9th ISE 2012, Vienna

PHYSICAL AND NUMERICAL HYDRAULIC MODELING IN SUPPORT OF FISH PASSAGE REGULATIONS

SHAWN P. CLARK, JONATHAN S. TOEWS
Department of Civil Engineering, University of Manitoba
Winnipeg, Manitoba, R3T 5V6, Canada

MARTIN HUNT, ROB TKACH
Ecohydraulics Studies, Manitoba Hydro, 360 Portage Ave.
Winnipeg, Manitoba, R3C 0G8, Canada

Corrugated metal pipe culverts are commonly selected for stream crossings throughout the world. The associated reduction in cross sectional flow area causes an increase in water velocity at the culvert, which may become a barrier to fish passage. Current regulations compare the swimming performance of fish that may use the culvert to the average velocity within the culvert at some design flow. To facilitate providing a more complete picture of the velocity distribution within the culvert, this paper will present results of the streamwise velocity distribution within the fully developed zone of CMP culverts flowing at uniform depth, as well as detailed velocity distributions near the culvert inlet under different inlet treatments. In addition, results from numerical modeling of velocity distributions within culverts using FLOW3D will be presented and compared to an empirical model for streamwise isovel predition. These results have the potential to be used during the design phase of a culvert installation to assist with determining whether the culvert will in fact present a velocity barrier to fish passage.

1 INTRODUCTION

Historically, stream crossings have been designed using considerations such as hydraulic efficiency, cost, reliability, safety, and other parameters. Corrugated metal pipe (CMP) culverts have been commonly installed at stream crossings because they can be sized to safely and reliably pass the design flow at a relatively low cost as compared to bridges. The associated channel constriction caused by culverts will result in a higher water velocity within the culvert than the upstream and downstream channel. In the past, this higher water velocity was generally not a concern. In fact, since higher velocities were likely the result of smaller, less expensive culverts, a high culvert velocity could be looked upon favorably. The head loss across a culvert crossing was sometimes minimized by inlet treatments to guide the upstream water as efficiently as possible through the culvert. This would result in lower head water elevations at the given design flow, and thus a smaller likelihood of upstream flooding.

Presently across North America, stream crossings on fish bearing streams are no longer designed in this manner. In order to minimize anthropogenic effects on aquatic species (particularly fish), culverts are now designed with some sort of velocity criteria enforced by a regulatory agency. Of all the aforementioned design considerations, this velocity criteria almost invariably governs the stream crossing design. This criteria generally states that the average water velocity (design discharge divided by cross sectional area) in the culvert should not exceed the swimming abilities of the design fish species. The culvert barrel length can come into play as well. Regulations may allow shorter culverts to have higher velocities than longer culverts.

The simplest way to accommodate these velocity criteria at a given stream crossing is to allow a greater cross sectional flow area within the culvert. This can be accomplished using larger culverts or crossings with multiple culvert barrels. More recently, the concept of stream simulation crossings has been gaining popularity.

All of these measures will incur an additional cost beyond that of a single barrel culvert sized only to have sufficient hydraulic capacity. This additional cost can be considerable at individual crossings, even without considering the many crossings that government agencies often construct in a given season. The taxpayers that incur these additional costs deserve to know that their money is being spent in a manner that effectively facilitates fish and other aquatic species to fulfill their various life cycle stages. If culverts are ‘over-designed’, and provide a more than adequate amount of cross-sectional flow area with water velocities much less than the swimming capabilities of local fishes, one could take the view that the taxpayer money was spent imprudently. Conversely, if current regulations allow water velocities that pose a velocity barrier to fish passage they are not serving their intended purpose.

Being good environmental stewards is a commendable goal, and in the case of stream crossings this can be facilitated through the use of scientifically-sound regulations. To ensure that these regulations accommodate the actual needs of the relevant aquatic species, our understanding of fish swimming performance as well as the flow characteristics within culverts must be improved. A research program has been developed at the University of Manitoba to investigate the complex flow conditions within partially-filled CMP culverts flowing at uniform depth. The region of rapidly varying flow near the culvert inlet has also been investigated in detail. Finally, empirical equations and commercially available software programs have been used to assist culvert designers with their task of estimating the flow characteristics within a CMP culvert.

2 BACKGROUND

While culverts have been studied from the perspective of trying to achieve the maximum flowrate through a given culvert at a given upstream headwater level, comparatively little work has been done on investigating the complex flow conditions within partially filled culverts. Ead *et al*. [3] used a 0.6 m diameter culvert in a lab and measured the velocity field using a Prandlt tube. The streamwise velocities were described by an empirical equation that they developed based on their results. Magura [7] performed laboratory tests on a CMP culvert with a projecting end inlet and measured the water velocity distribution using an Acoustic Doppler Velocimeter (ADV). Abbs *et al*. [1] investigated the velocity and turbulence intensity within a CMP culvert flowing under a backwater effect. Clark and Kehler [2] and Kehler and Clark [5] used the same laboratory setup that will be described in this paper to investigate the flow conditions within the fully developed region of partially-filled CMP culverts flowing at uniform depth. The three components of water velocity were measured using a Sontek MicroADV. Water velocities, development length, turbulence parameters and fitted boundary shear stress distributions were reported. Empirical equations for the cross-sectional streamwise velocity distribution, the boundary shear stress distribution and the centerline turbulence intensity profile were developed and were found to compare well to the measurements.

3 EXPERIMENTAL SETUP

To gain a better understanding of the flow conditions within a typical CSP culvert, a physical model was constructed at the University of Manitoba’s Hydraulics Research & Testing Facility. This 21 m long, 0.8 m diameter culvert had 13 x 68 mm annular corrugations and was able to be placed at slopes ranging from 0 to 1.5%. Access holes were cut in the obvert to allow water velocity measurement using Sontek acoustic Doppler velocimeters (down-, side-, and upwards-looking) that were maneuvered throughout the culvert cross section with an automated traversing mechanism. Water level was measured using manometers placed along the centerline of the culvert invert.

4 OBJECTIVES

Since 2009 a research program at the University of Manitoba has been ongoing with the overall objective of providing a better understanding of culvert hydraulics for the purpose of facilitating more scientifically-based fish passage regulations. The initial focus was on fully developed uniform flow with and without gravel embedment. Results from this portion of the work can be found in Kehler [5] and Clark and Kehler [2]. During these tests it was noted that flow conditions near the inlet region of the culvert were dissimilar from the fully developed region because of the rapid acceleration and deceleration caused by the constriction. It was hypothesized that conditions could exist where the fully developed portion of a culvert would allow upstream fish passage but where the conditions near the inlet could present a velocity barrier to fish passage. A series of tests were developed to map the flow conditions near the culvert inlet for a variety of inlet configurations in order to use this data to assess whether in fact this region could present a barrier. Finally, a project was undertaken to assess methods of estimating water velocity distributions within a CSP culvert. This was done from the perspective of providing practical guidance and tools to hydraulic engineers at the design phase.

Each of these objectives, summarized as Fully Developed Uniform Flow, Culvert Inlets, and Isovel Predictions will be discussed in more detail within this paper.

5 FULLY DEVELOPED UNIFORM FLOW

As previously mentioned, much of the work on this topic has been reported in Kehler [6] and Clark and Kehler [2], however, up to this point the results from the embedded tests have been unreported publically. In Manitoba, circular CSP culverts installed at crossings that require fish passage are mandated to have the invert embedded with gravel to a height of 10% of the diameter or 0.3 m, whichever is greater. As there appeared to be little to no research that reported the hydraulic conditions within such an installation prior to starting this project, it was apparent that a knowledge gap existed.

Figure 1 shows the streamwise isovels for fully developed flow with no embedment and with 30% gravel embedment. The isovels are distinctly different. By inserting gravel embedment the water level is at a higher location within the culvert, and the cross sectional area becomes nearly rectangular in shape. Flow conditions therefore more closely resemble flow in a rectangular flume than flow in a circular culvert. The maximum water velocity moves closer to the water surface, and the individual isovels become more rectangular in shape. The bottom corners of the culvert have relatively low water velocities. The total percentage of cross sectional area with a streamwise water velocity less than U_{avg} is greater when gravel is placed on the bottom of the culvert.

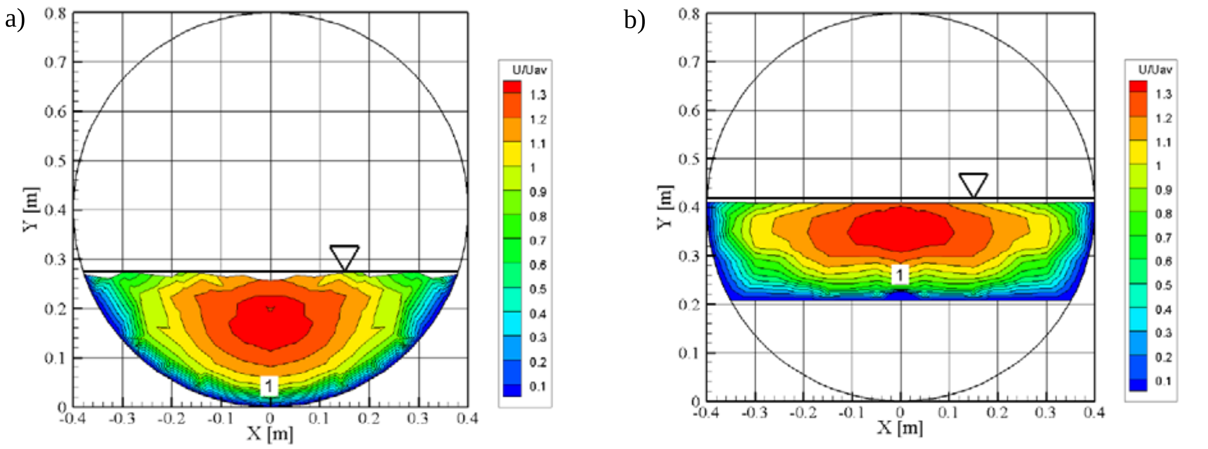

Figure 1. Normalized isovels for a) non-embedded fully developed flow, and b) fully developed flow with 30% gravel embedment.

6 CULVERT INLETS

The headwater box measured 4.5 m long and 2.75 m wide, which resulted in a box width to culvert diameter ratio of 3.4. The flow entering the upstream end of the headwater box passed through a flow straightener before reaching the culvert inlet. Three different inlet configurations were used. The simplest was a vertical headwall that was flush to the end of the culvert. A 45° wingwall inlet was also used, where the wingwalls extended from the edge of the culvert to the headwater box walls. The projecting end inlet had a simulated 3:1 (horizontal : vertical) sloped embankment that passed through the culvert lip at a height of 10% of the diameter from the invert. The ADV's collected a set of 12,000 samples at each point location. The sampling rate of the 16 MHz and 10 MHz ADV's were 50 and 25 Hz, respectively.

The flow structure was assumed to be symmetrical about the culvert centerline, which allowed us to sample only one half of the culvert. The traversing mechanism was set to sample at 4 locations: 0.25, 0.5, 1 and 2 times the culvert diameter downstream of the inlet. Measurements were made on an offset grid pattern spaced 2.5 cm in the horizontal and 5 cm in the vertical.

A typical sample of the results is shown in Figure 2a and 2b which were taken at 0.25 and 2 diameters downstream from the projecting end inlet, respectively. The contours represent the streamwise velocity normalized by the average water velocity (U_{avg}) in the fully developed region. Very near the entrance, note the roughly rectangular core of high velocity centred in the culvert, extending over the entire water depth. A nearly vertical line of high shear separates the high velocity core from the recirculation zone on the culvert edge. The velocities here are very low or even negative (ie. water moving upstream). Figure 2b shows that by 2 diameters the velocity distribution has changed significantly, and is well on its way to becoming closer to a typical fully developed velocity distribution.

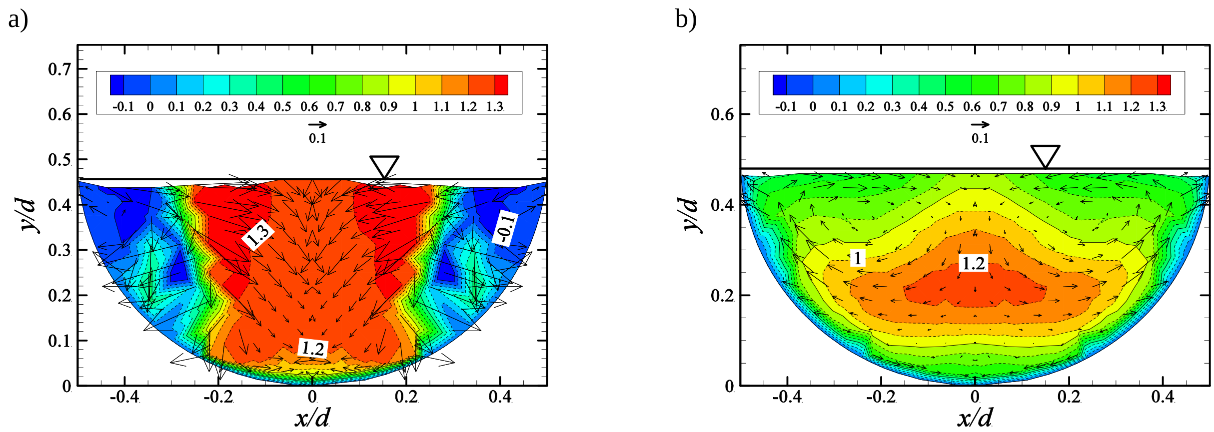

Figure 2. Normalized isovels with secondary circulation vectors at sample locations.

Since current regulations focus on the average culvert velocity, it was of interest to determine the percentage of the cross sectional flow area that had a water velocity less than U_{avg}. Figure 3 shows this percentage for all four streamwise locations for each of the three types of inlets. For each inlet type the percentage of cross sectional area with streamwise velocity less than U_{avg} increases with distance from the inlet. The largest difference between the inlet types can be seen at the cross section located 0.25D downstream from the inlet. The most hydraulically efficient option, the 45° wingwall, has higher water velocities over more of the cross sectional area than the others. This is because the wingwalls are more streamlined than the other options, which reduces the amount of flow separation around the culvert edges at the inlet.

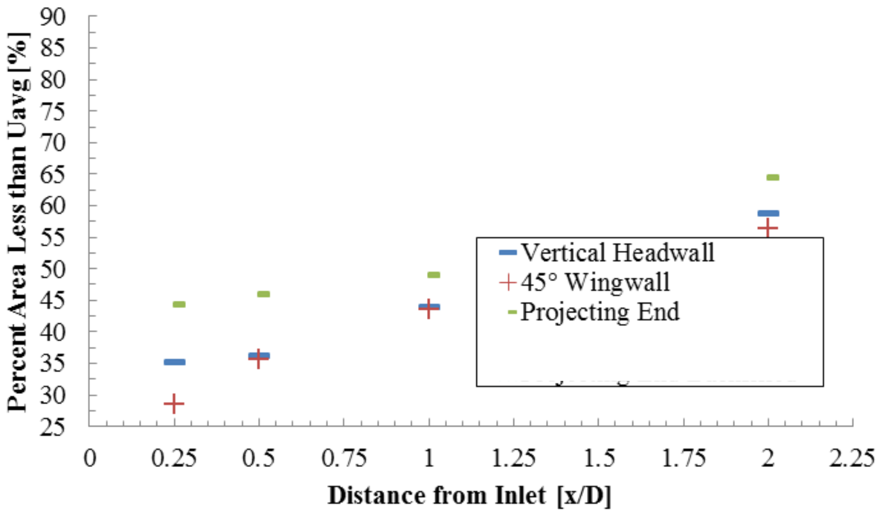

Figure 3. Percentage of the cross sectional flow area with streamwise velocity less than U_{avg}.

7 ISOVEL PREDICTIONS

Rather than using U_{avg} to quantify the flow conditions within a CSP culvert, having a reliable means of predicting the streamwise isovels would allow designers to add an additional dimension to their velocity predictions. While Clark and Kehler [2] developed an empirical means of predicting such isovels, these results were based on a series of 5 laboratory experiments, with the calibration coefficients manually optimized using all available data. It was anticipated that using a commercially available software program could perhaps allow culvert designers to have a more universally applicable tool. The three dimensional finite volume software program Flow3D was chosen because of its wide range of successful applications and its prevalence with local hydraulic engineering companies. The objective of this project was to assess whether Flow3D was an appropriate choice for culvert designers. In order to have calibration and verification data available, several additional physical model tests were conducted in the same manner as Kehler [6] but under different flow conditions.

7.1 Flow3D Model Development

The headwater box (vertical headwall inlet) and culvert were separated into two blocks and meshed separately. The full 21 m of culvert length was used in the mesh. Early attempts at assuming symmetry and modeling only half of the culvert were unsuccessful. Simulations were run to steady state or 700 seconds of simulation time due to quasi steady state conditions occurring. The steady state conditions within the model were set so that the change in total mass, total fluid energy, total solid energy, average mean kinetic energy, average mean turbulent energy and average mean turbulence has reached a value of less than 1.0%.

Since the purpose of this assessment was to evaluate the model for practical design purposes, the run times were of particular importance. During the design phase, one can often proceed through an iterative procedure, changing culvert diameter, roughness, inlet treatment, etc. It was therefore impractical to have simulation times that could be measured on the order of weeks. This had a direct implication on the allowable grid size of the mesh. For instance, the culvert corrugations could not be resolved in the mesh. Instead, the culvert was modeled as a smooth surface and an appropriate roughness value was applied after calibration. After some trial and error, an appropriate mesh was selected. In Figure 4, the streamwise water velocity profile at the culvert centerline are compared for several different meshes. Mesh 3 was selected for further use, as the additional computation time associated with Mesh 4 and beyond were considered to be unwarranted.

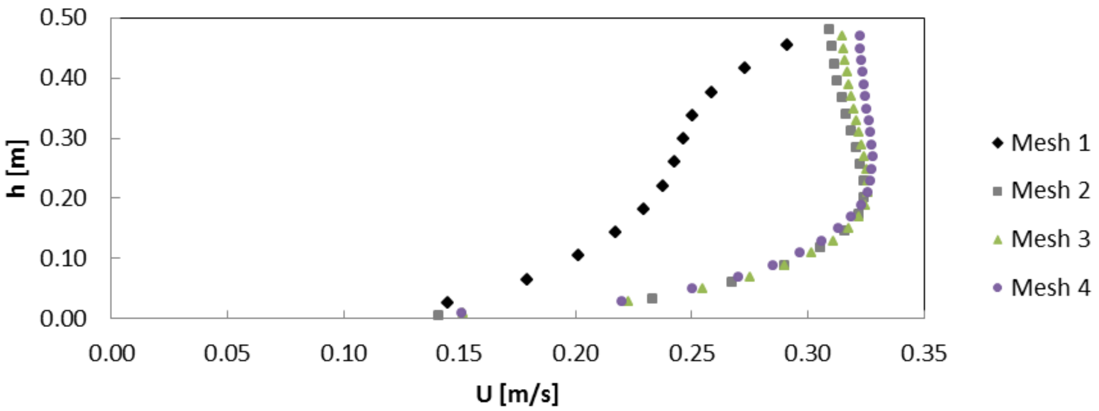

Figure 4. Centreline streamwise velocity profiles for several Flow3D runs using different meshes.

When simulating the experimental test conditions, run times varied, but were typically measured on the order of days and were generally less than one week. This length of time may be considered unacceptable by some potential users.

An upstream flow rate and downstream pressure (ie. water level) were specified as boundary conditions, where the downstream level was determined by calculating the normal depth for the given conditions. Further boundary conditions such as wall and symmetry boundaries were identified as necessary in the model. Initial conditions for the upstream headwater box and the culvert were set to be equal to the flow rate divided by the estimated cross sectional area in each location. The minimum time step was set exceptionally low at 0.000001 s to ensure that the time step would not force the model to exit before steady state conditions had been met.

Of the multitude of physics models that can be utilized within Flow3D, only two were used here. The first was the gravity model. Gravity in the z-direction was set to 9.806 m/s^2, while gravity in the x-direction was adjusted to take into account the slope of the culvert model. This was done to allow the model to be set up horizontally to prevent irregular fragments along the length of the culvert barrel within the CFD model. The second physics model that was utilized was the viscosity and turbulence model. The viscous flow setting was used to better model the flow characteristics within the culvert. Additionally, flow was assumed to be turbulent and the Renormalized group (RNG) turbulence model was utilized. A series of tests were done and it was determined that the RNG model was the most efficient in this particular case, while still providing acceptable results. Additionally, the RNG model is generally the most robust transport turbulence model available within the Flow3D software (Flow Science [4]).

7.2 Clark and Kehler(2011)

As previously mentioned, additional physical model experiments were conducted in order to have sufficient experimental data for verification of the Flow3D model. This also presented the opportunity to compare the results from these tests to the empirical predictions of Clark and Kehler [2]. The benefit of this model is that the main two equations can be coded in a software program such as Matlab, or even using Microsoft Excel. Once coded, streamwise isovels for a particular culvert design can be estimated in a matter of minutes, rather than days, which provides a much greater opportunity for some level of iteration in one's culvert design. The equations can be found in Clark and Kehler [2] and will not be repeated herein.

7.3 Modelling Results

Figure 5a shows the measured results from Test 10 of the physical model. The black dots denote measurement locations, however, note that measurements were only taken from half of the culvert and were mirrored across the centerline. As one might expect, the measured water velocity near the wetted perimeter of the culvert is very low, while the water velocity near the centre of the culvert is greater. The maximum water velocity is depressed from the water surface. By concentrating on the isovel $U/U_{avg} = 1$, it can be seen that very little of the water within the culvert flows at a velocity equal to U_{avg}. in essence, one could imagine a predicted isovel Again, this is important since current regulations concentrate on U_{avg} alone. Figure 5b shows the predicted streamwise isovels from Flow3D. There is obvious similarity between the Flow3D simulated isovels and the measurements. One area of dissimilarity is near the top 'corners' of the culvert and along the water surface, where the simulation overestimates the water velocity. Figure 5c shows the empirical predictions of Clark and Kehler [2], which also closely resemble the measurements, in particular near the water surface.

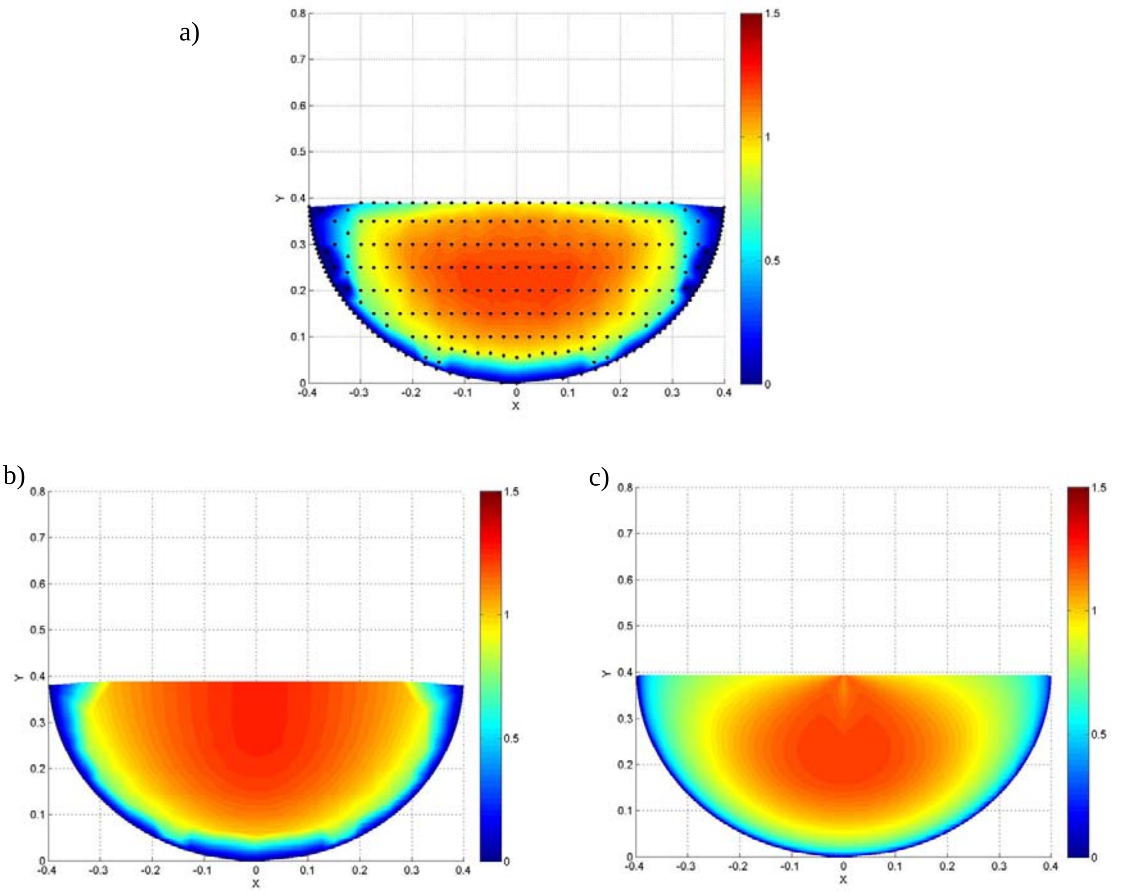

Figure 5. Normalized streamwise isovels for a) measured data, b) Flow3D predictions, c) Clark and Kehler (2011) predictions.

In order to objectively evaluate these prediction methods, the following methodology was used. The measurement isovels were subtracted from the prediction isovels (both already normalized), which resulted in a two-dimensional plot of the percent that the predictions deviated from the measurements. Software was then used to sum up the total flow area where the predictions were within plus or minus some specific percentage of the measurements. These results are summarized in Table 1, where the first row corresponded to the case where the entire flow area was predicted to be flowing at U_{avg}. One can note that only 15.5% of the cross sectional area has a velocity that is within +/-5% of U_{avg}, suggesting that U_{avg} may not be a good estimate of the streamwise velocity distribution within a CSP culvert. By comparison, 54.7 % and 73.6% of the flow area have velocities that are within 5% of the measurements for the Flow3D and empirical models, respectively, indicating that these two methods of isovel estimations are clearly far superior to simply assuming that the water flows at U_{avg}. As expected, as we increase the percent error, a greater percentage of the predicted isovels fall within this error. For the empirical model, 88% of the flow area is predicted with an error of +/-20% or less.

Table 1. Inter-model comparison of the percentage of the total cross sectional area (%A) where the predicted water velocity is within some percentage of the measured water velocity for Test 10 of the physical model.

Prediction Method	**%A +/-5%**	**%A +/-10%**	**%A +/-15%**	**%A +/-20%**
U_{avg}	15.5	32.5	53.3	69.0
Flow3D	54.7	72.0	78.0	80.2
Clark and Kehler (2011)	73.6	84.9	87.1	88.3

8 CONCLUSIONS

In this paper results from a set of physical and numerical model tests have been presented. These results should be of use to culvert designers that are interested in estimating the variation in streamwise velocity near a culvert inlet as well as within the fully developed zone of CSP culverts flowing at uniform depth. Specific conclusions from this work are:

- By adding gravel embedment the cross sectional flow area and streamwise isovels become more rectangular in shape, and there is an increase in the amount of cross sectional flow area that has a streamwise velocity less than U_{avg}.
- Flow characteristics near a CSP culvert inlet are greatly dissimilar from characteristics within the fully developed zone.
- Flow separation at a culvert inlet can cause recirculation zones on either side of the culvert. Water velocities in these zones can be very low, and can even be in the upstream direction.
- The more streamlined a culvert inlet treatment, the less severe of a recirculation zone will be developed. When combined with a lower water depth due to the flow acceleration near the inlet, this causes more of the cross sectional area to have a water velocity that is greater than U_{avg}.
- Estimating that the water velocity within the fully developed zone of a CSP culvert flowing at uniform depth is equal to U_{avg} provides a very simple, yet imprecise view of the streamwise isovels in this location.
- Numerical models such as Flow3D can provide a much more realistic estimate of the streamwise isovels, however, this comes at the expense of long computation times.
- The comparatively simple empirical formulae developed by Clark and Kehler [2] provide a more accurate estimate of streamwise isovels than Flow3D when compared to the experimental results discussed here.

9 ACKNOWLEDGEMENTS

The authors wish to thank the Natural Sciences and Engineering Research Council of Canada, Manitoba Infrastructure and Transportation, Manitoba Hydro, Fisheries and Oceans Canada and the Canadian Steel Pipe Institute for their financial support. The assistance of Garrett Ward with some of the experimental and numerical work is also greatly appreciated.

REFERENCES

[1] Abbs, T.J., Kells, J.A. and Katopodis, C., "A Model Study of the Hydraulics Related to Fish Passage Through Backwatered Culverts", *Proc. 18th Canadian Hydrotechnical Conference, CSCE*, Winnipeg MB, Canada, (2007), 13 p.

[2] Clark, S.P., and Kehler, N. "Turbulent flow characteristics in circular corrugated culverts at mild slopes", *Journal of Hydraulic Research*, Vol. 49, No. 5, (2011), pp. 676-684.

[3] Ead, S. A., Rajaratnam, N., Katopodis, C. and Ade, F. "Turbulent Open-Channel Flow in Circular Corrugated Culverts", *Journal of Hydraulic Engineering*, Vol. 126, No. 10, (2000), pp. 750-757.

[4] Flow Science, Inc., "*Flow-3D User's Manuals*", Version 9.4, Santa Fe, NM. (2009).

[5] Kehler, N., and Clark, S., "A Preliminary Study of Development Lengths within Circular Corrugated Metal Pipe Culverts", "*Proc. 7th International Symposium on Ecohydraulics*", Concepcion, Chile, (2009) paper # conf187a399, IAHR.

[6] Kehler, N. J. "*Hydraulic Characteristics of Fully Developed Flow in Circular Culverts*", M.Sc. thesis, Department of Civil Engineering, University of Manitoba, Winnipeg, MB., (2009).

[7] Magura, C. R. "*Hydraulic Characteristics of Embedded Circular Culverts*", M.Sc. thesis, Department of Civil Engineering, University of Manitoba, Winnipeg, MB., (2007).