Modelling Bridge Piers and Afflux in TUFLOW

The notes below were compiled from TUFLOW Support emails relating to modelling bridge piers in 2D and estimating the bridge afflux; always an interesting topic for numerical modellers!

The model provided uses a 5m grid and represents the bridge piers using polygons in a 2d_zsh layer to modify the elevations in and around the 2D cells. The results are presented in Figure 1 and Figure 2 which shows the cell resolution. It can be seen that the main piers (2 larger central piers) are influencing the predicted flow patterns, whilst the smaller outer piers are not.

The central larger piers are effectively blocking out two cells (10m) across the flow. This is over-representing the size of these piers which are 7.5m wide.

The thin polygons that represent the smaller bridge piers raise a single line of cell sides parallel to the flow. The cell sides modified are shown in red in Figure 3. Whilst the flow across the channel (perpendicular to the flow) is blocked, there is no influence in the primary direction of flow.

The outcome is that the 7.5 piers are too wide and the 1.2m piers are not represented. The net effect is that the overall waterway blockage is about correct.

Ultimately the critical issue is whether the correct afflux is being produced by the model.

The preferred approach in TUFLOW is to use 2D flow constriction shapes (2d_fcsh layer), rather than blocking out cells, especially if the piers are represented by only a few cells or less.

In TUFLOW, 2D flow constrictions allow you to

- restrict the flow width of the cells;
- introduce additional losses (to represent sub grid features such as the bridge piers); and
- place a lid (obvert / soffit / low chord) on the cells.

In this case the water is not predicted to submerge the bridge deck, so only the additional losses and the restriction of flow width parameters apply. Flow constriction shapes are described in section 4.7.2 of the 2010 TUFLOW manual.

Two approaches of applying flow constrictions were tested:

1. A single flow constriction shape is applied across the entire waterway; and
2. Flow constrictions are placed only at the TUFLOW cells that contain the bridge piers.

The first approach replicates that for applying losses according to industry standard publications such as “Hydraulics of Bridge and Waterways” (Bradley 1978). This publication can be found online at:
http://www.fhwa.dot.gov/engineering/hydraulics/pubs/hds1.pdf.

The second approach is an adaptation of the same logic but allows the modeller to apply the parameters only to the 2D cells that contain the bridge piers. This has the advantage of producing more realistic flow patterns, and representing each bridge pier individually rather than lumped together.

Hydraulics of Bridge and Waterways estimates the additional losses based on the pier shape and fraction of the waterway blocked. A backwater coefficient is determined using the charts provided, and is applied in TUFLOW as an energy loss based on the velocity head.

$$\Delta h = \zeta_a \frac{V^2}{2g}$$

Where ζ_a is the calculated coefficient, based on the figures/charts in Bradley (1978). This figure is reproduced here in Figure 4.

For calculating the J value (fraction of the area blocked by piers), the entire bridge opening is determined for the first approach (one form loss value for the entire bridge, see Figure 5), and for each individual span for the second approach as discussed below (example pier shown in Figure 6).

Approach 1:
The entire bridge the length is approximately 270m and there are 19.8m of bridge piers (4 by 1.2m piers and 2 by 7.5m piers). This equates to a Blockage fraction (J value) of 0.073. Using the Hydraulics of Bridge and Waterways chart (Figure 4) and assuming a single rounded pier shape a form loss coefficient (ΔKp) of 0.14 is determined. This method is presented visually in Figure 5 and Figure 7.

Approach 2:
The calculation is repeated for each individual pier by assessing the flow area blocked on a span by span approach, in a similar method to a parallel channel analysis. For each pier, the flow width from mid pier to mid pier is calculated, and the pier width is used to calculate a J value for each pier. This is shown visually in Figure 6.

Using the same method as above a pier coefficient (ΔKp) can be determined for each pier. As the flow constriction is only applied to the TUFLOW cells where the bridge piers are, this loss (ΔKp) is factored, based on the number of cells in the span and the number of bridge pier cells. For example, if the bridge span is 50m (as in Figure 6), this is 10 TUFLOW cells, however the 7.5m pier is only modelled with 2 TUFLOW cells. In this case the calculated ΔKp is multiplied by 5 (ie. 10/2).

The additional (ΔKp) form loss values and blockage width to the TUFLOW cells are applied using the 2d_fcsh layer attributes. For this second approach care has been taken to apply this to the correct number of TUFLOW cells.

Analysis:
For both of the approaches described above the flow constrictions have been setup with and without additional blockages being applied to the TUFLOW cells. The logic

behind this is that the Hydraulics of Bridge Waterways accounts for the blockage and so including the blockage on the TUFLOW cells may over-represent the losses.

A comparison of the different approaches is presented below. The five approaches analysed are:

Table 1 Bridge Pier Modelling Approaches

Scenario	Description
No Bridge	No bridge piers included
Scen1	Bridge piers included by blocking out the cells (raising elevations). This is the method used in the model provided to TUFLOW Support.
Scen2	Single form loss coefficient applied across entire waterway, ie. form loss applies to all cells across river. Form Loss Coefficient (FLC) = 0.14
Scen3	As for Scen2 but percentage blockage also applied, ie. loss and blockage applies to all cells across the river. Form Loss Coefficient = 0.14, blockage = 7.3%
Scen4	Form loss calculated per bridge span and applied only at pier cells. No blockage applied. The FLC values adopted for the smaller piers were 0.07 (1.2m pier in 30m waterway), this was factored to 0.42 to apply only at the pier cells. For the larger piers the unfactored FLC was 0.32 which was factored to 1.60, to apply only at the pier cells..
Scen5	As for Scen4 but blockage also applied. For example at a 1.2m pier with a 5m cell, a 24% blockage is applied at pier cells.

The velocity magnitudes and directions for the methods above are presented in Figure 8 through to Figure 13.

A water level profile was extracted in the vicinity of the bridge for each scenario; the location of the profile extraction is presented in Figure 14. The water level profiles are shown below in Figure 15, for the 20 Year event.

Predicted velocities in the vicinity of the bridge are generally less than 1 metre per second. Average velocities through the bridge section range from 0.54 to 0.61 m/s (depending on the method of modelling the bridge). These are detailed in the table below for the 5 scenarios. The average velocities in the table below are calculated from the flow through the bridge divided by the flow area through the bridge.

Table 2 Average Velocities Through Bridge Structure

Scenario	Average Velocity (m/s)
No Bridge	0.54
Scen1	0.61
Scen2	0.54
Scen3	0.58
Scen4	0.54
Scen5	0.60

These low velocities produce the small affluxes shown in Figure 15. An afflux calculation for the piers based on the Hydraulics of Bridge Waterways method was carried out as an independent check. The values used were the predicted average velocity and the calculated incremental backwater coefficient for piers (ΔKp) of 0.14. The afflux calculated is 2.6mm (see Table 3).

Table 3 Afflux Due to Bridge Pier Desktop Calculations

Parameter	Description
0.61	Predicted Average Velocity (m/s)
0.019	Dynamic Head (v2/2g)
0.00263	Afflux (m)
2.63	Afflux (mm)

Affluxes were calculated across the bridge by subtracting the water level at chainage 390m from 490m. For each of the methods of modelling the bridges, the total afflux is calculated as well as the increase in afflux compared to the scenario without the bridge in the model. The afflux has been calculated as the increase in water level upstream due to the bridge.

The approaches that block out the individual 2D cells (Scen1 and Scen5, 4.4 and 4.9mm) overestimate the afflux compared with the Hydraulics of Bridge Waterways method (2.6mm).

Table 4 Modelled Affluxes 20 Year Event

Scenario	Total Head Drop (mm)	Afflux (Compared to No Bridge) (mm)
No Bridge	3.3	-
Scen1	7.7	4.4
Scen2	5.5	2.2
Scen3	5.9	2.5
Scen4	6.0	2.6
Scen5	8.2	4.9
HBW	---	2.6

In order to test the losses under higher velocities, the model was run with a multiplier of five (5) on the flows whilst the downstream tail water was left unchanged. The average velocities for this case are presented below in Table 5, with the highest velocities being around 2.5m/s for this event.

The velocity magnitude and directions for the 5 x 20yr inflow event are presented in Figure 16 through to Figure 21.

Table 5 Average Velocities Through the Bridge Structure; 5 x 20yr Inflows

Scenario	Average Velocity (m/s)
No Bridge	1.87
Scen1	2.04
Scen2	1.87
Scen3	2.01
Scen4	1.87
Scen5	2.08

The profiles for this case are presented below in Figure 22. The afflux calculation using Hydraulic of Bridge Waterways is presented in Table 6. This afflux calculation depends on which modelled velocity from the table above is used, the afflux ranges from 25mm to 31mm.

Table 6 Afflux from Desktop Calculations; 5 x 20yr Inflows

Lower Range	Upper Range	Description
1.87	2.08	Predicted Average Velocity (m/s)
0.178	0.220	Dynamic Head ($v^2/2g$)
0.0250	0.0309	Afflux (m)
25.0	30.9	Afflux (mm)

The modelled affluxes across the bridge structure for the five scenarios are presented in Table 7.

Based on the results Scen2 (28mm) gives the closest results to the Hydraulics of Bridge and Waterways calculation of 25mm to 31mm. The case with the bridge modelled by blocking out the cells (central piers only), produces the highest afflux (51mm) and overestimates the losses, compared to the Hydraulic of Bridge Waterways calculation.

Table 7 Affluxes Across Bridge; 5 x 20yr Inflows

Scenario	Total Head Drop (mm)	Afflux (Compared to No Bridge) (mm)
No Bridge	28	-
Scen1	79	51
Scen2	56	28
Scen3	61	33
Scen4	61	33
Scen5	68	40
HBW	---	25 to 31

Conclusion:
Whilst the approaches that remove or block the cells at the piers produce what would appear as more realistic flow patterns, care needs to be taken to ensure that the affluxes predicted are representative. This model and other modelling carried out using a variety of 2D software (fixed grid and flexible mesh) tends to show that blocking out cells/elements for bridge piers will overestimate the afflux compared to the Hydraulics of Bridge Waterways approach.

There are a number of methods in TUFLOW for modelling of bridges that allow the modeller to apply parameters from publications such as the Hydraulics of Bridge Waterways. As with any numerical model outputs, it is essential that these models are used within their capabilities to ensure the results produced are truly representative of the bridge afflux.

In this document we have not explored TUFLOW's capabilities for modelling bridges in 2D once the bridge deck has become submerged. TUFLOW is able to model this situation, but like all modelling, cross-checking through benchmarking to established publications and theory is an important part of the modelling process.

For complex or poorly understood obstructions to flow, Computational Fluid Dynamics (CFD) may be required as another means of establishing the losses that can then be applied to 2D solvers such as TUFLOW.

Figures:

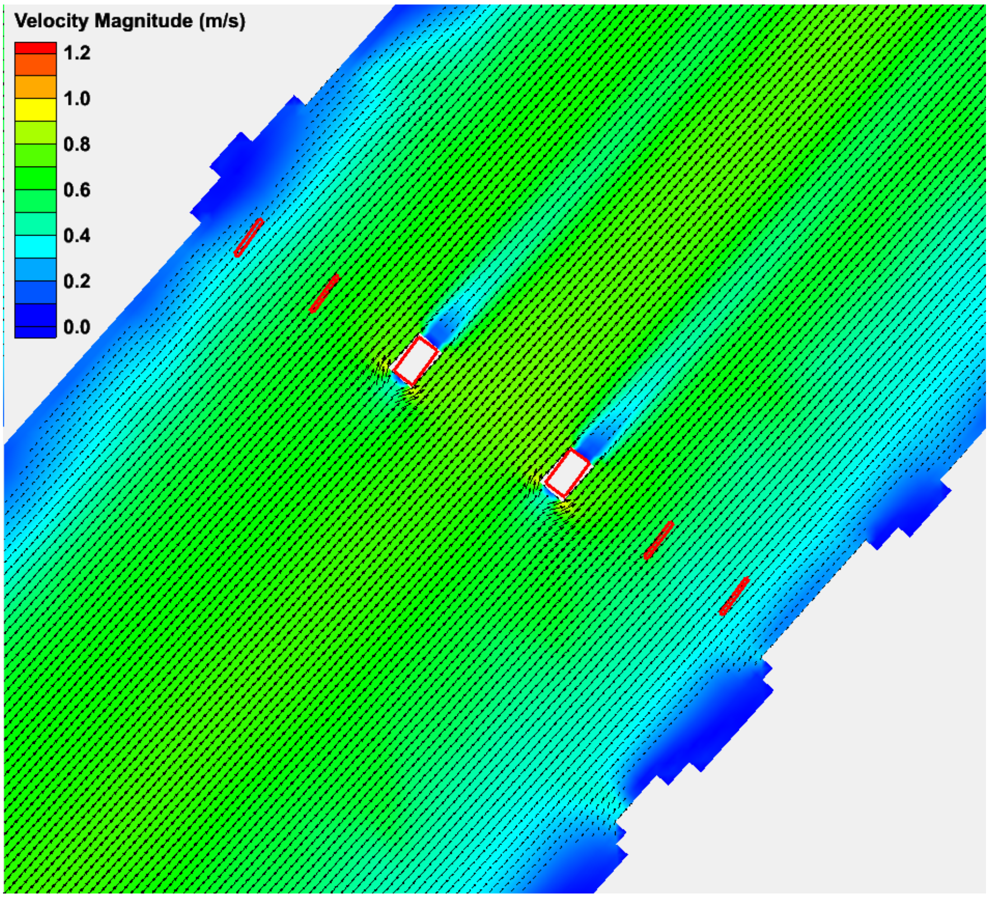

Figure 1 Velocity Magnitude and Direction; Piers Modelled by Modifying the Zpts

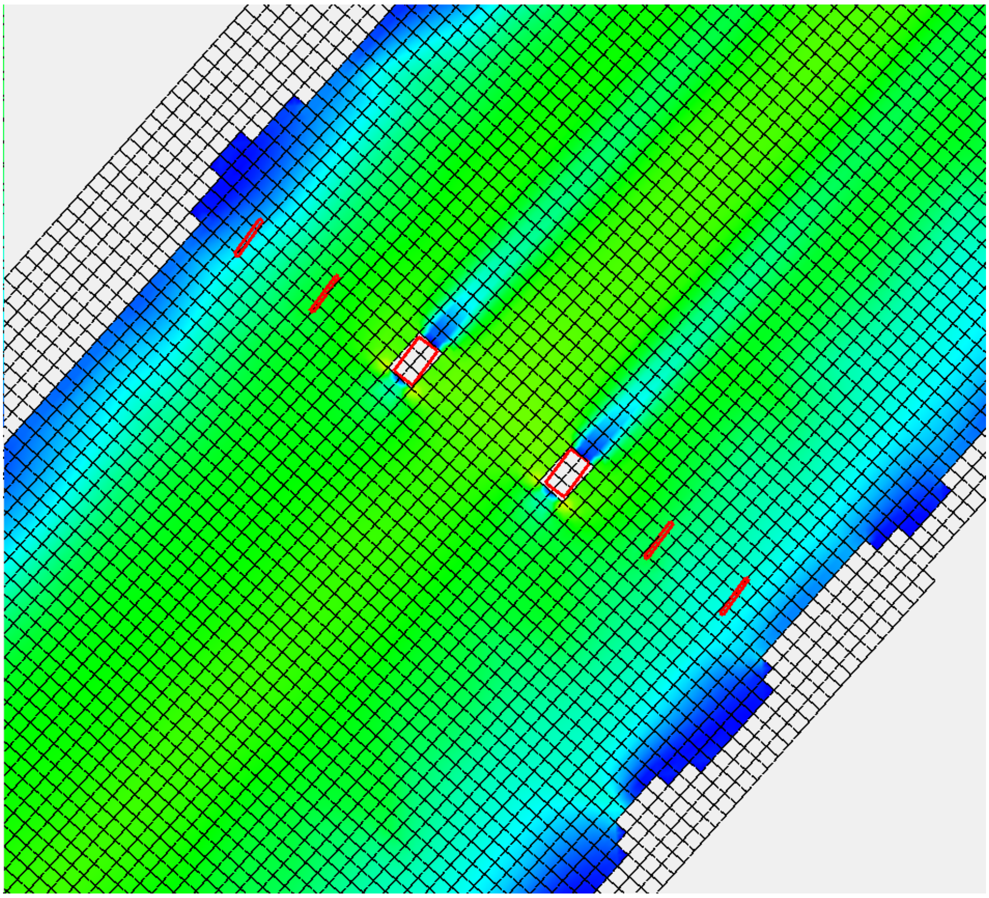

Figure 2 TUFLOW Grid Resolution

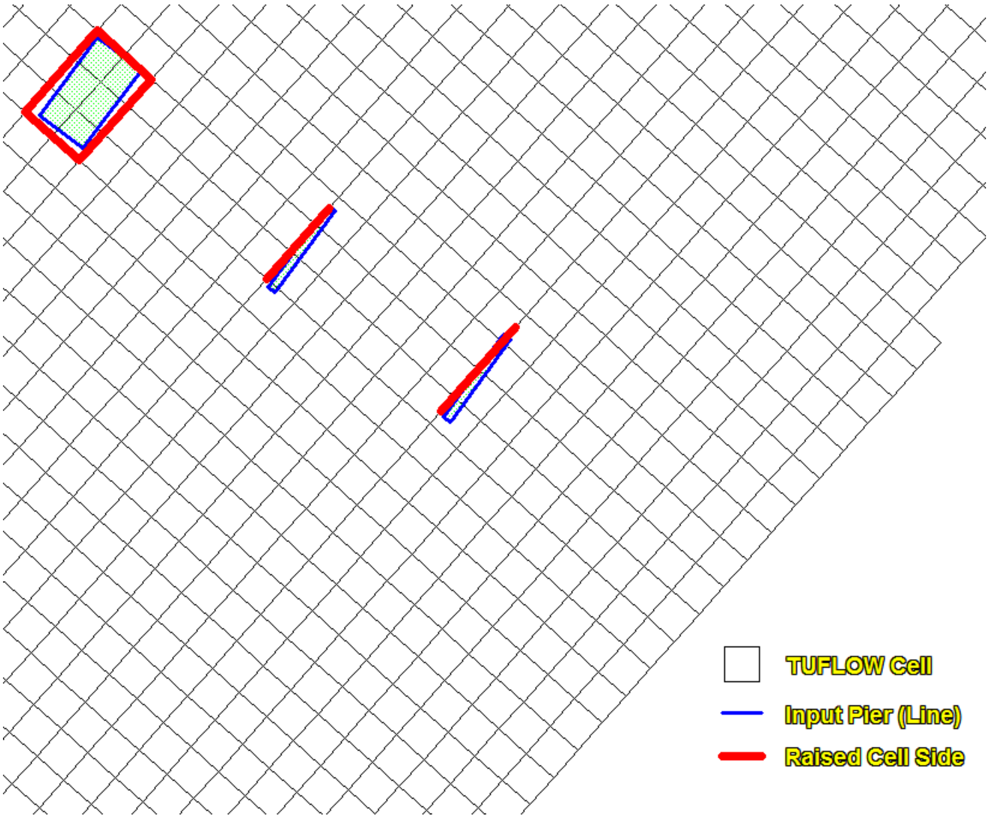

Figure 3 TUFLOW Cells and Pier Locations; 5m model

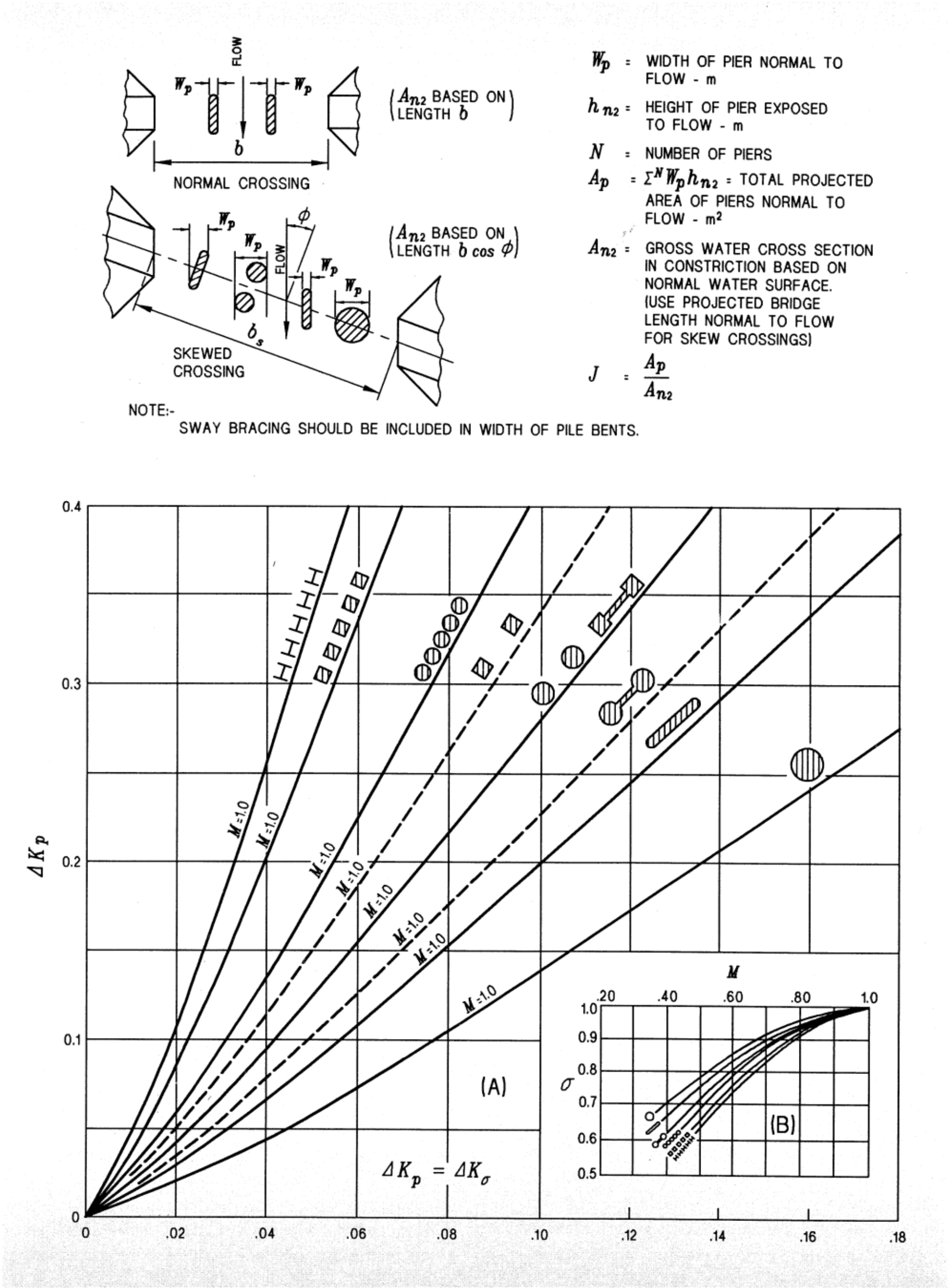

Figure 4 Incremental Backwater Coefficient for Piers (Hydraulics of Bridge and Waterways, 1978)

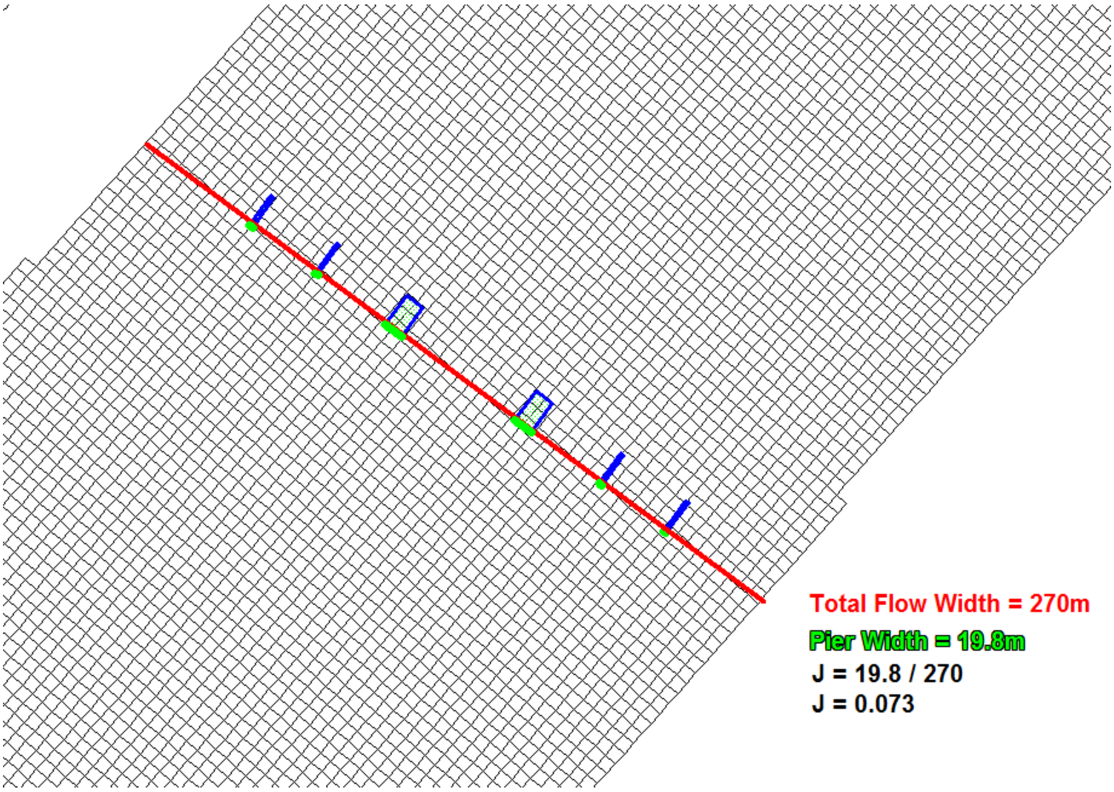

Figure 5 Blockage Calculation for Entire Bridge (Cell Size Independent)

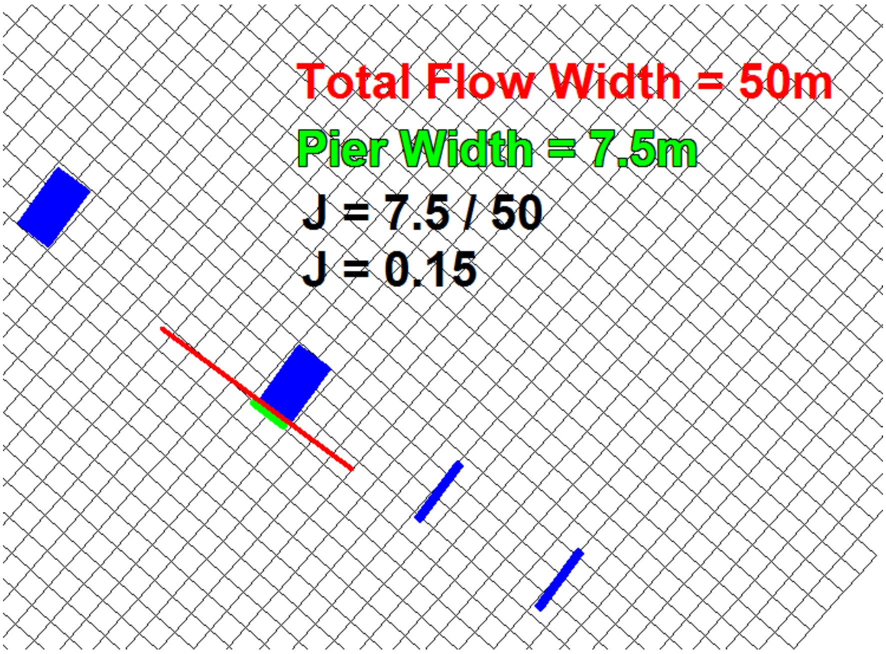

Figure 6 Blockage Calculation at Example Span

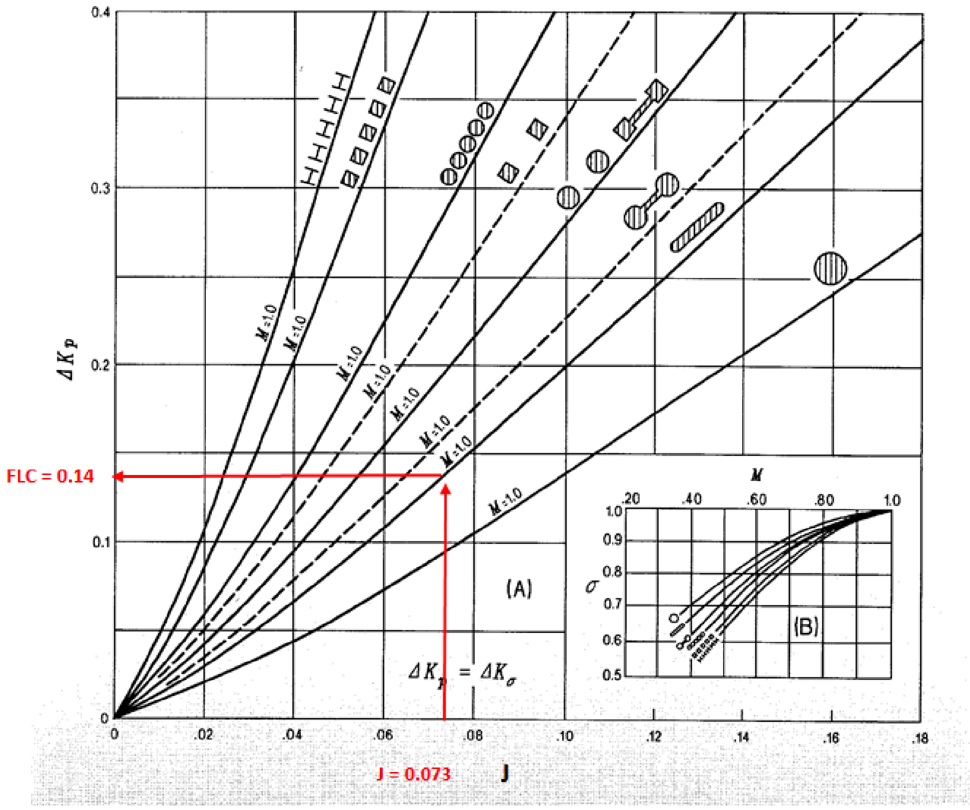

Figure 7 Formloss Coefficient for Entire Bridge

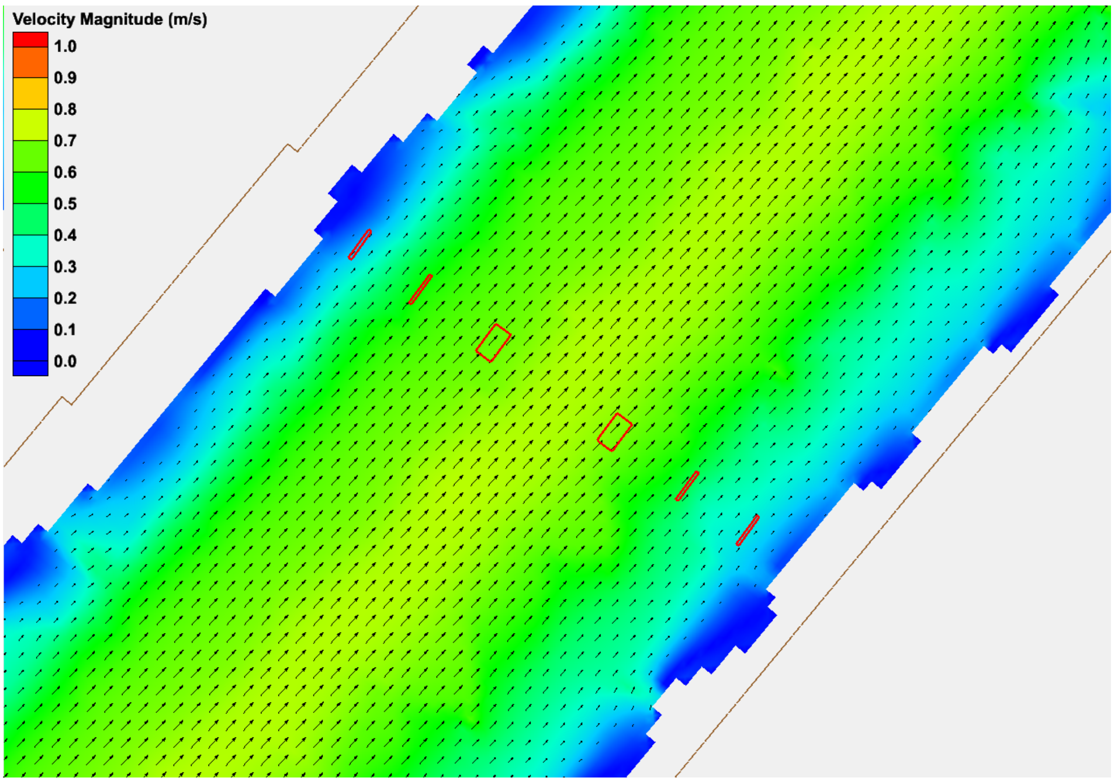

Figure 8 Velocity Magnitude and Direction, 20yr Inflow; No Bridge

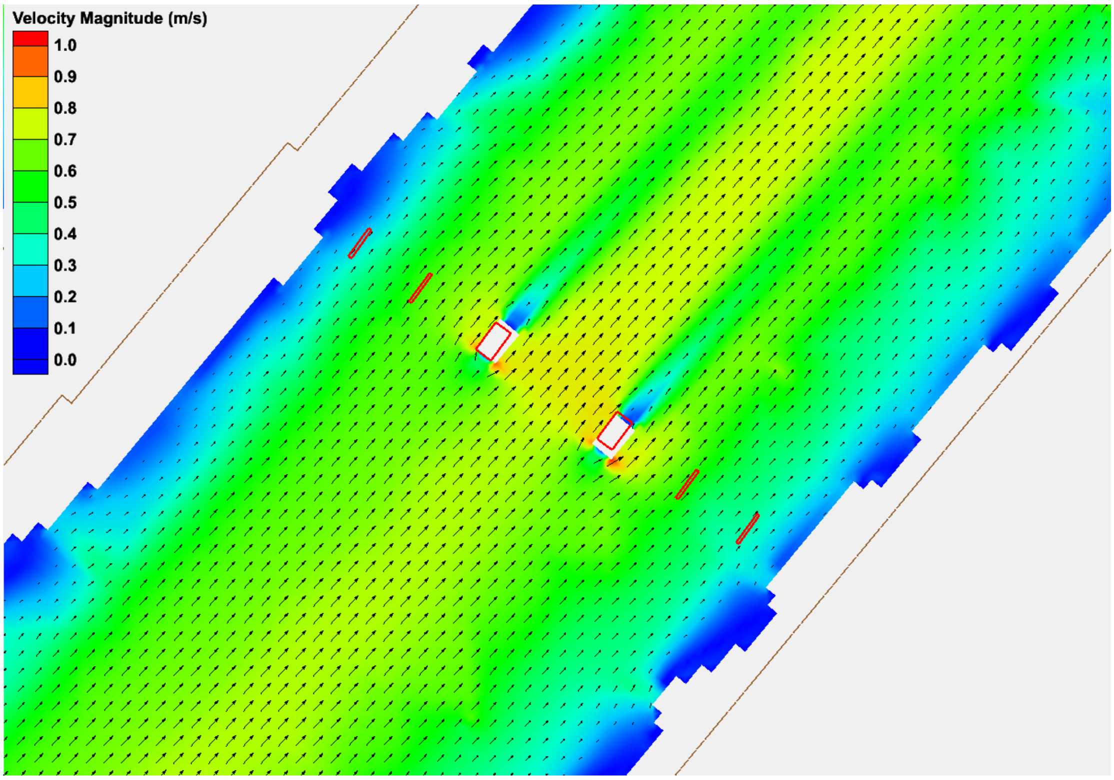

Figure 9 Velocity Magnitude and Direction, 20yr Inflow; Scen1

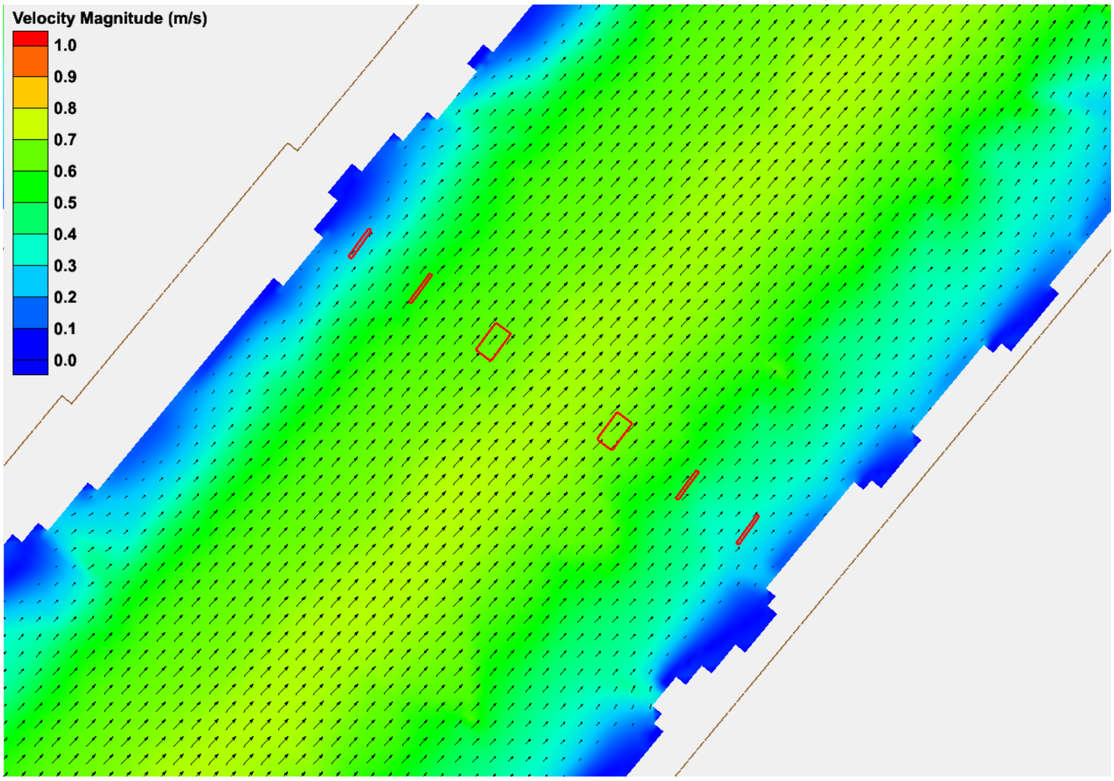

Figure 10 Velocity Magnitude and Direction, 20yr Inflow; Scen2

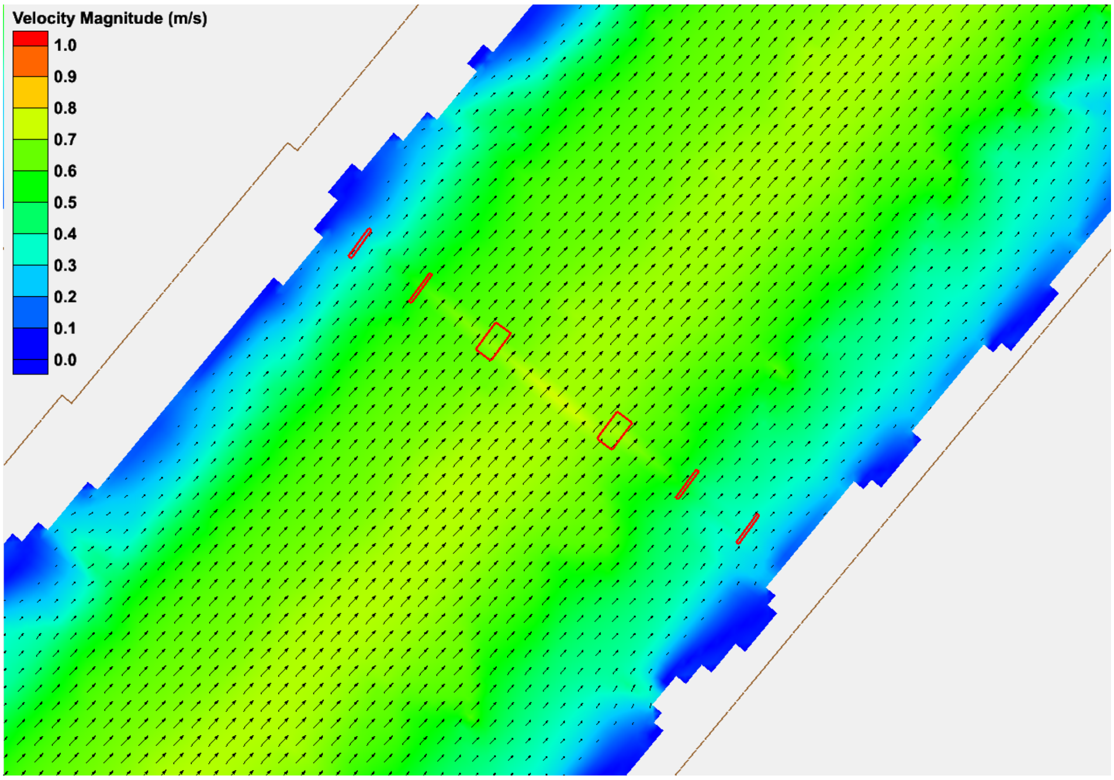

Figure 11 Velocity Magnitude and Direction, 20yr Inflow; Scen3

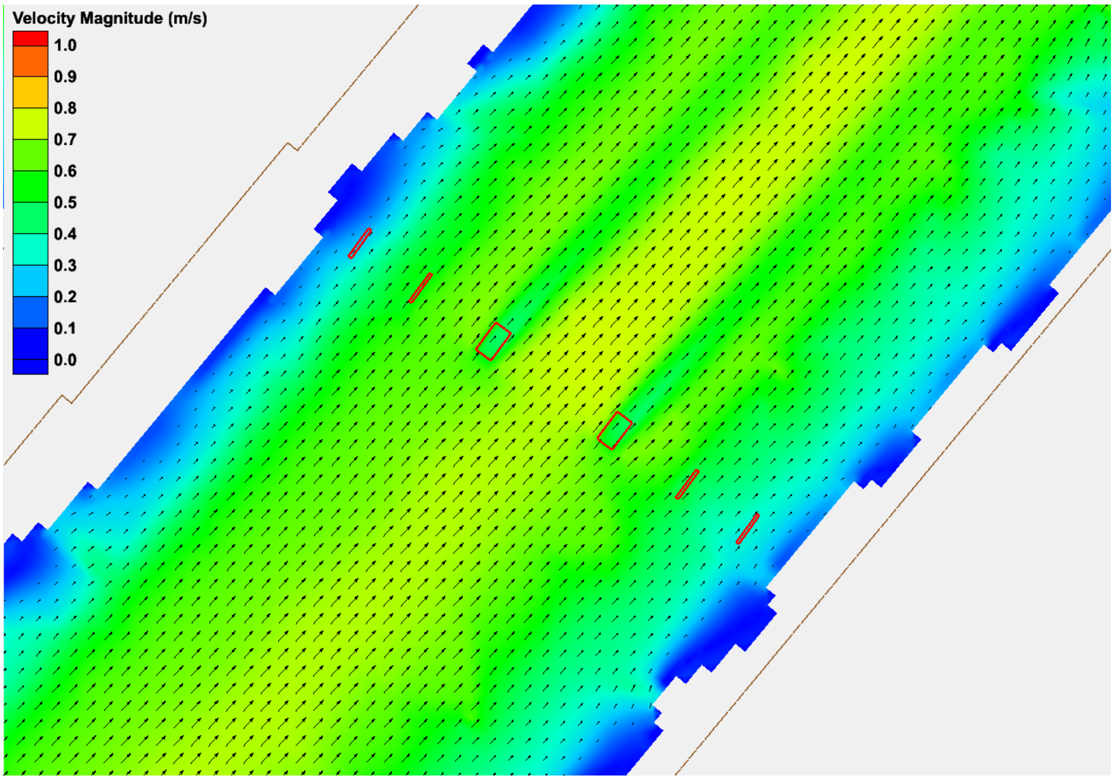

Figure 12 Velocity Magnitude and Direction, 20yr Inflow; Scen4

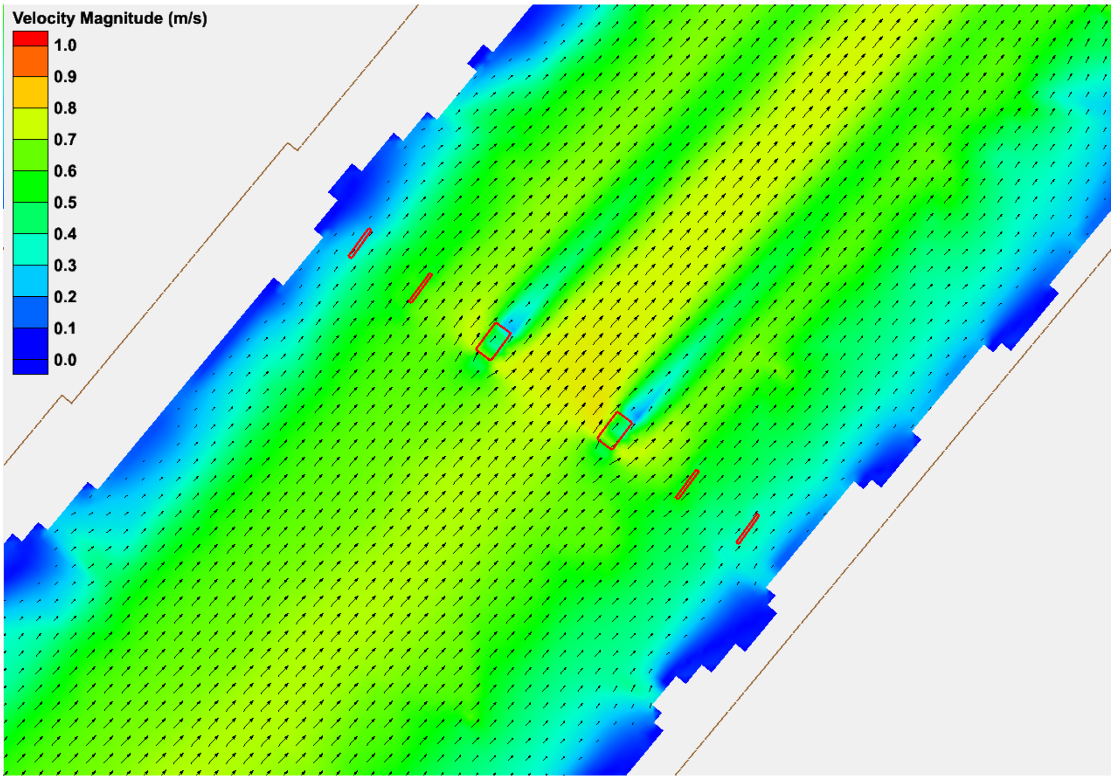

Figure 13 Velocity Magnitude and Direction, 20yr Inflow; Scen5

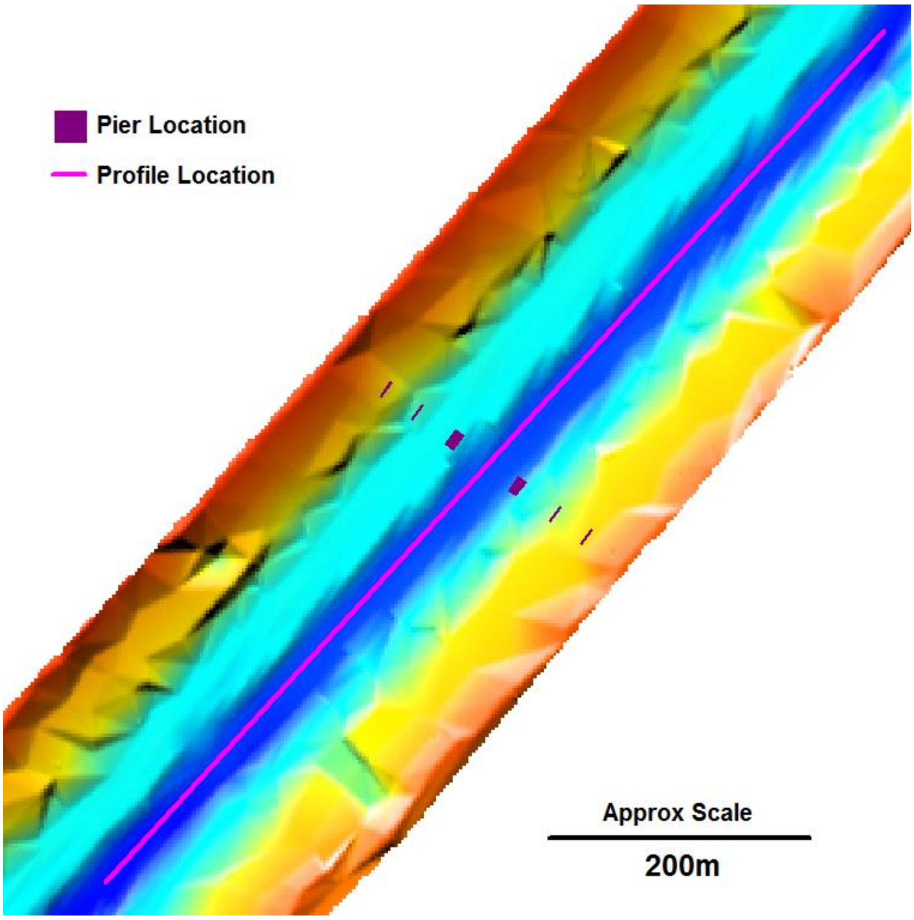

Figure 14 Profile Extraction Location

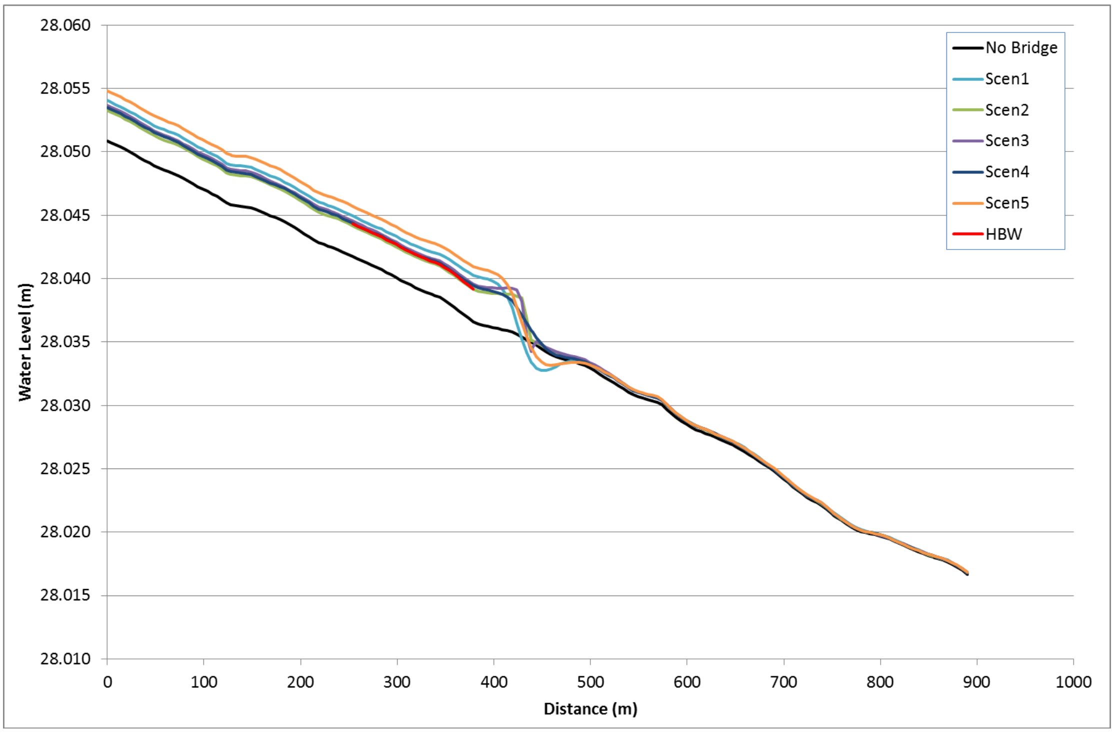

Figure 15 Water Surface Profiles; Bridge Modelling Methods

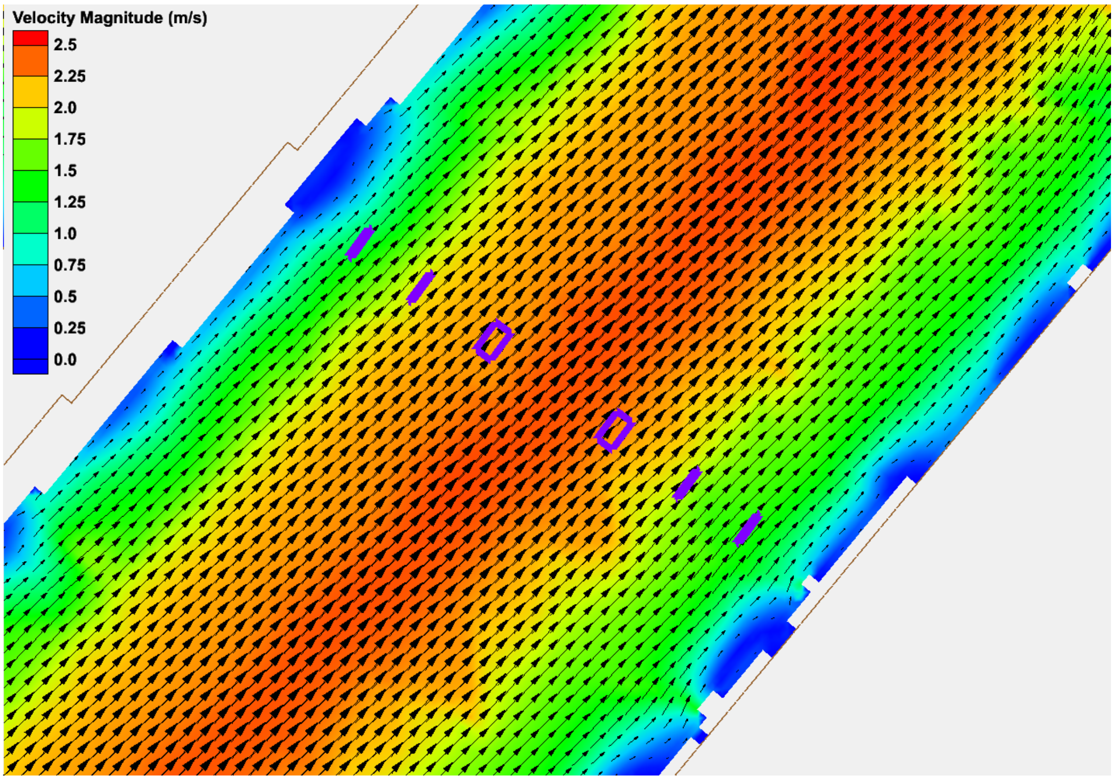

Figure 16 Velocity Magnitude and Direction, 5 x 20yr Inflow; No Bridge

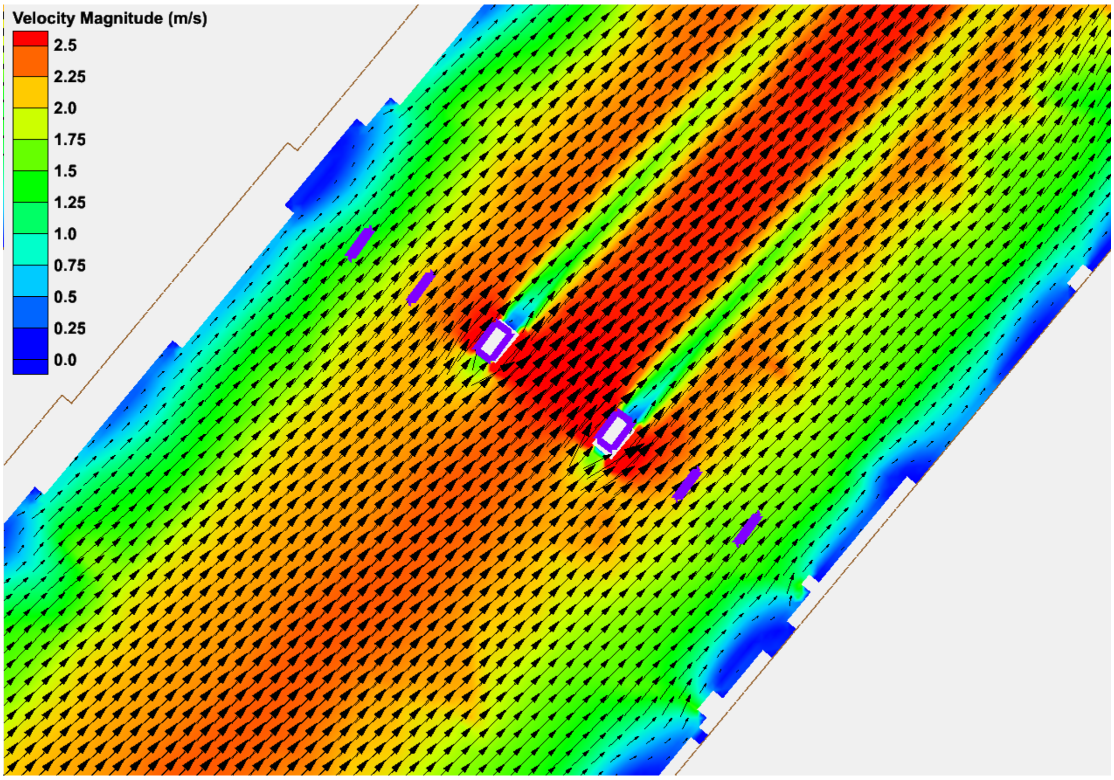

Figure 17 Velocity Magnitude and Direction, 5 x 20yr Inflow; Scen1

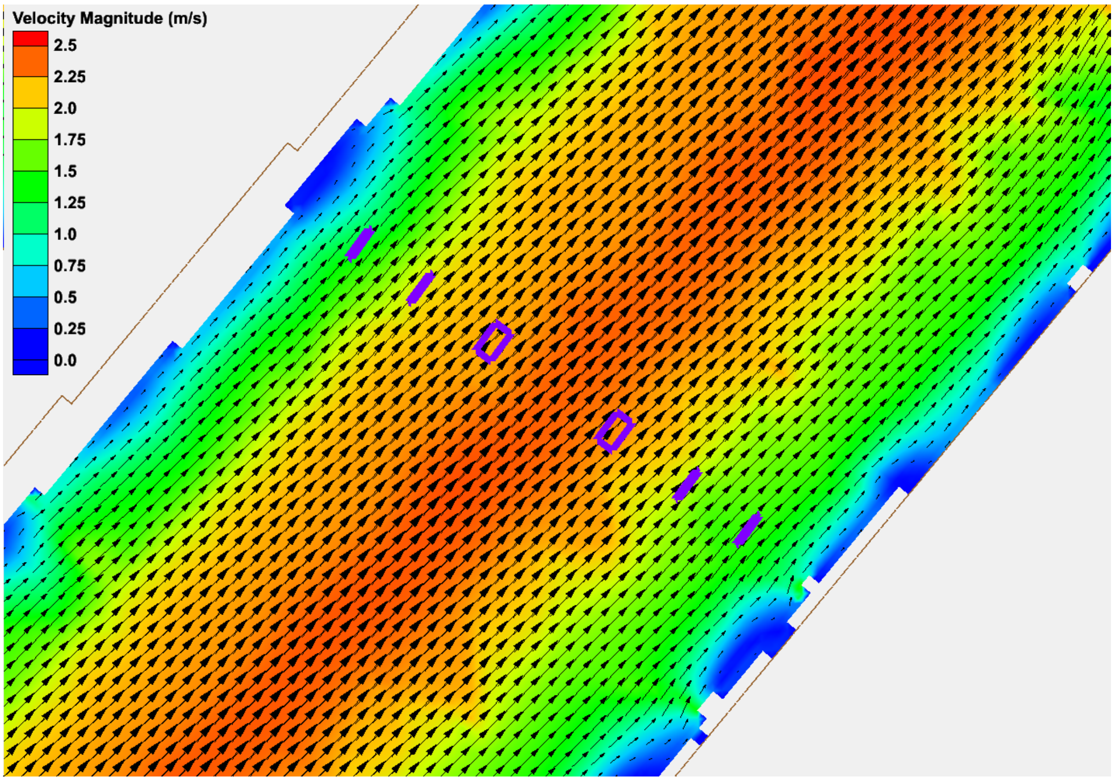

Figure 18 Velocity Magnitude and Direction, 5 x 20yr Inflow; Scen2

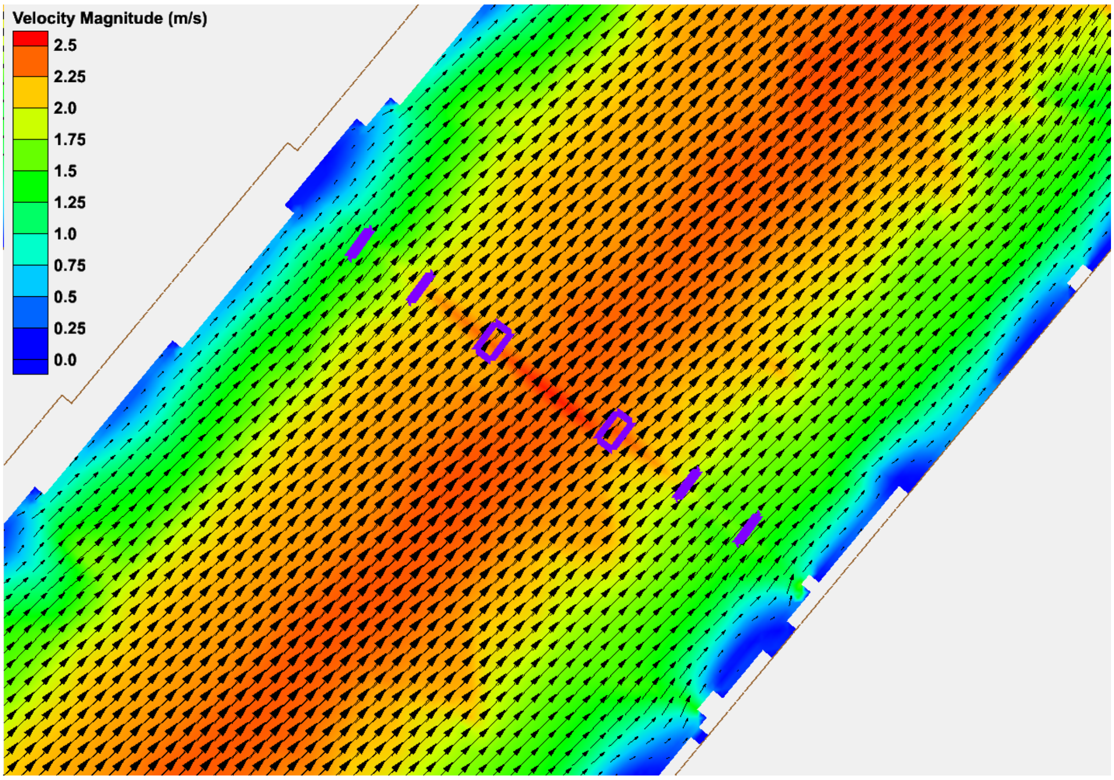

Figure 19 Velocity Magnitude and Direction, 5 x 20yr Inflow; Scen3

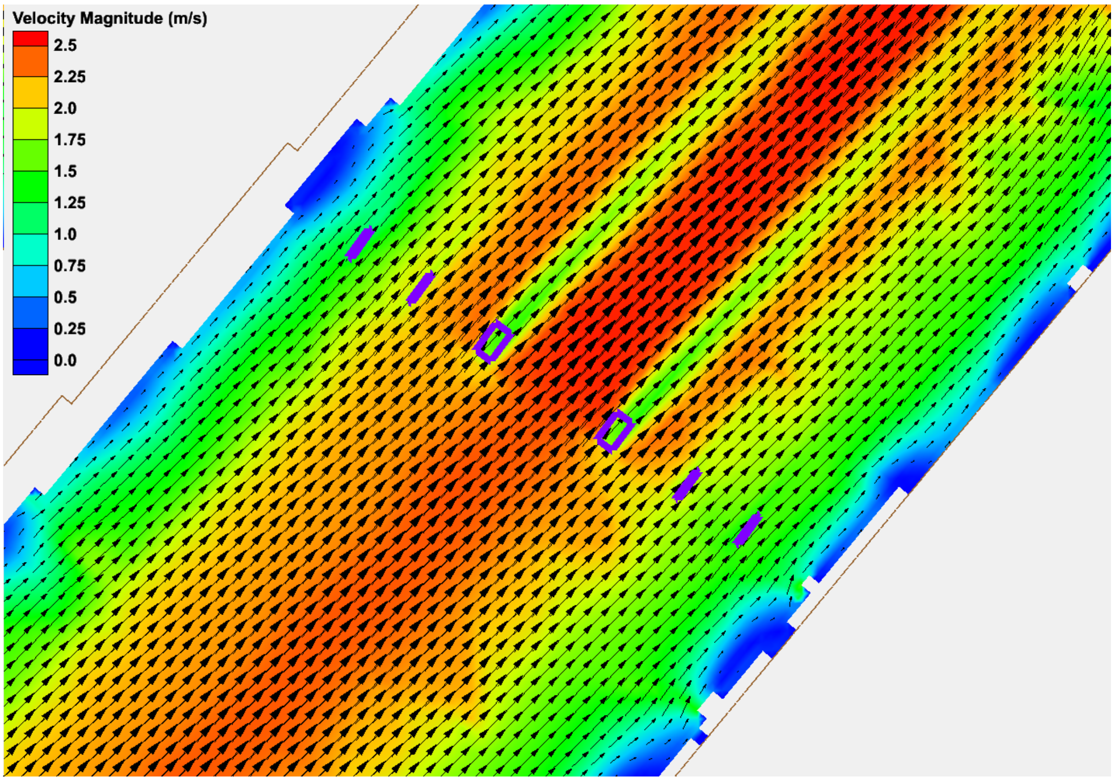

Figure 20 Velocity Magnitude and Direction, 5 x 20yr Inflow; Scen4

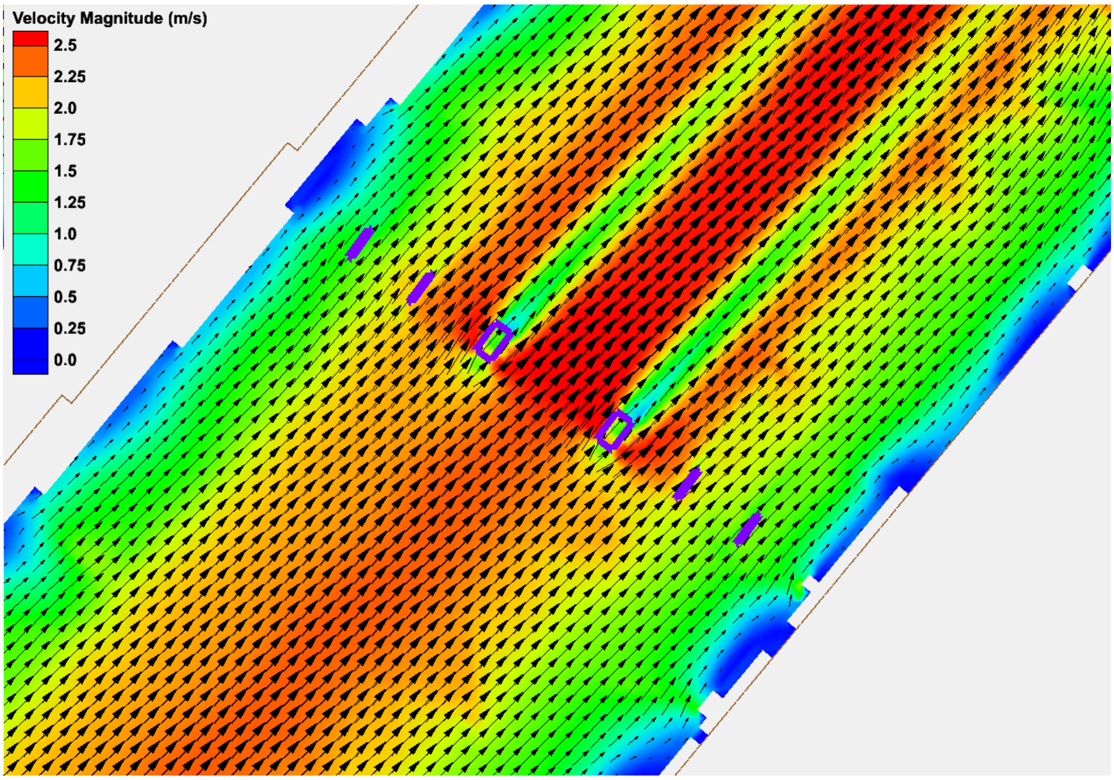

Figure 21 Velocity Magnitude and Direction, 5 x 20yr Inflow; Scen5

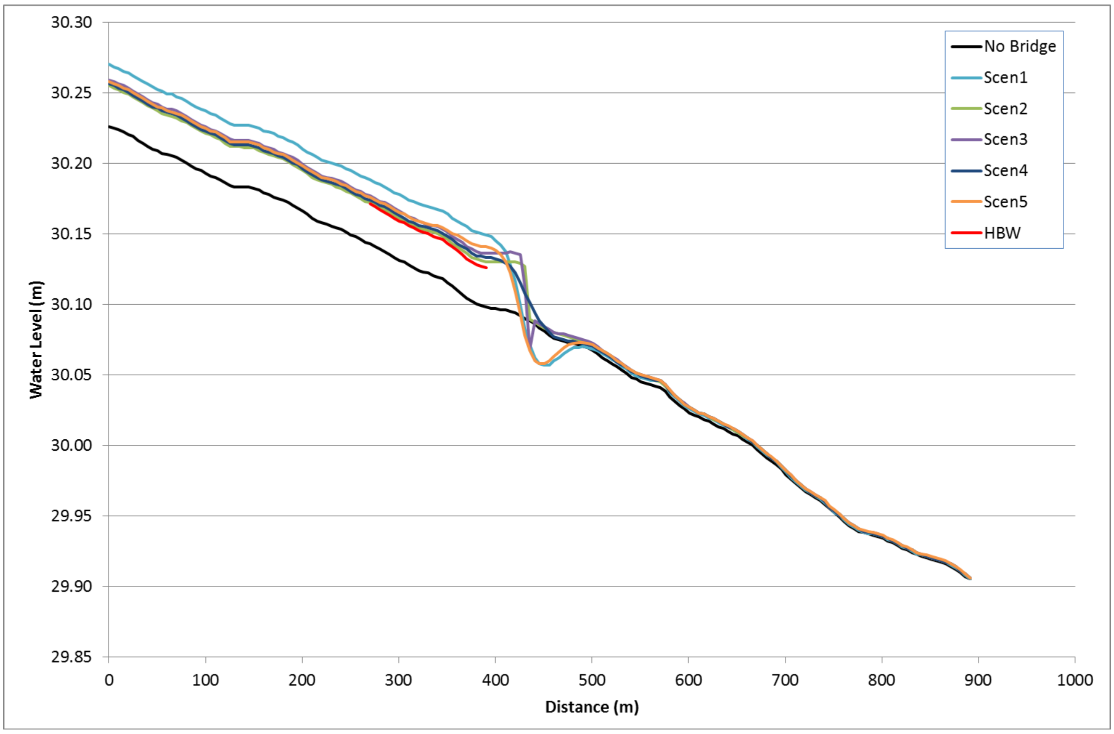

Figure 22 Water Surface Profiles, 5 x 20yr Inflows

TUFLOW
Flood and Tide Simulation Software