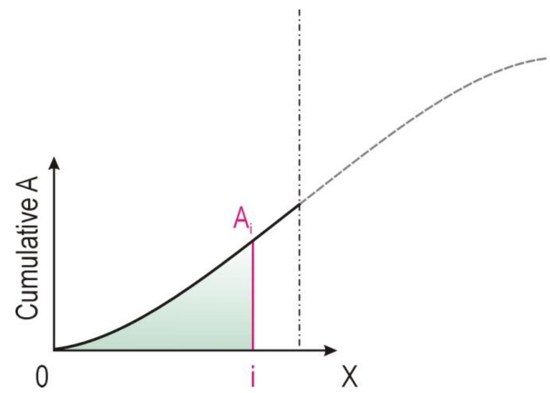



Figure 2.4: Cumultive area from the wall to the centerline

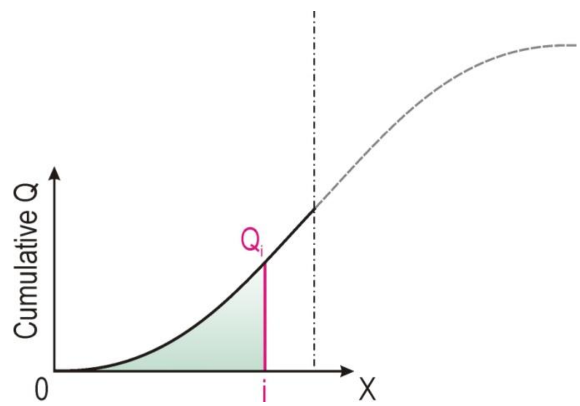



Figure 2.5: Cumulative discharge from the wall to the centerline

Equation 2.1 can be used to compute the average velocity between the outer culvert wall and position i:

$$(V_{AVG})_i = \frac{Q_i}{A_i} \qquad 2.1$$

For the flow condition illustrated in Figure 2.2, the cumulative discharge (Q), cumulative area (A), and the corresponding average velocity can be computed with MATLAB, as shown in Figure 2.6 through Figure 2.8.

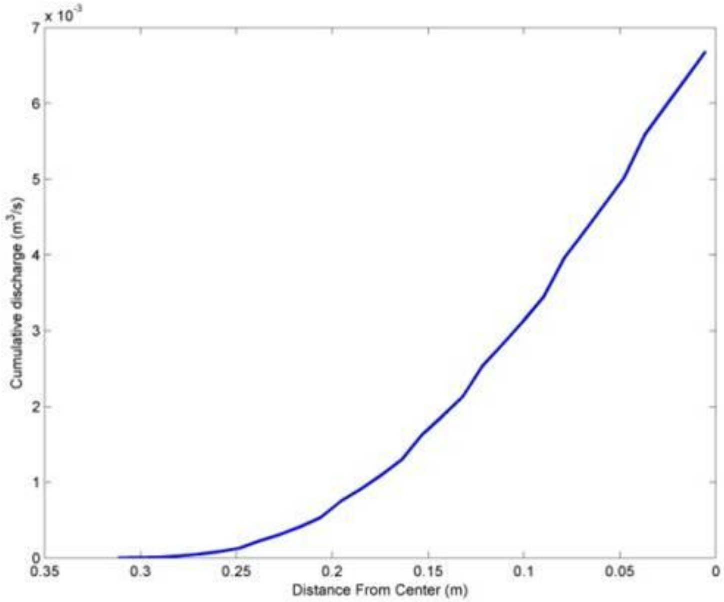



**Figure 2.6: Cumulative discharge from the wall to the centerline in a cross-section of 6 inch water depth with symmetry boundary**

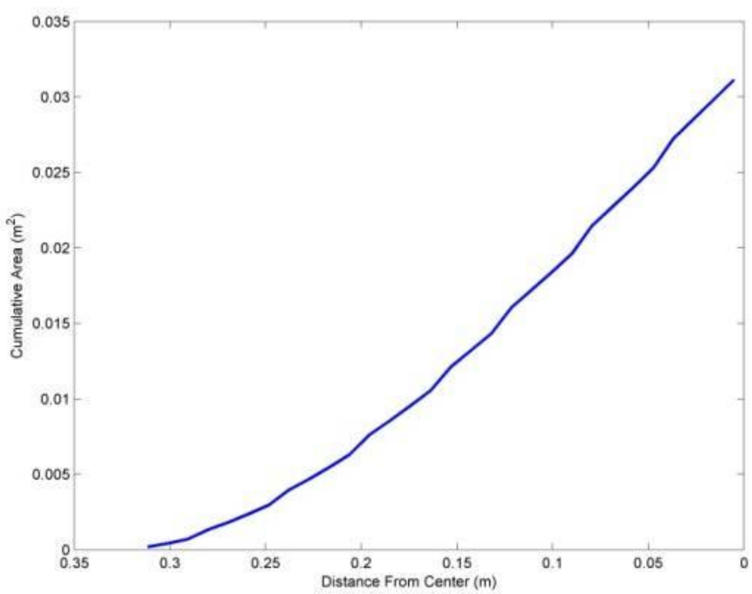



**Figure 2.7: Cumulative area from the wall to the centerline in a cross-section of 6 inch water depth with symmetry boundary**

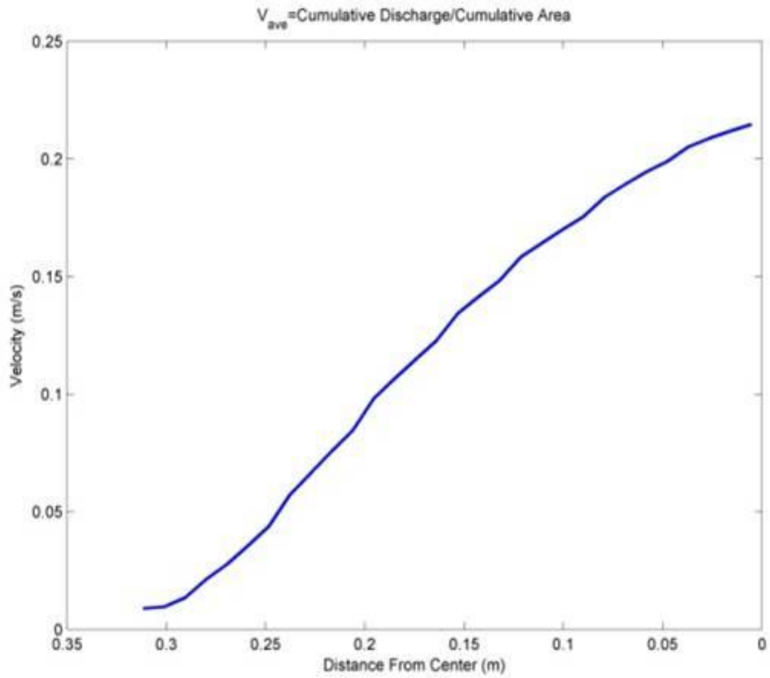



Figure 2.8: average velocity curve in a cross-section of 6 inch water depth with symmetry boundary

## 2.2. Porous Media Modeling

A porous medium is a material whose internal structure consists of solid elements permeated by fine-scale voids that permit the passage of fluids. To model flow through a porous media, the STAR-CCM+ User Guide notes that it is not the details of the internal flow in the porous region that is of interest, but rather the macroscopic effect of the porous medium on the overall fluid flow. The effect of the porous medium on the flow is defined using lumped parameters, which are typically taken to be resistance coefficients for a source term in the momentum equation. These parameters are given as input to the software, so as to reflect the effect of the porous media on the flow that borders the porous section. As a gravel bed in a culvert, the porous section should have the effect of a very rough wall with a small slip condition at the interface that is determined by the flow resistance of the porous media. Various CFD tests have been conducted to determine if a porous media model appears to be a viable way to model large diameter gravel in the bed of a culvert. The results obtained have been presented in the following sections indicating that the porous media does have the expected effects as a flow resistance similar to a rough wall when it is parallel to a primary unobstructed flow.

### 2.2.1. Setting Porous Region Values

In simulating the porous medium, it is important to understand the parameters that determine the flow resistance in the porous region. The first and the foremost parameter defined in setting up a porous region is porosity ($\chi$). Porosity is defined as the ratio of the open volume to the total volume of the porous medium. This is used to modify the fluid density ($\rho$) in appropriate terms of the continuity, energy and species equations by the product $\rho\chi$ ($\chi$ being the porosity of the material), to reflect the fact that only part of the volume is occupied by the fluid.

### 2.2.2. Isotropic Resistance Using the Ergun Equation (Forchheimer for Packed Beds)

The equation describing the flow through a porous medium is often Darcy's law, which relates the flow velocity to the pressure gradient based on a measure of permeability. Permeability is a measure of the ability of a porous material (or the gravel) to allow fluids to pass through it. This law can be expressed as:

$$-\nabla_P = \frac{\mu}{K_P} v \qquad 2.2$$

Where $\mu$ is the fluid molecular viscosity, $K_P$ is the permeability (considered to be an intrinsic property of the porous medium) and $v$ is the superficial velocity through the medium.

As the flow velocity increases, the relationship between velocity and pressure gradient becomes nonlinear. Dupuit and Forchheimer (as reported by [7]) proposed the addition of a quadratic term as follows:

$$\nabla_\mathrm{p} = \frac{\mu}{\mathrm{k_p}} v + \beta\rho v^2 \qquad 2.3$$

Equation 2.3 is commonly known as the Forchheimer equation [26]. The beta factor must be deduced depending on the particular flow and medium of interest, and is commonly determined by experiment.

An empirical form of the Forchheimer equation is derived for a pressure driven flow through a packed bed and given as the Ergun equation (as discussed in [7]). This is an empirical model for the pressure drop, over a length, L, of fluid flowing through a packed bed of spheres and expressed as follows:

$$-\frac{dp}{L} = \frac{150\mu(1-\chi^2)v}{\chi^3 {D_p}^2} + \frac{1.75\rho(1-\chi)v^2}{\chi^3 D_p} \qquad 2.4$$

Where $\rho$ is the fluid density and $D_p$ is the mean diameter of particles in the porous medium. In the current CFD investigation, $D_p$ is taken as 12mm for a culvert whose diameter is 36 inches and corrugation size 3 inch by 1 inch. The mean diameter of the particle corresponds to the manning's value of n = 0.023 for a randomly packed bed of spheres [6]. On comparing the above equations, the expression for permeability and the beta factors can be determined as:

$$\frac{1}{k_p} = \frac{150\mu(1-\chi^2)v}{\chi^3 {D_p}^2} \quad 2.5$$

And

$$\beta = \frac{1.75\rho(1-\chi)v^2}{\chi^3 D_p} \quad 2.6$$

The final viscous and inertial terms used in STAR-CCM+ are obtained as:

$$P_v = \frac{150\mu(1-\chi^2)v}{\chi^3 {D_p}^2} \quad 2.7$$

And

$$P_i = \frac{1.75\rho(1-\chi)v^2}{\chi^3 D_p} \quad 2.8$$

### 2.2.3. Implementation of Porous Media in STAR-CCM+

CD-adapco's STAR-CCM+ software provides a mechanism by which field functions can be used to modify the physics models of the flow or define parameters in physics models. The specification of porous inertial resistance and porous viscous resistance along with the other input conditions and values are enabled for a porous region to specify the coefficients of flow resistance. These coefficients are used to calculate the porous media sink terms in the momentum equation, as detailed in the formulation.

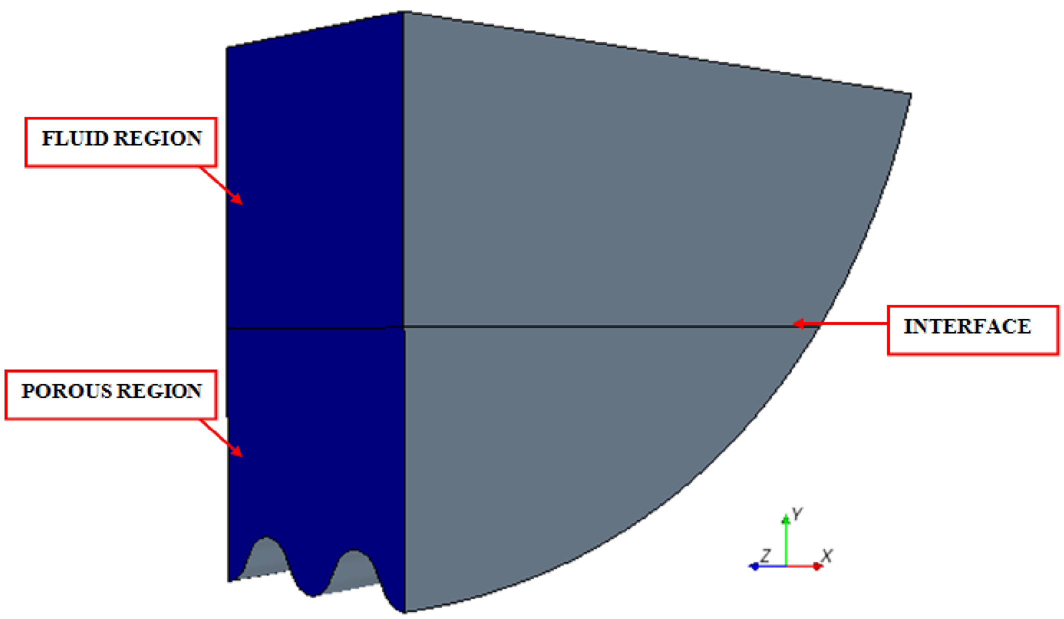



Figure 2.9 CAD model representing the computational domain used for porous media modeling

### 2.2.4. Description of the Physical Model

In this study, a simulation model was developed using the commercial CFD software STAR-CCM+. A small section of the culvert barrel was modeled with cyclic boundaries at the inlet and outlet sections of the computational domain in the fluid region as in the previous studies for flow through a culvert with no gravel. For simulating the flow within the porous region, the ends of the domain with the cyclic boundaries are treated as walls in the lower porous region because the cyclic boundary conditions cannot be applies when there are multiple types of inlet and outlet boundaries to be coupled. A 36 inch diameter culvert with corrugation size 3 inches by 1 inch has been used for this study. A flow depth of 6 and 9 inches for a flow velocity of 0.71 feet/second is considered for this study.

### 2.2.5. Boundary Conditions for the Porous and the Fluid Regions

The physical model consists of two regions namely the fluid region, which is the focus of the study and the porous region (gravel section) which is being simulated using the porous media approach. The following are the boundary conditions used for the fluid region.

**Table 2.1 Boundary conditions**

| Boundary | Name | Type |
|---|---|---|
| Face at minimum value | Inlet | Cyclic boundary condition |
| Face at maximum x value | Outlet | Cyclic boundary condition |
| Water surface | Top | Symmetry plane |
| Centerline | Center | Symmetry plane |
| all other surfaces | Barrel | No-slip wall |

For the Porous region, the boundary at the centerline is taken to be a symmetry plane and all other surfaces are considered as no-slip wall. In Figure 2.9, the interface between the fluid and the porous region, which is taken to be an in-place boundary is shown.

### 2.2.6. Meshing Methodology

A strategic meshing technique has been adopted to mesh the model to get good CFD results. It is very important to have an optimum number of cells in the mesh to have a good flow resolution. As a part of the meshing strategy, a volumetric control (annulus ring) has been created along the corrugated section. The mesh is refined along the volumetric control with respect to the other regions of the computational domain. The refinement of the mesh is defined by specifying a reduction of mesh size for volume within an annulus intersecting the model as shown in Figure 2.10. The volumetric control body intersecting the corrugated section provides a means to refine the mesh in the corrugated region of interest. The refined mesh enables better resolution of the flow field with recirculation zones at the troughs between the corrugations. Also a block has been created at the interface of the fluid and the porous regions where the mesh is further refined. The purpose of creating a block for mesh refinement is the same as that of creating an annulus ring around the corrugated section. Meshing also includes a prism layer consisting of orthogonal prismatic cells running parallel to the wall boundaries, which constitutes a boundary mesh that is good for the application of wall functions to compute the shear stress at the wall boundaries.

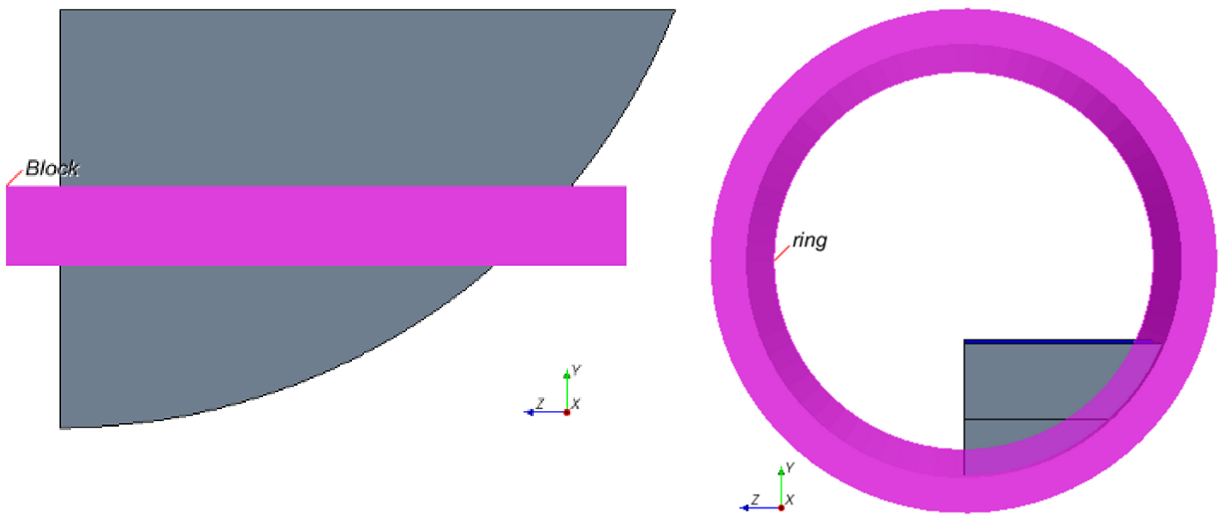



**Figure 2.10 Volumetric controls created around regions of major interest for mesh refinement**

From the mesh refinement studies, done to find the optimum base size of the mesh for a 36 inch diameter of the culvert with corrugation size 3 inches by 1, for a flow depth of 6 inches and 9 inches it was found that a base size of 5 mm and 67 % refinement in the volumetric controls is a good size for the mesh which gives mesh independent simulation results with adequately fast run times. The same base size of the mesh and refinement along the volumetric controls has been used for the present study.

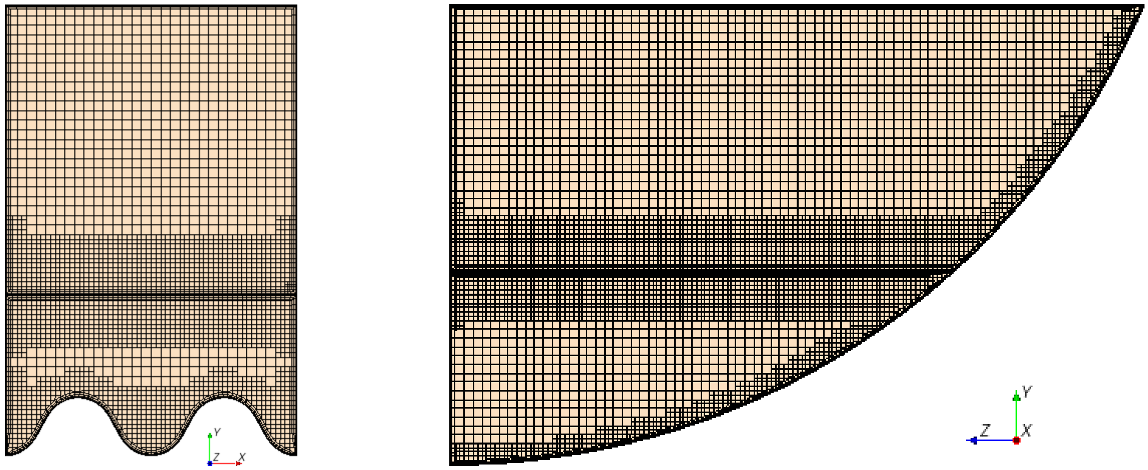



Figure 2.11 Cross sectional view of the mesh scenes

### 2.2.7. Comparison of the Reduced Section with an Increased Section for a Reduced Culvert Section

A reduced section with cyclic boundary conditions has been used for all CFD tests. Working with a reduced section is computationally fast and allows a couple of tests can be completed in a day. In order to check the reliability of the results with blocking walls in the porous section at the cyclic boundaries of the culvert, a comparison study has been conducted to check for significant differences in both the fluid and the porous regions when the modeled section length is significantly increased allowing more opportunity for induced flow to develop in the upper part of the porous media when compared to a reduced section of only two corrugation crests.

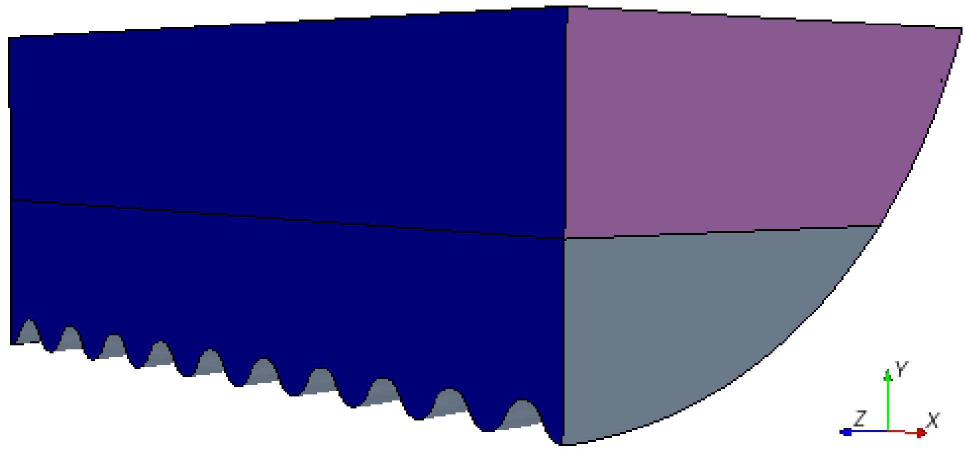



Figure 2.12 Increased section of the culvert (nearly three times bigger than the reduced section) used for porous media modeling

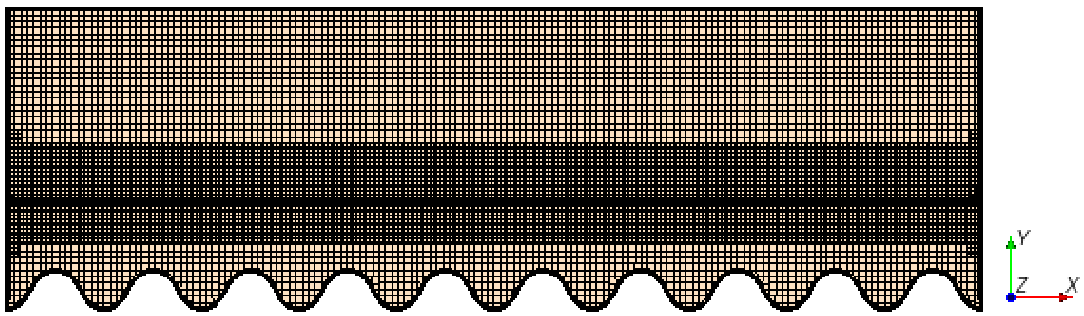



Figure 2.13 Cross sectional view of mesh scene along on a plane taken along the length of the increased section

For this comparison study, an increased section has been created. The increased section is 5 times longer than the reduced section. The CFD run corresponding to the increased section has been set mesh and flow parameters as that of the reduced section. After mesh generation, it was found that the volume mesh for the reduced section contains about 554176 cells and the volume mesh for the increased section contains about 2468562 cells. The results of each of these tests are presented in the following sections.

### 2.2.8. Results and Discussion

Line probes have been created along the flow section in the STAR-CCM+ software. In this particular case line probes have been created at a trough and a crest which are the regions of major interest. Each of the line probes created has 30 points on the line. The value of the velocity magnitude of the flow is extracted at that particular point. Velocity profiles have been plotted using the line probes at a trough and a crest for both the reduced barrel and the increased sections. By taking a close look at the velocity profiles, it is possible to better analyze the nature of the flow.

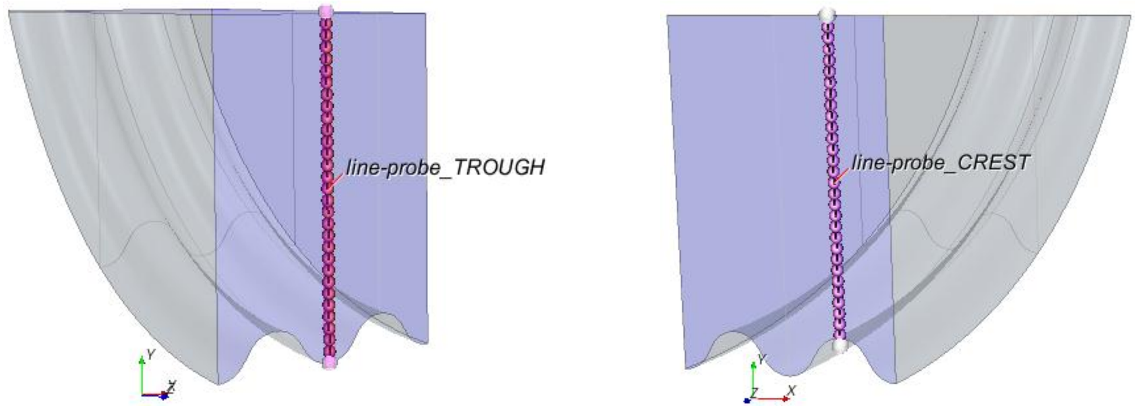



Figure 2.14 Image depicting the line probes created at a trough and a crest for a reduced section

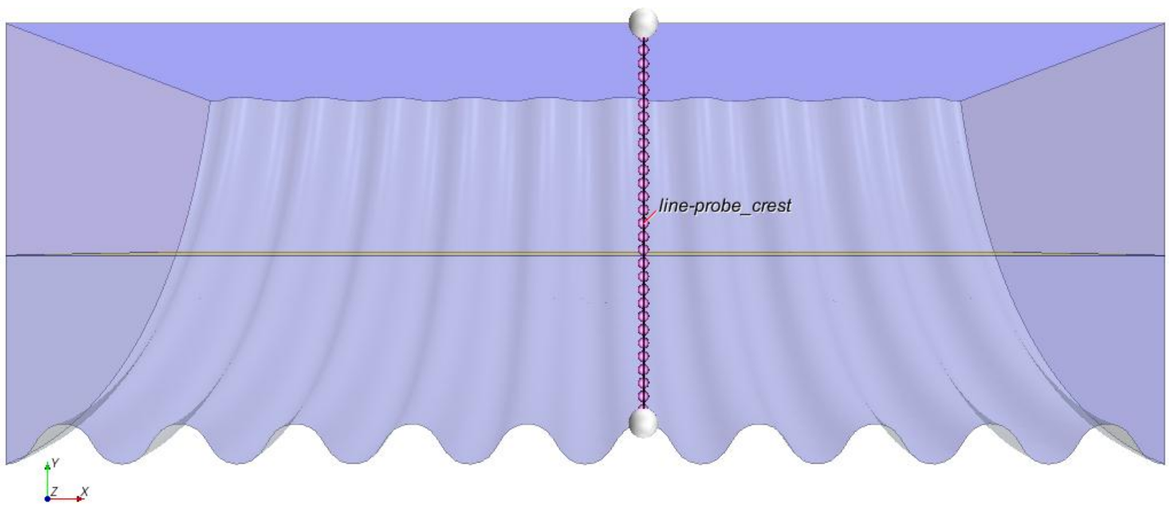



Figure 2.15 Image depicting the line probes created at a trough and a crest for an increased section

In Figure 2.16, the x-axis of the plot represents velocity and the y-axis represents the position of the line probe at a trough in the vertical direction. The minimum unit on the y-axis is 0.15 m and the maximum unit is 0.45 m. The y coordinate of the boundary representing the water surface (namely the top of the reduced culvert section in the CFD study) is at 0.17 m and the y coordinate of the boundary representing the bottom of the culvert at the wall is at 0.4572 m. The same CAD model has been used for all the CFD simulations with the co-ordinates of the reduced symmetric barrel section considered from a trough to a trough as mentioned above. The top surface of the culvert is simulated as a symmetry plane as mentioned previously which represents an imaginary plane of symmetry in the simulation. It implicates an infinitely spread region modeled as if in its entirety. The bottom of the culvert is simulated as a wall with a no slip condition. When velocity is plotted against position, the velocity at the wall is zero, the first point plotted is the velocity in the cell next to the wall and increases with distance from the wall.

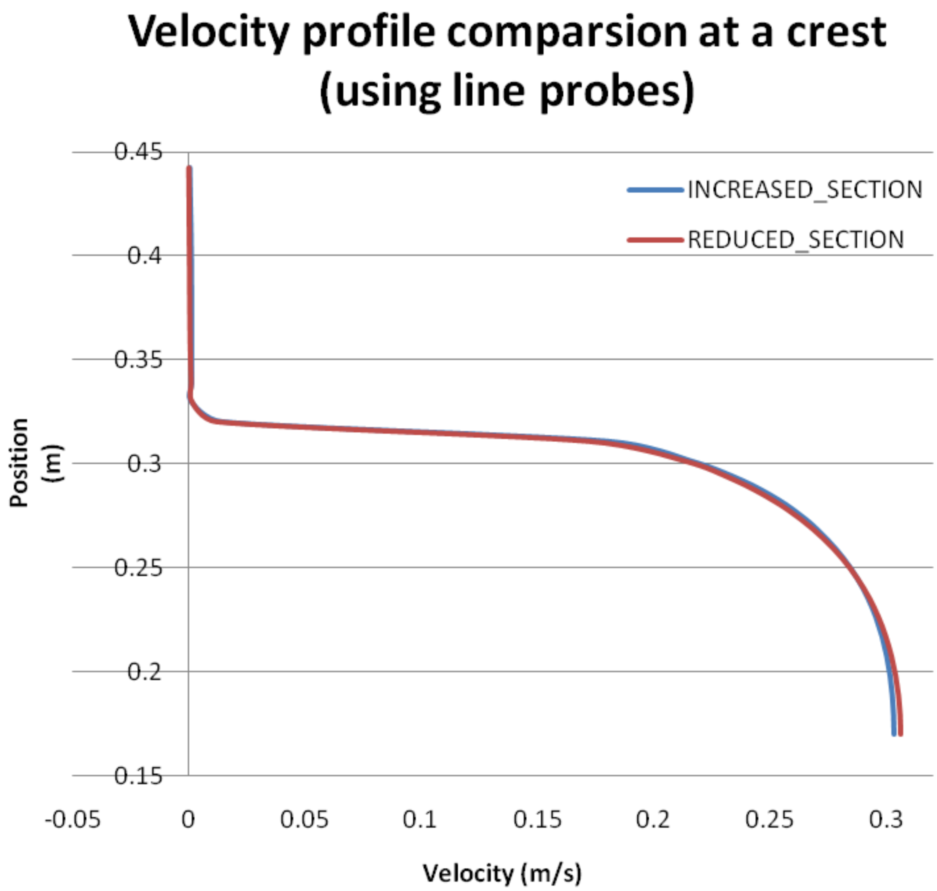



Figure 2.16 Velocity profiles for both reduced and increased sections at a crest

In Figure 2.16, the velocity and the position corresponding to the line probe at a crest are plotted on the x and y axis respectively. Figure 2.17 shows a similar comparison at a trough. The velocity in the porous region is very close to zero. The velocity profile in the fluid region is close to that of one with a wall boundary. The profiles for reduced and long barrel sections are nearly identical, and therefore the shorter section appears to be reasonable to use for determining the cross section velocity distribution above the porous media in parametric tests for different conditions and culvert geometry parameters with a gravel bed.

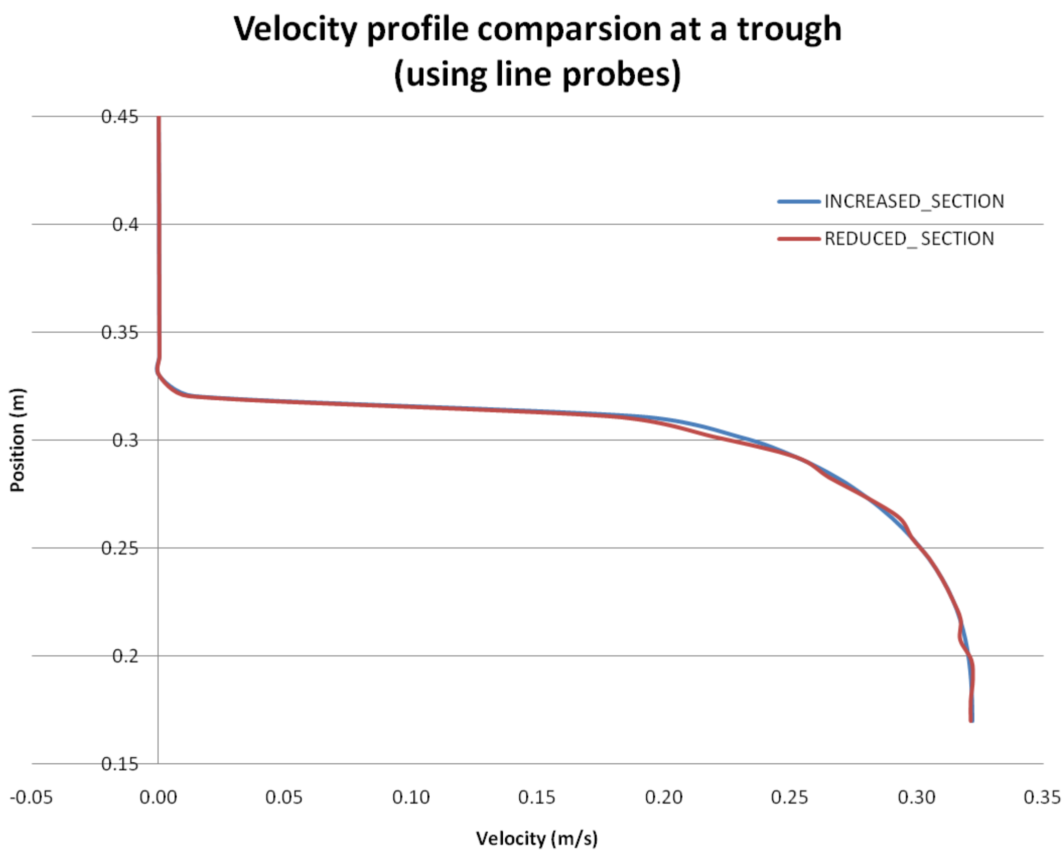



Figure 2.17 Velocity profiles for both reduced and increased sections at a trough

Uniform strips were created on the plane section at a crest in the test case. Figure 2.19 shows the odd numbered strips created on the plane section at a crest in the fluid region. This procedure is carried out by creating multiple "Thresholds" of 1 cm width along the plane section that align with grid cell faces. They are aligned with cell faces to avoid interpolation error and obtain the mean strip averaged velocity based on cell centroid values. After the thresholds are created, there is a "Report" feature available in STAR-CCM+ which calculates the surface averaged velocity over a vertical strip object.

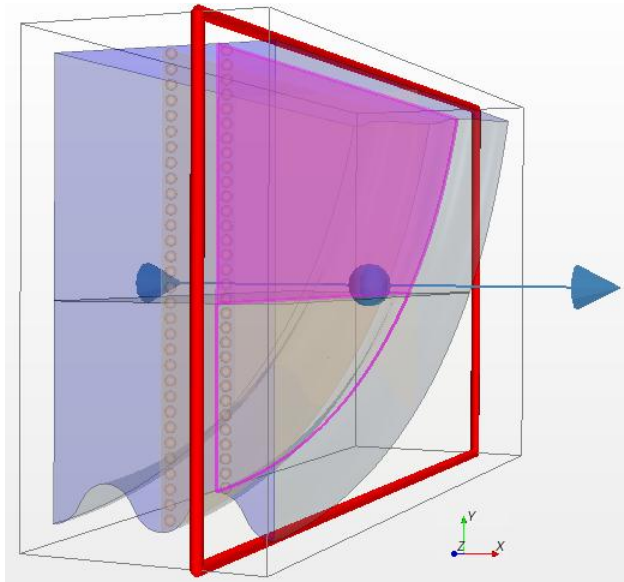



Figure 2.18 Cross sectional plane created at a crest in the porous region for a reduced section

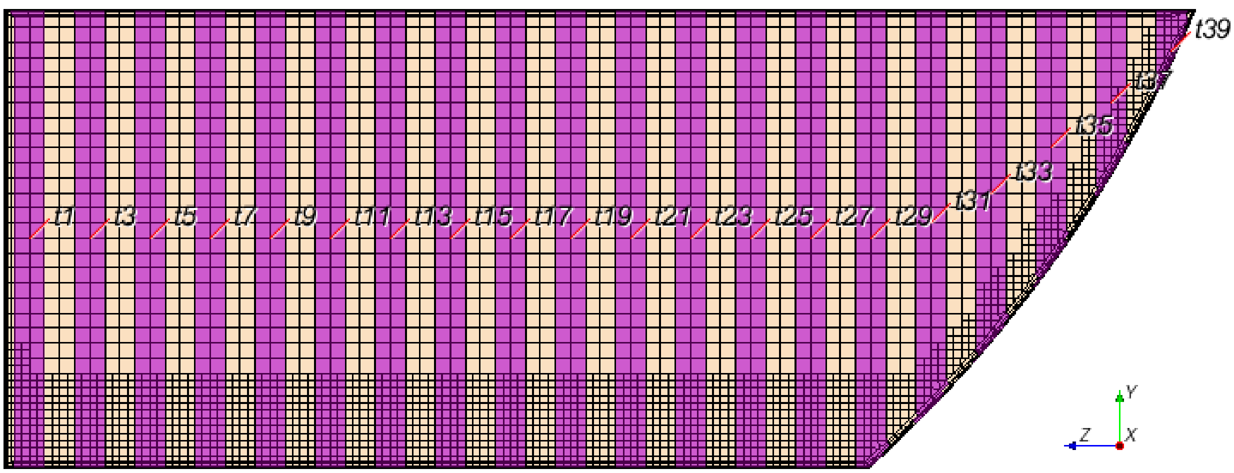



Figure 2.19 Representation of odd numbered Uniform strips of 1 cm width created along the fluid section

In Figure 2.20 a plot of surface-averaged velocities of the strips on the plane section at a crest is plotted across the width of a cross section, where the centerline is at zero. The y axis of the plot indicates the position of a strip and the x axis of the plot indicates the strip-averaged velocity.

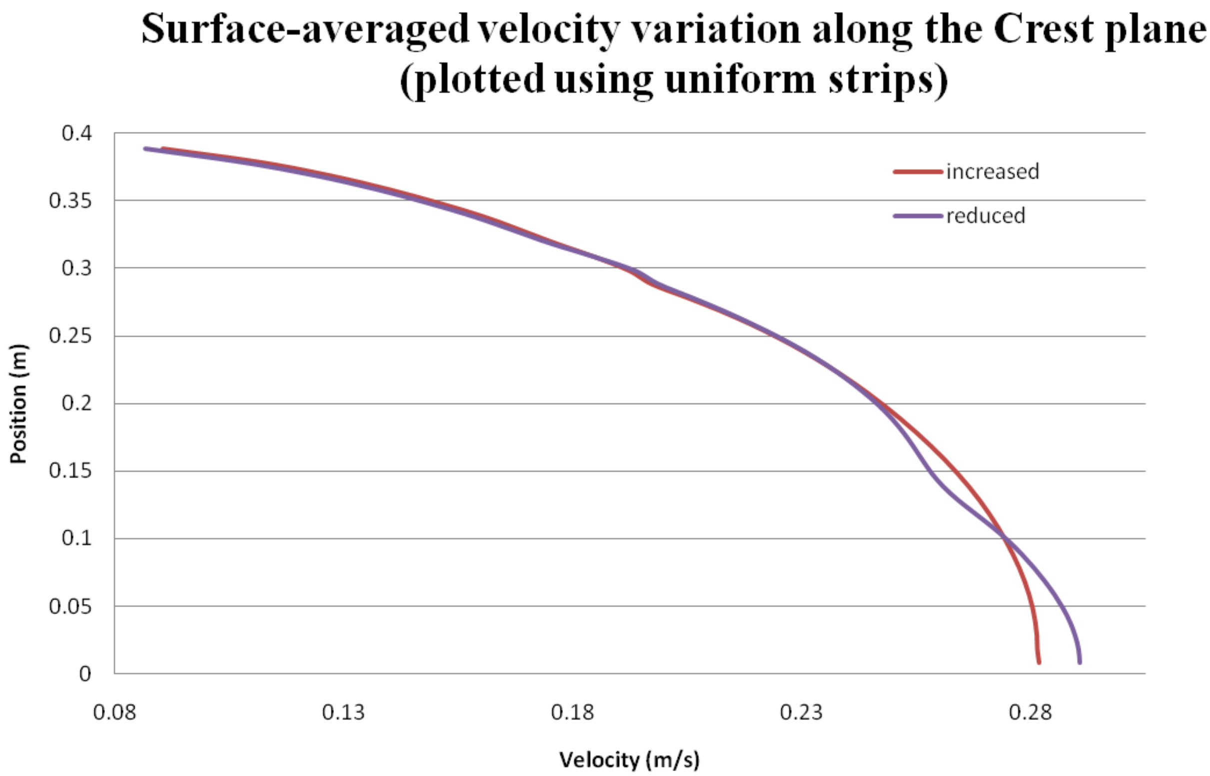



**Figure 2.20 Surface-averaged velocity variation along the uniform strips plotted using " Threholds"**

The origin on the y-axis corresponds to the centerline of the barrel section and the maximum point on the y-axis corresponds to the corrugated wall. It can be seen that for both the increased and reduced section the maximum velocities differ only slightly. The percentage difference in the results for the increased section and the reduced section is small, 3.7% and considered to be adequate for engineering analysis.

Figure 2.21 shows the velocity distribution over the culvert cross section with a porous media gravel bed with velocity near zero. The effect of the porous bed on the flow above is as expected: it acts as a wall-like flow resistance bringing the velocity down near zero at the interface between the free flow region and the porous media. There is a small amount of waviness in the contours approximately midway between the centerline and the outer culver wall that may merit some further investigation. A possible fish passage zone pattern is apparent as a lower velocity yellow region away from the centerline.

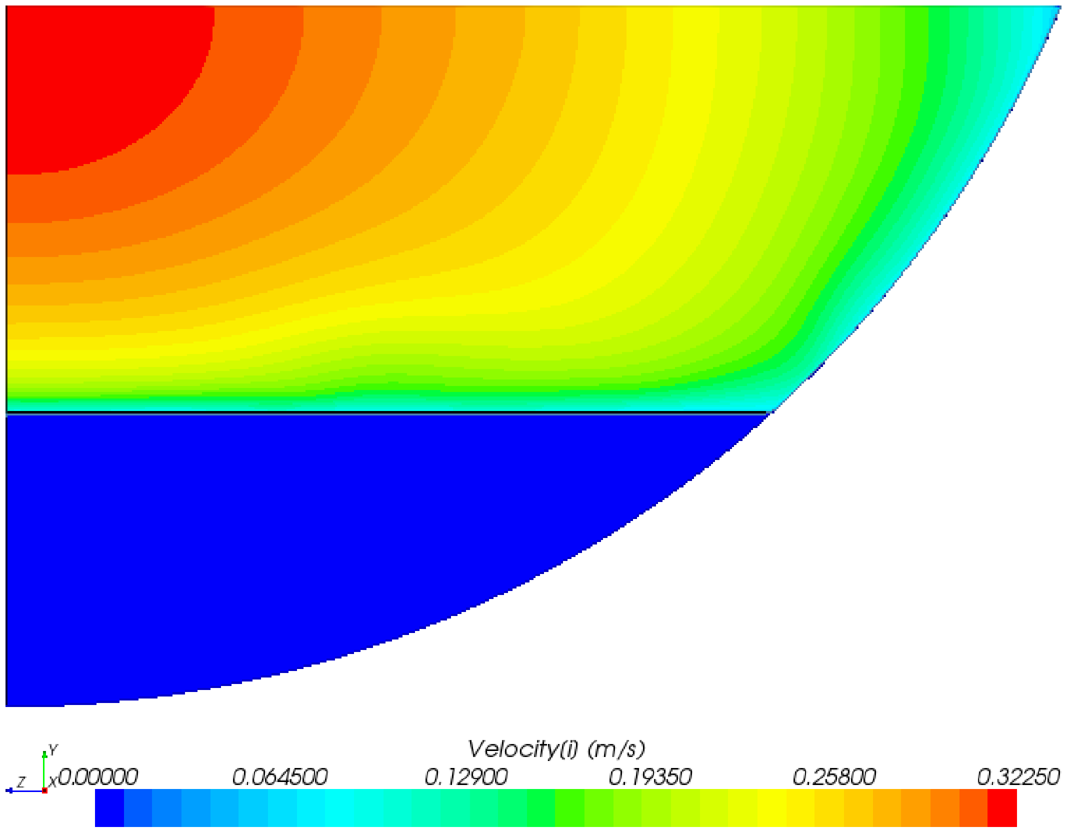



Figure 2.21: Velocity distribution over cross section at a crest showing the variation above the porous media gravel bed

# 3. References

1. 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 assesment 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.