Modelling Energy Losses at Structures

Bill Syme

Modelling Hydraulic Structures
Today's Focus

Today: Cross-Drainage Structures

- Bridges and Culverts

Pipe Networks
To be covered in webinar Feb 16, 2022

- Inlets, Pipes, Manholes

Operational Structures
To be covered in webinar Apr 13, 2022

- Gated structures, Pumps, etc

TUFLOW

Modelling Hydraulic Structures
Agenda

A bit of theory

- What are energy or form losses?

Modelling Approaches

- 1D, 2D, 3D

Benchmarking

- Constrictions, Piers and Decks

First presented on this topic in 2001!

- Syme, W.J. (2001) *Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme* The Institution of Engineers, Australia Conference on Hydraulics in Civil Engineering, Hobart, November 2001
- https://www.tuflow.com/media/4984/2001-modelling-of-bends-and-hydraulic-structures-in-a-2d-scheme-syme.pdf

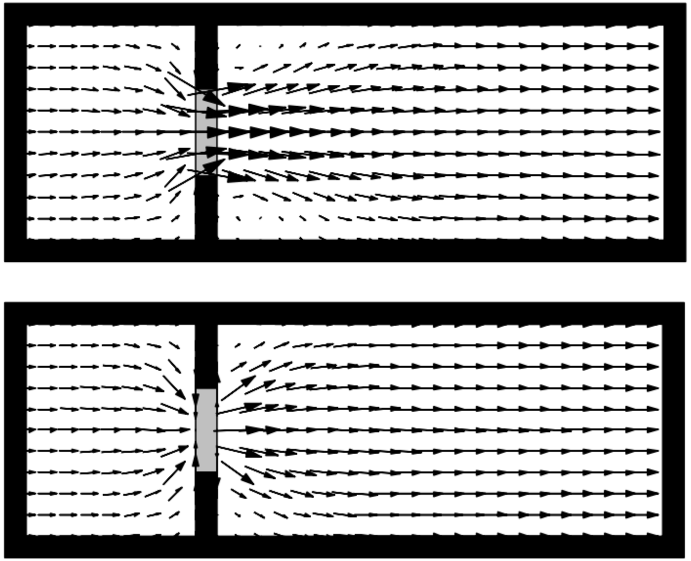


Figure 5 - Flow Patterns for 2D Box Culvert Models
(Culvert as 2D Cells on Top and Culvert as 1D Element at Bottom)

Modelling Hydraulic Structures

What are Form Losses?

What are Form Losses?
Energy Dissipation

Energy of flowing water is dissipated as heat primarily due to

- Bed resistance
  (e.g. Manning's equation)
- Changes in flow formation
  (wherever there is change in velocity magnitude and direction)

Changes in flow formation pronounced at

- Bends
- Constrictions
  (e.g. cross-drainage structures)
- Referred to as **Form Losses**

What are Form Losses?
Understanding the Energy of Water

Total Energy = h + $V^2/2g$

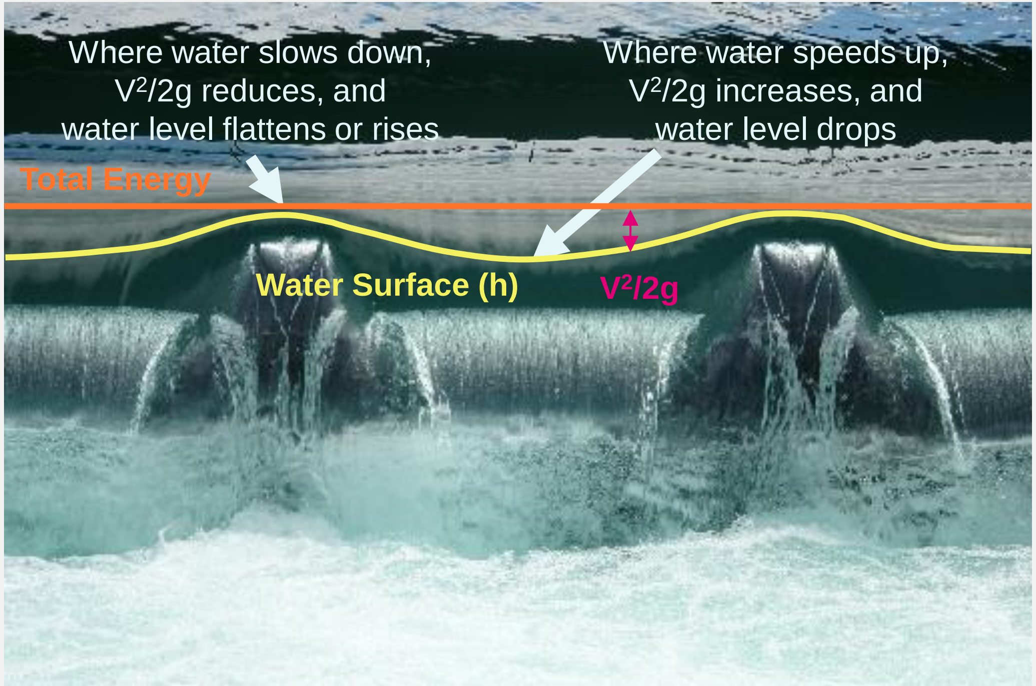
- = water level + kinetic energy

Form Losses

- Loss of kinetic energy, $V^2/2g$
- V = 1 m/s; kinetic energy = 0.05 m
- V = 4 m/s; kinetic energy = 0.82 m

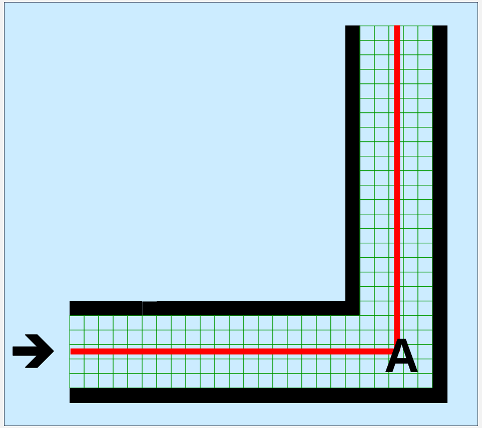
Form loss coefficient (K)

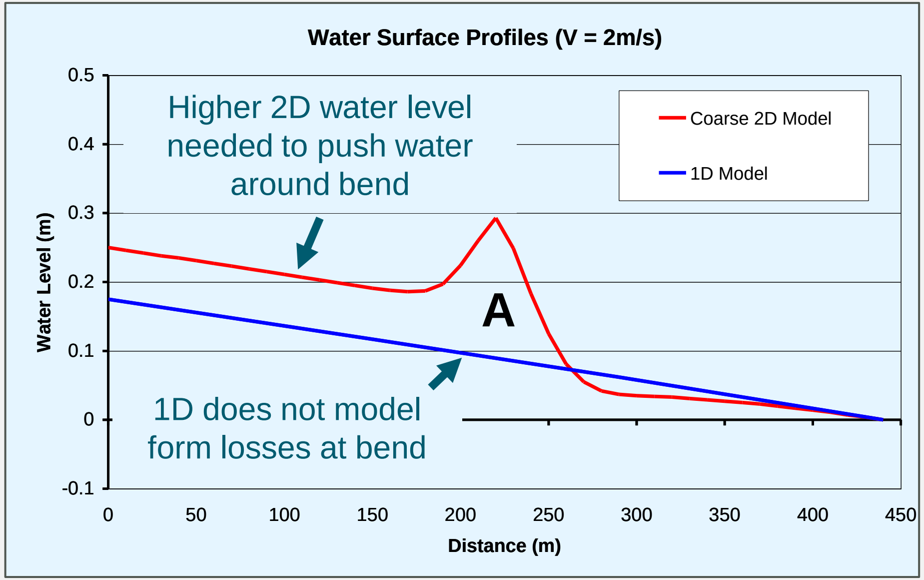
- Proportion of kinetic energy ($V^2/2g$) lost

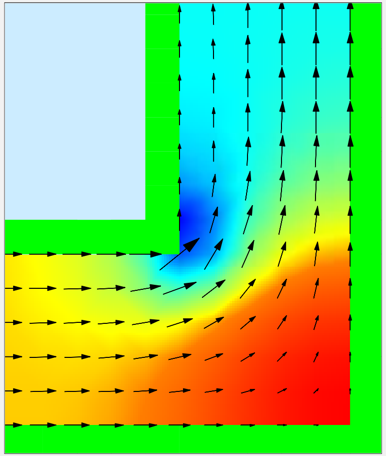


TUFLOW

What are Form Losses?
Right-Angled Bend Example – 1D versus 2D







What are Form Losses?
River Bend Form Losses

1D Equations

- Don’t simulate form losses
- Need to apply a form (bend) loss

2D Equations

- Simulates form losses
- But don’t simulate all form losses such as those in the vertical
  (eg. helicoidal circulations)

3D Equations

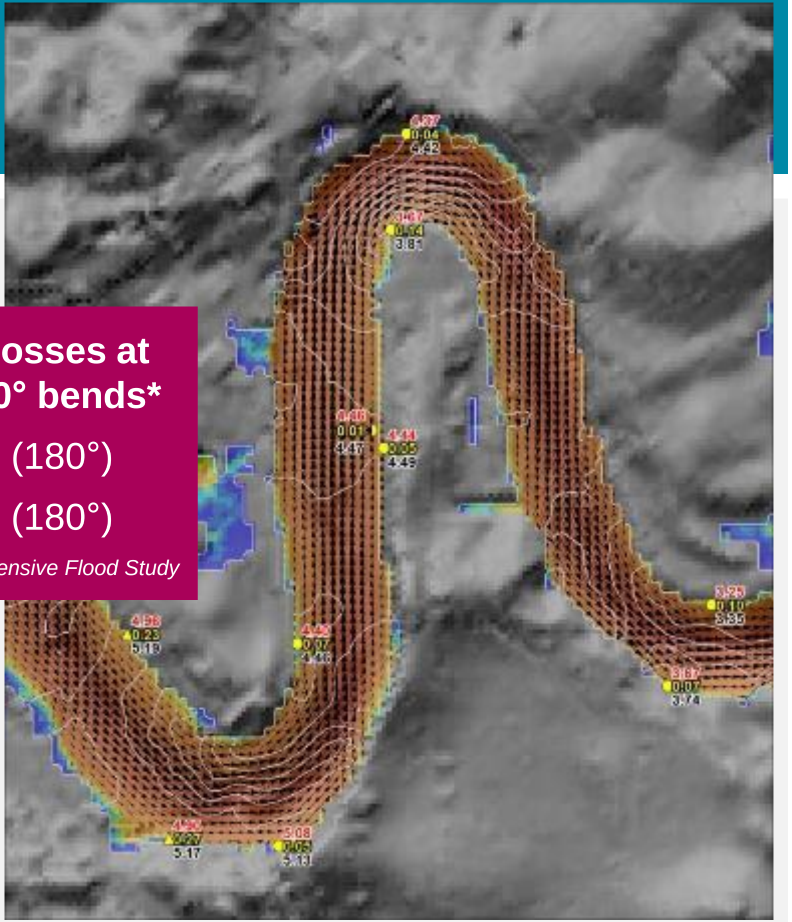
- Layered 3D should be closer again, but there are assumptions
- CFD using the Navier-Stokes equations should be closest

Can’t use the same form loss coefficients between 1D, 2D and 3D

Calibration of form losses at numerous 45° to 180° bends*

- 1D: 0.5 (45°) to 1.5 (180°)
- 2D: 0.1 (45°) to 0.4 (180°)

** Brisbane River Catchment Comprehensive Flood Study*



TUFLOW

Modelling Hydraulic Structures

1D Approach

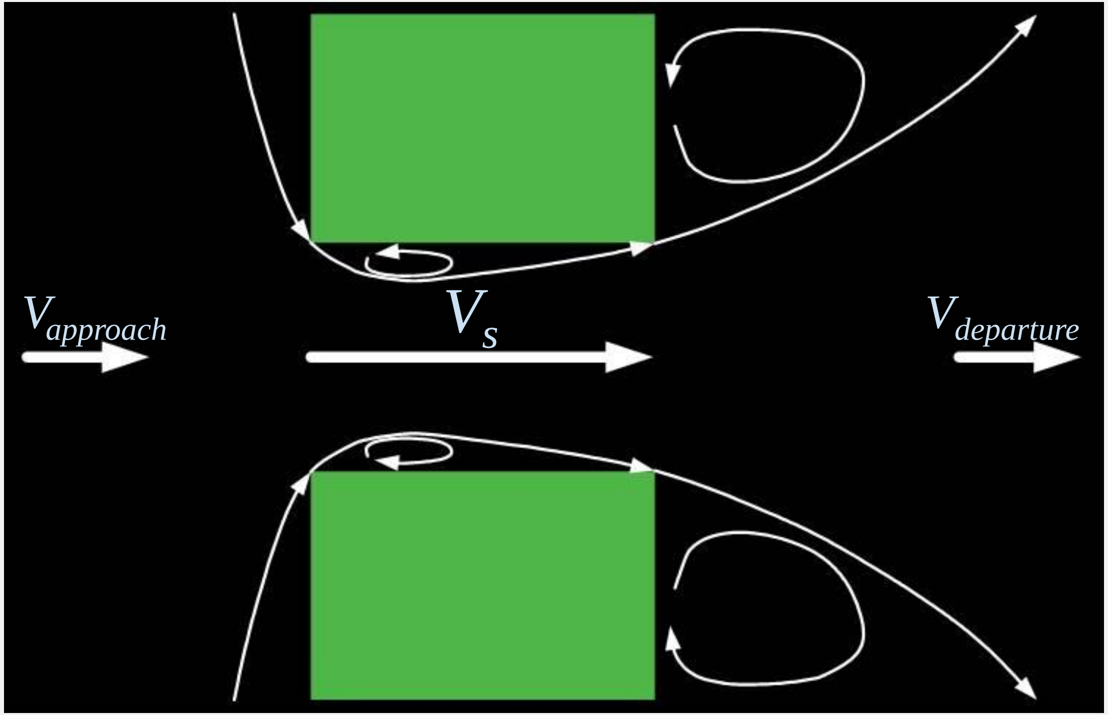
1D Approach
Entrance/Exit Loss Coefficients

Velocities are uni-directional

1D cannot implicitly simulate form losses

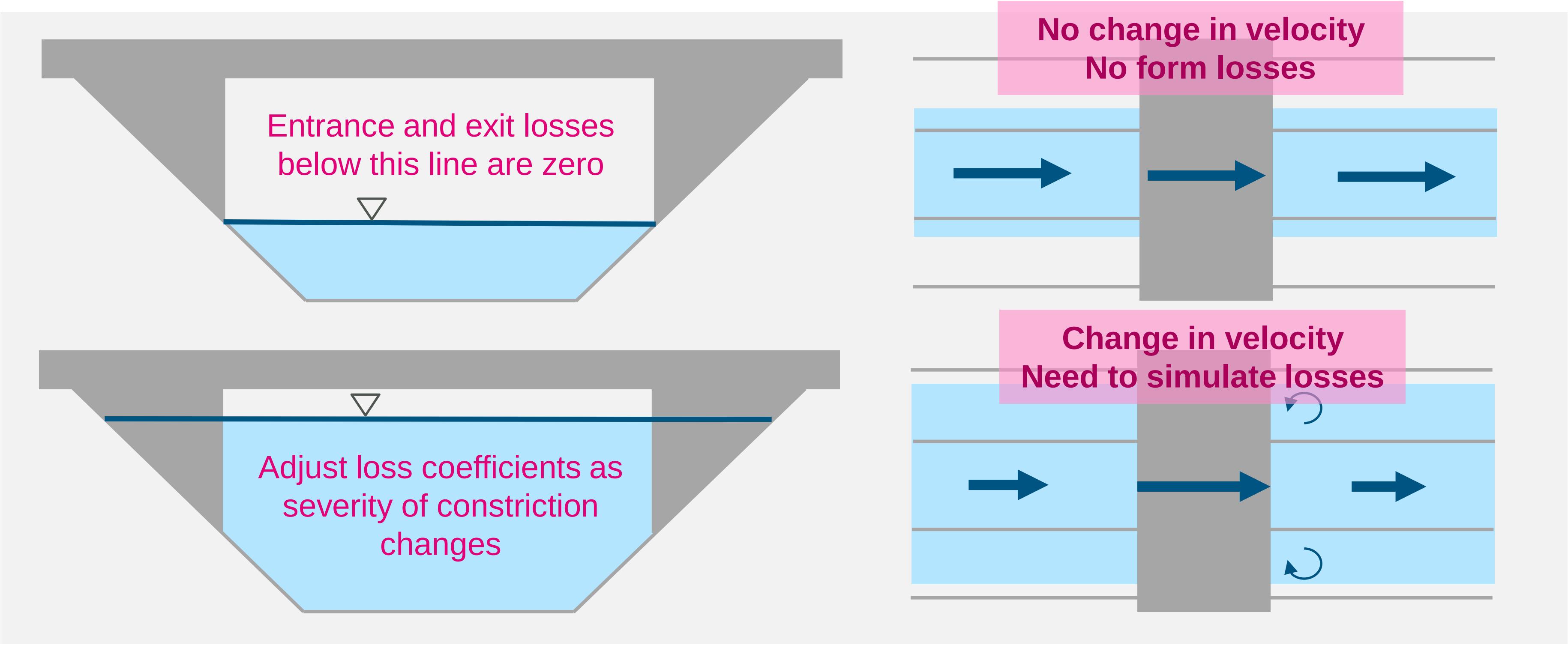
Need to explicitly specify energy lost using form loss coefficients

$$\Delta h = (K_{en} + K_{ex})\frac{V_s^2}{2g}$$



TUFLOW

1D Approach
Need to Adjust Loss Coefficients with Height



1D Approach
Need to Adjust Loss Coefficients with Height

$$K_{en\_adj} = K_{en}\left[1 - \frac{V_{app}}{V_s}\right]$$ - **based on testing**

$$K_{ex\_adj} = K_{ex}\left[1 - \frac{V_{dep}}{V_s}\right]^2$$ - **derived from theory**

- Published values for K_{en} and K_{ex} typically for $V_{app} = 0$ and $V_{dep} = 0$
- K_{en} = 0.2 to 0.7 – depends on entrance configuration – use 0.5 if in doubt
- K_{ex} = 1.0 but in some situations maybe less depending on how outlet is modelled

Energy loss is

$$\Delta h = (K_{en} + K_{ex})\frac{V_s^2}{2g}$$

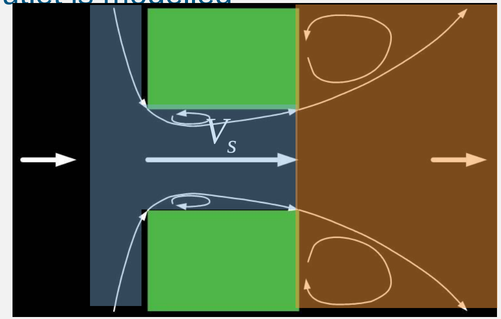
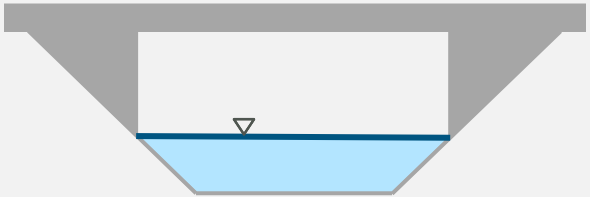


Table 7.16.5 – Entrance (energy) loss coefficients [1]

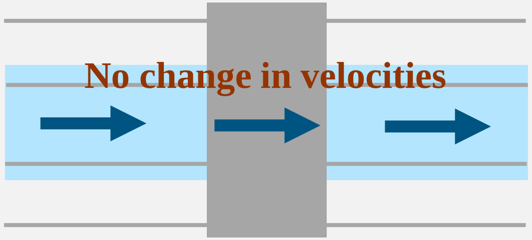
Type of structure and design of entrance	Coefficient K_e
Concrete pipe:	
Projecting from fill, socket end (groove end)	0.2
Projecting from fill, square cut end	0.5
Headwall or headwall and wing walls:	
• socket end of pipe (groove end)	0.2
• square edge	0.5
• rounded (radius = D/12)	0.2
• mitred to conform to fill slope	0.7
• end section conforming to fill slope.	0.5
Hooded inlet projecting from headwall	Note [2]
Corrugated metal pipe:	
Projecting from fill (no headwall)	0.9
Headwall or headwall and wing walls square edge	0.5
Mitred to conform to fill slope	0.7
End section conforming to fill slope	0.5
Reinforced concrete box:	
Headwall parallel to embankment (no wing walls):	
• square edged on 3 edges	0.5
• rounded on 3 edges to radius of 1/12 barrel dimension.	0.2
Wing walls at 30° to 70° to barrel:	
• square edged at crown	0.4
• crown edge rounded to radius 1/12 barrel dimension.	0.2
Wing walls at 10° to 25° to barrel:	
• square edged at crown.	0.5
Wing walls parallel (extension of sides):	
• square edged at crown.	0.7

QUDM, 2017

1D Approach
Need to Adjust Loss Coefficients with Height

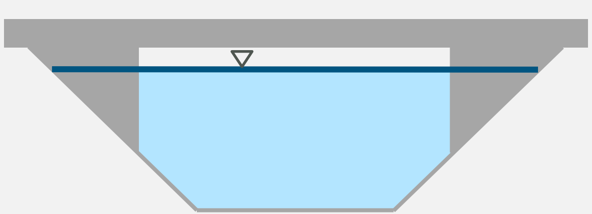


Entrance and exit losses below this line are zero

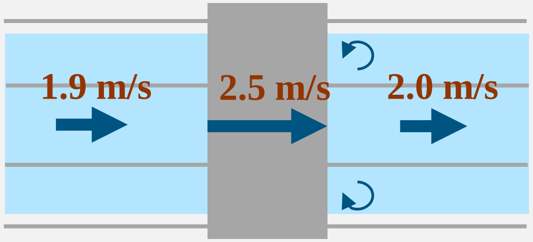


No change in velocity
No form losses

$$K_{en\_adj} = 0.0$$
$$K_{ex\_adj} = 0.0$$
$$\Delta h = 0.0$$



Adjust loss coefficients as severity of constriction changes



$V_{structure} > V_{channel}$

Adjust coefficients to correctly simulate form losses

$$K_{en\_adj} = 0.5 * \left[1 - \frac{1.9}{2.5}\right] = \mathbf{0.12}$$
$$K_{ex\_adj} = 1.0 * \left[1 - \frac{2.0}{2.5}\right]^2 = \mathbf{0.04}$$
$$\Delta h = (0.12 + 0.04) * \frac{2.5^2}{2g} = 0.05\ m$$

Modelling Hydraulic Structures

2D Approaches

2D Approach
Looks Impressive

But is it accurate?

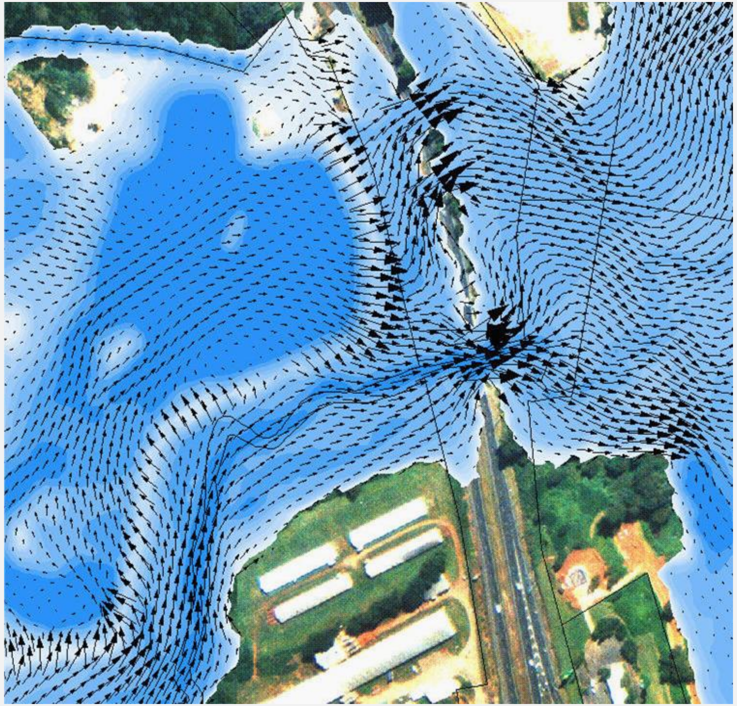
Q: Does it implicitly model form losses?

A: It does, but it can't model all of them

- 2D does not account for
  - Sub-cell form losses (e.g. piers, vena-contracta)
  - 3D form losses (e.g. helicoidal flows, vertical movement)

Q: How do we account for missing form losses?

A: Very good question!



TUFLOW

2D-2D-2D Approach
How to account for missing form losses?

Not applicable to apply 1D form losses

- Otherwise duplicates losses

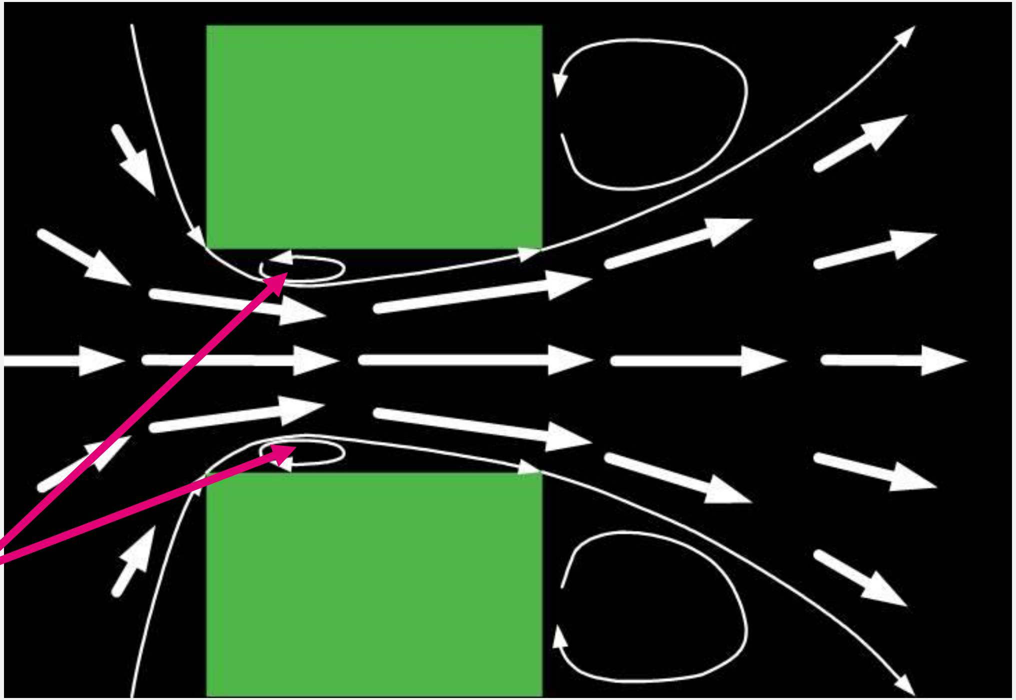
Options

- Add additional (small) form losses
- Use a finer mesh (often not practical though)

Other considerations

- Losses sub-cell need a good turbulence model

e.g. vena-contracta form
losses need a very fine mesh



2D-1D-2D Approach
Can we apply 1D form losses, K_{en} and K_{ex}?

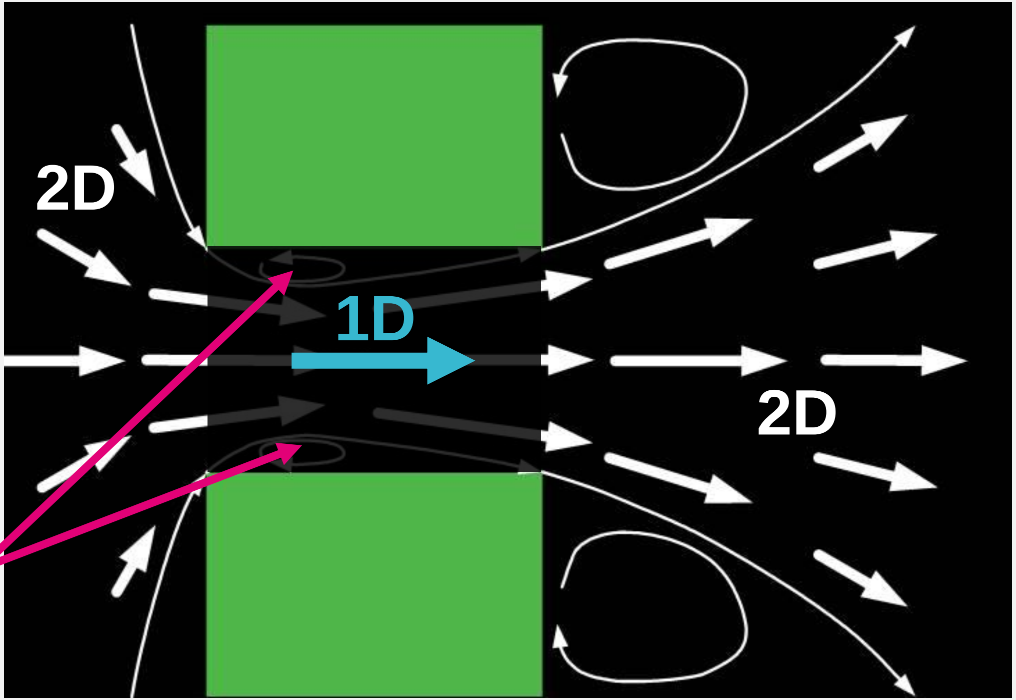
NO!

- 2D entrance losses will now mostly occur in 1D element
- But, partial or all exit losses still occur in 2D cells

Therefore, not correct to apply all 1D form losses

  - Otherwise duplicates losses

Vena-contracta form losses now occur in 1D element



TUFLOW

Modelling Hydraulic Structures

3D Approaches

3D Approaches
Layered 2D

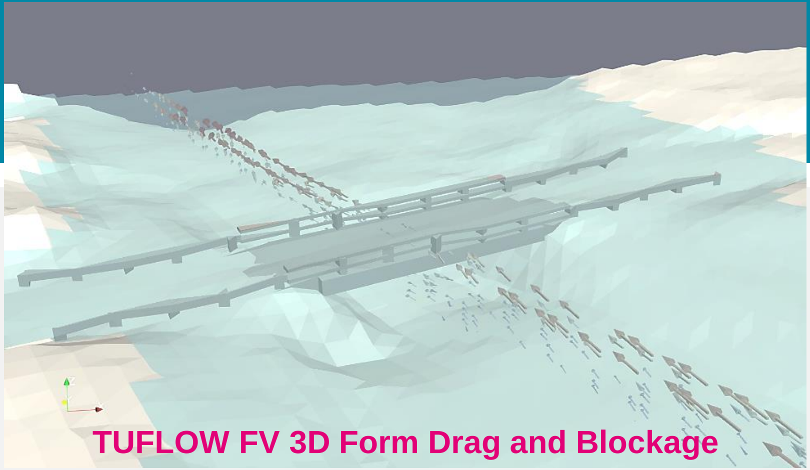
Horizontal mixing of mass and momentum

- Sigma, Z or Z-Sigma hybrid vertical geometry
- Eddy viscosity models used for sub-grid scale mixing (Wu, Smagorinsky, etc)

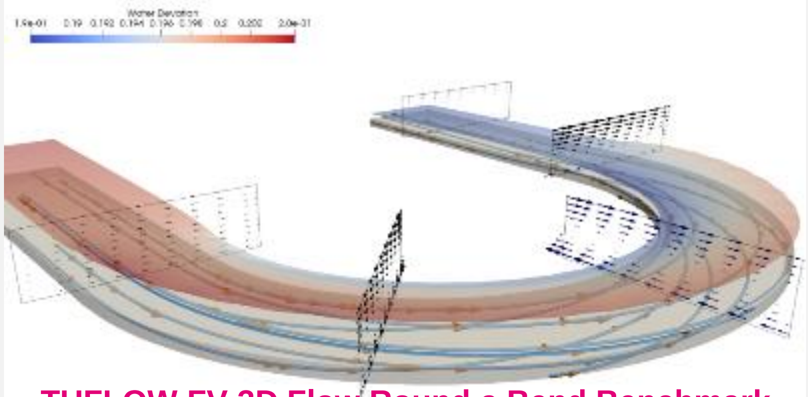
Vertical mixing of mass and momentum

- Mixed vertically due to shear between layers and associated turbulence
- Eddy viscosity models used for sub-grid scale mixing (e.g. k-epsilon, k-omega)
- Can be influenced by stratification/buoyancy
- Hydrostatic assumption
  - Vertical acceleration of fluid motion is assumed to be negligible

The next phase in hydraulic structure modelling?



TUFLOW FV 3D Form Drag and Blockage



TUFLOW FV 3D Flow Round a Bend Benchmark

3D Approaches
CFD (Computational Fluid Dynamics)

CFD solvers

- Compute full 3D fields for pressure, velocity and turbulence
- No hydrostatic assumption
- Full Navier-Stokes fluid equations

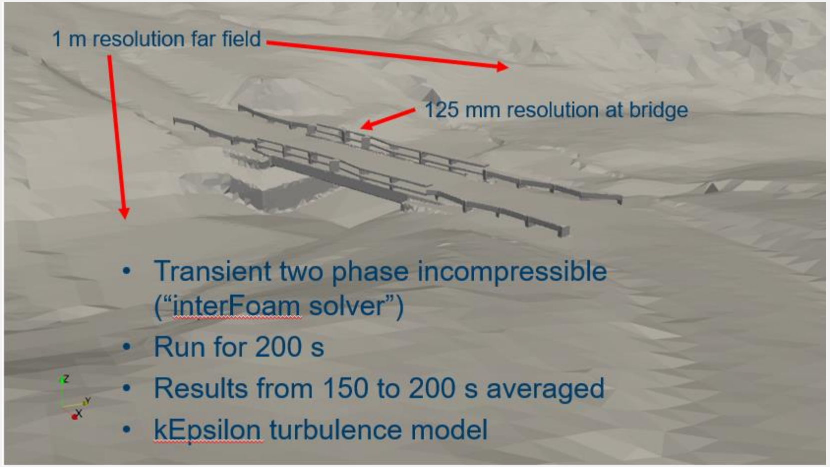
Mesh

- Very high resolution – represents structure in detail
- No sub-modelling of blockage factors or loss coefficients

Eddy viscosity model critical
(for non-laminar flows)

Very long simulation times

- Not yet practical for most real-world flood study investigations



3D Approaches
Physical Modelling

There is (was) a saying:

"No one believes a computer model
(except for the modeller)

but everyone believes a physical model
(except for the modeller)"

TUFLOW

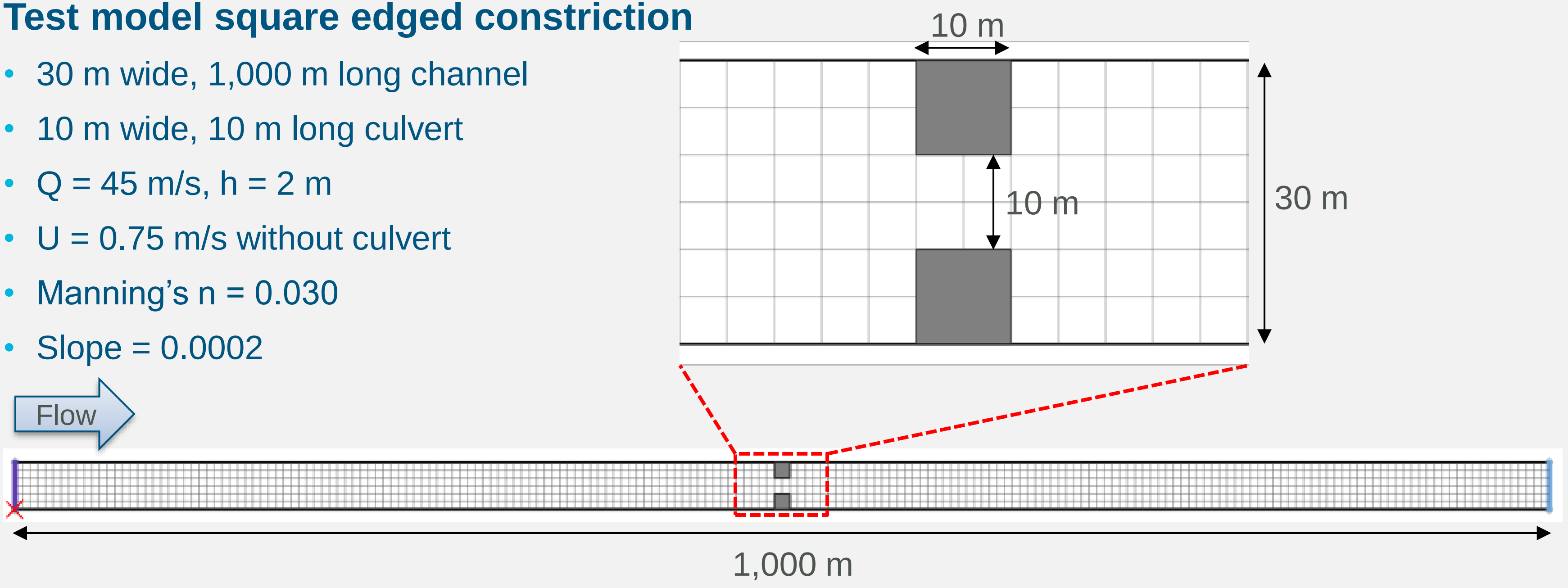
Modelling Hydraulic Structures

Constrictions

Constrictions Benchmark Test

Test model square edged constriction

- 30 m wide, 1,000 m long channel
- 10 m wide, 10 m long culvert
- Q = 45 m/s, h = 2 m
- U = 0.75 m/s without culvert
- Manning's n = 0.030
- Slope = 0.0002



Constrictions – Benchmark Test
1D-1D-1D Model

$$K_{en\_adj} = K_{en}\left[1 - \frac{V_{app}}{V_s}\right]$$ - based on testing

$$K_{ex\_adj} = K_{ex}\left[1 - \frac{V_{dep}}{V_s}\right]^2$$ - derived from theory

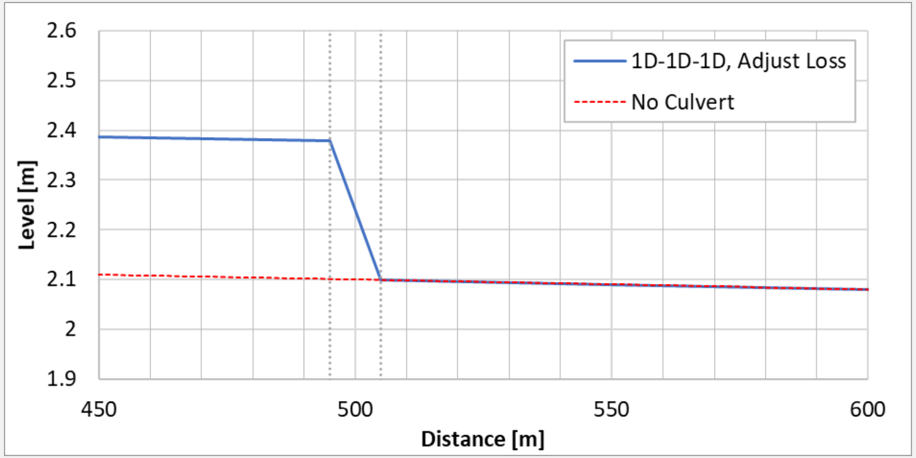
1D-1D-1D Model

- Afflux 1D solution = 276 mm

Desktop calculation of losses

- $K_{en\_adj} = 0.5\left[1 - \frac{0.66}{2.4}\right] = \mathbf{0.36}$
- $K_{ex\_adj} = 1.0\left[1 - \frac{0.75}{2.4}\right]^2 = \mathbf{0.48}$
- $\Delta h_{form\ loss} = \frac{K_{en\_adj} + K_{ex\_adj}}{2g} V_s^2$ = 250 mm
- plus $\Delta h_{Extra\ bed\ friction}$ = 21 mm
- Afflux = 271 mm

Reinforced concrete box:	
Headwall parallel to embankment (no wing walls):	
• square edged on 3 edges	0.5
• rounded on 3 edges to radius of 1/12 barrel dimension.	0.2



TUFLOW

Constrictions – Benchmark Test
1D-1D-1D – No Adjustment of Losses

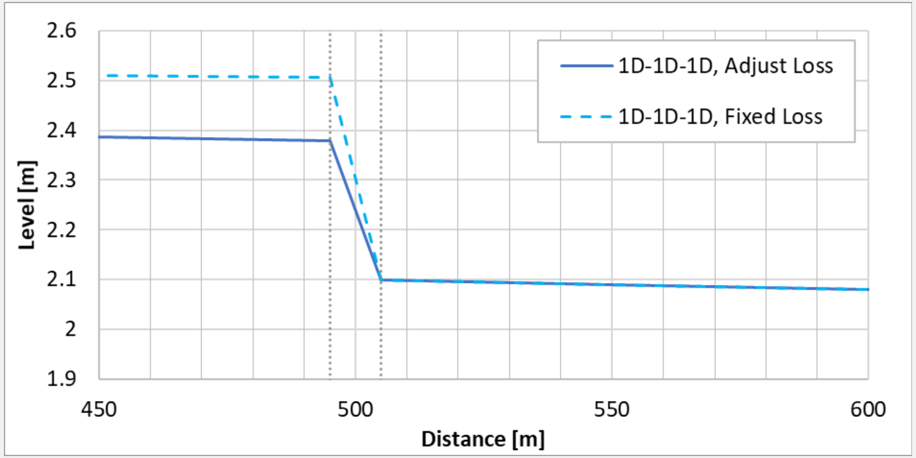
What happens if the structure losses are not adjusted?

- $K_{en} = 0.5$
- $K_{ex} = 1.0$

Afflux is over-predicted

- Represents the case of still water to still water (e.g. lake discharging to another lake)
- $V_{approach}$ and $V_{departure}$ ~ 0.0

Note: Published K values are usually based on this scenario

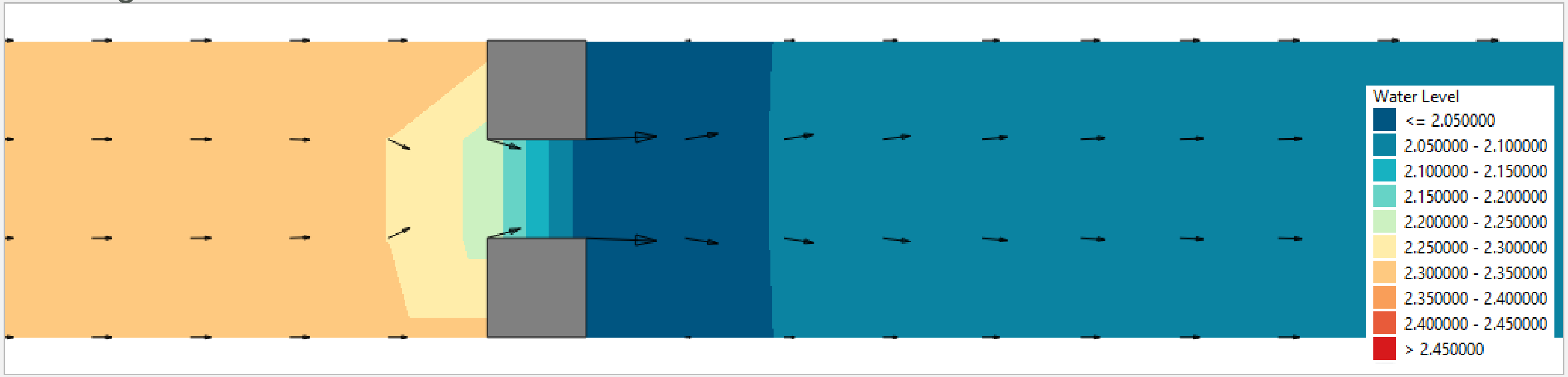


Constrictions – Benchmark Test
2D-2D-2D – Cell Size Results Convergence

Let's model in 2D only

- 10, 5, 2.5,1 and 0.5 m

10 m grid

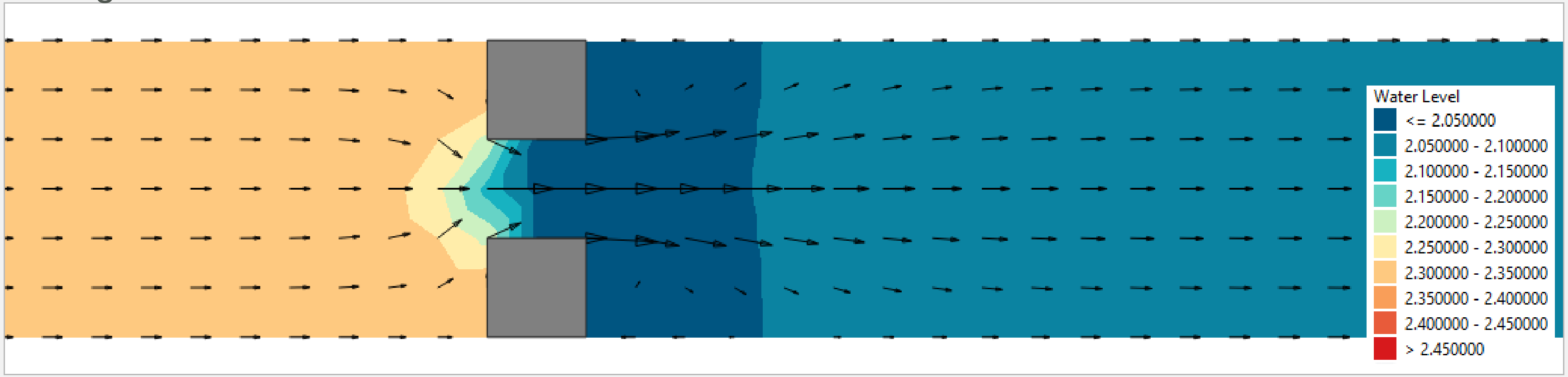


Constrictions – Benchmark Test
2D-2D-2D – Cell Size Results Convergence

Let's model in 2D only

- 10, 5, 2.5,1 and 0.5 m
- Afflux increases with smaller cell size

5 m grid

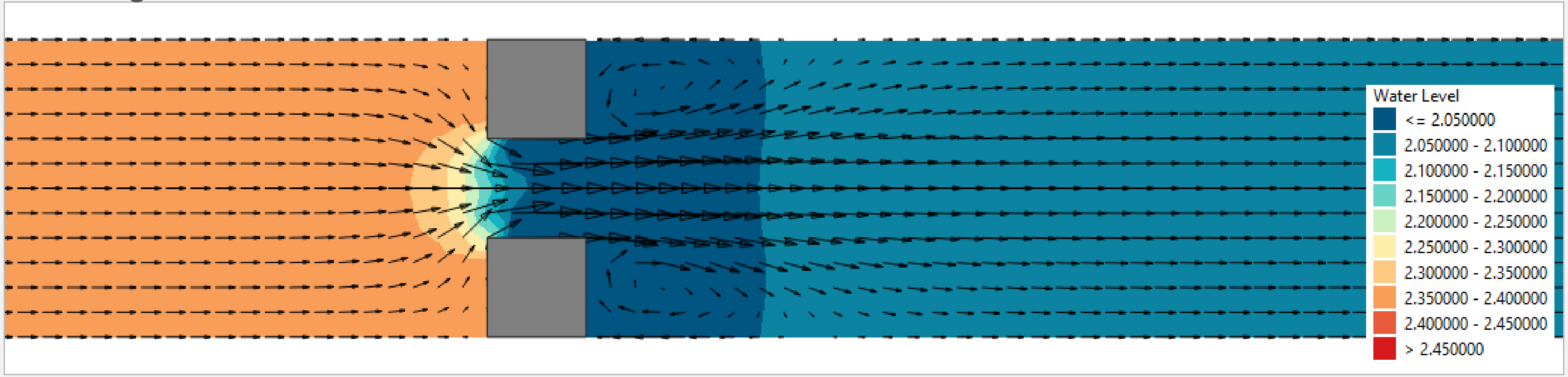


Constrictions – Benchmark Test
2D-2D-2D – Cell Size Results Convergence

Let’s model in 2D only

- 10, 5, 2.5,1 and 0.5 m
- Afflux increases with smaller cell size
  - Finer meshes better resolve form losses

2.5 m grid



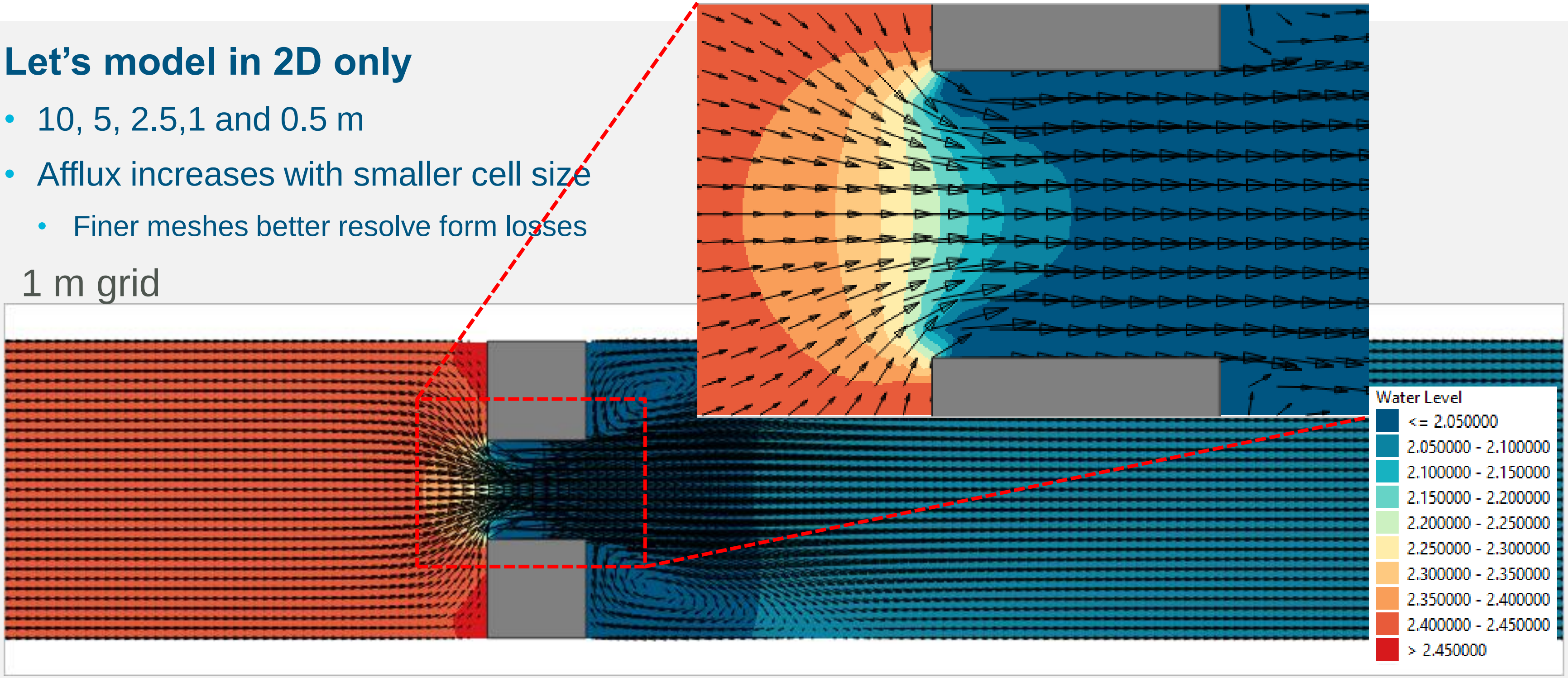
TUFLOW

Constrictions – Benchmark Test
2D-2D-2D – Cell Size Results Convergence

Let's model in 2D only

- 10, 5, 2.5,1 and 0.5 m
- Afflux increases with smaller cell size
  - Finer meshes better resolve form losses

1 m grid



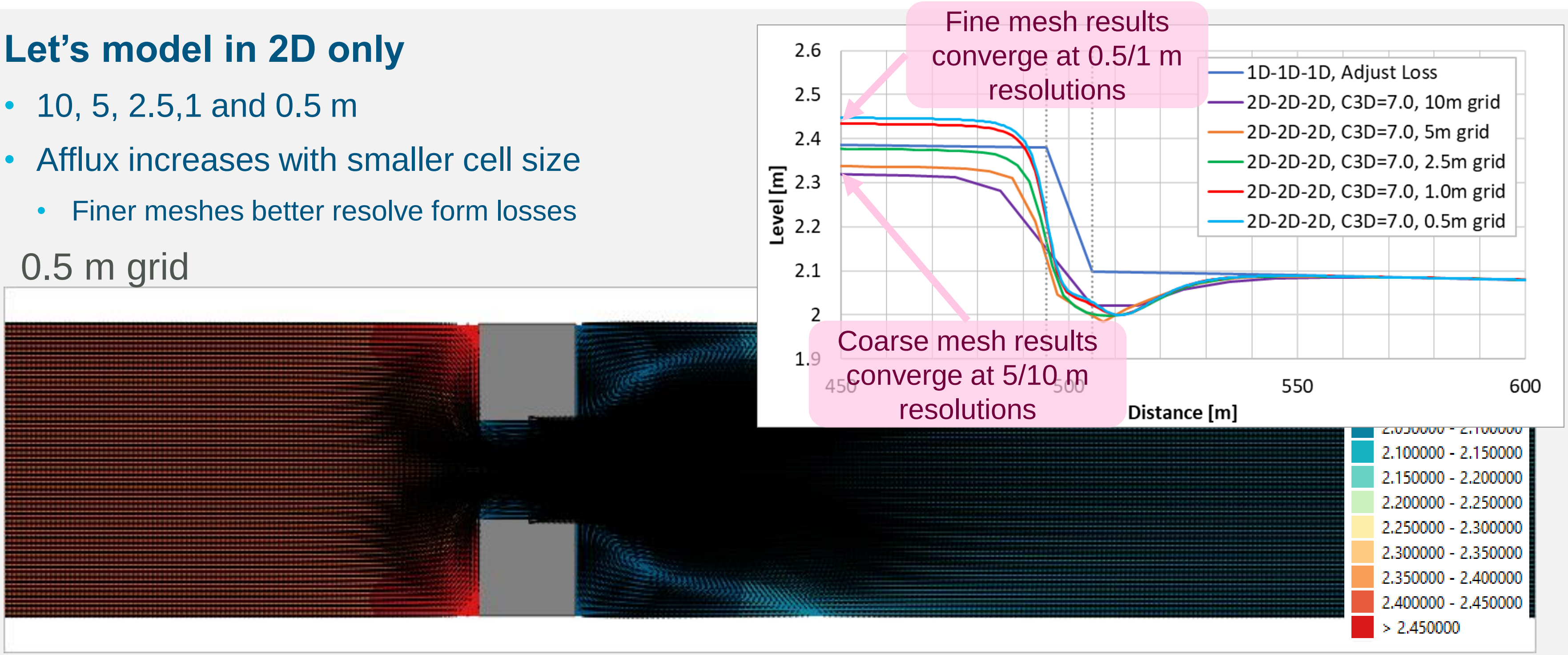
TUFLOW

Constrictions – Benchmark Test
2D-2D-2D – Cell Size Results Convergence

Let's model in 2D only

- 10, 5, 2.5,1 and 0.5 m
- Afflux increases with smaller cell size
  - Finer meshes better resolve form losses

0.5 m grid

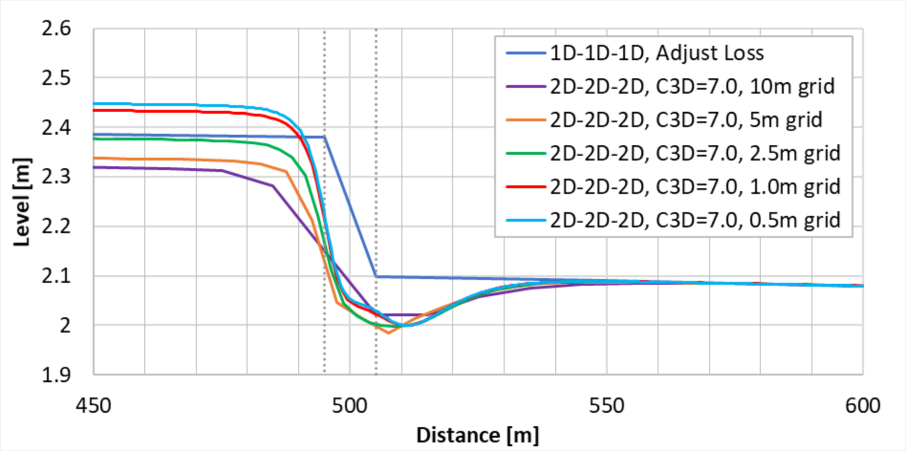


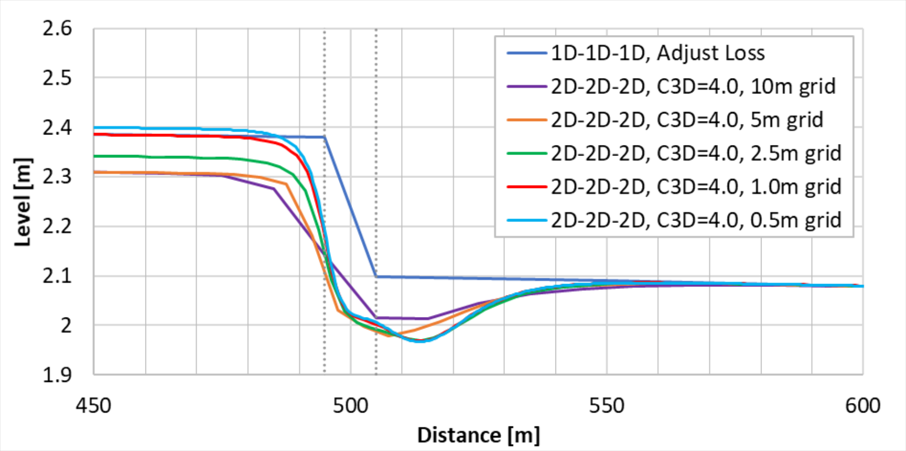
TUFLOW

Constrictions – Benchmark Test
2D-2D-2D – Calibration

Fine mesh

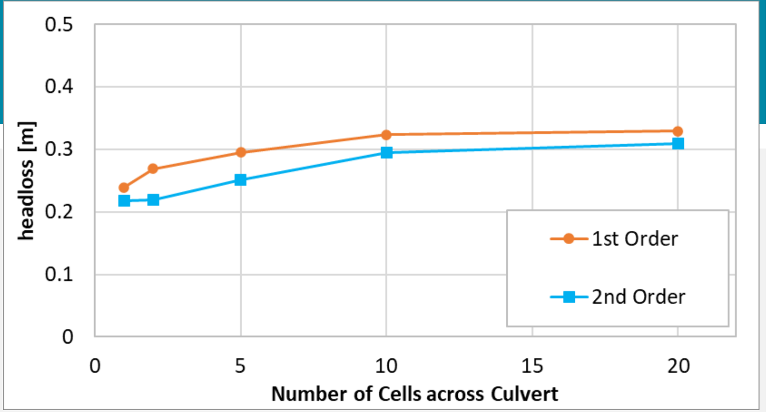
- Sub-grid turbulence representation critical
- Wu = 4 best match to theoretical result
- 0.2 m pressure / 0.1 m turbulence





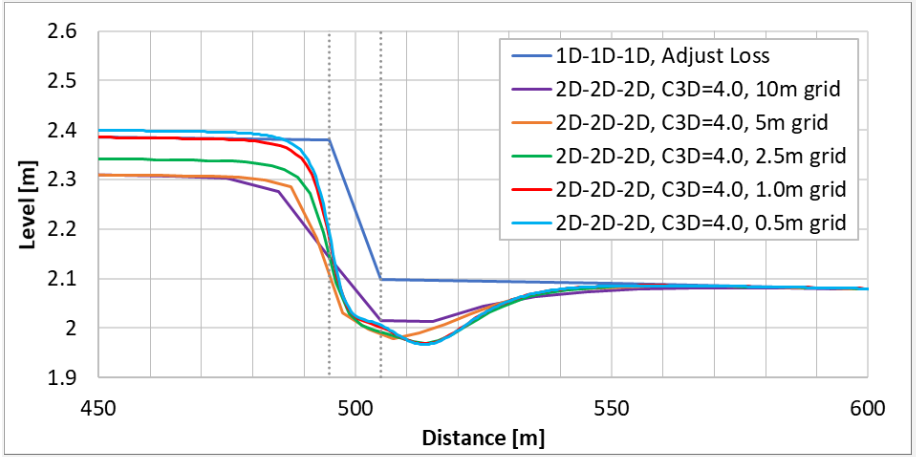
TUFLOW

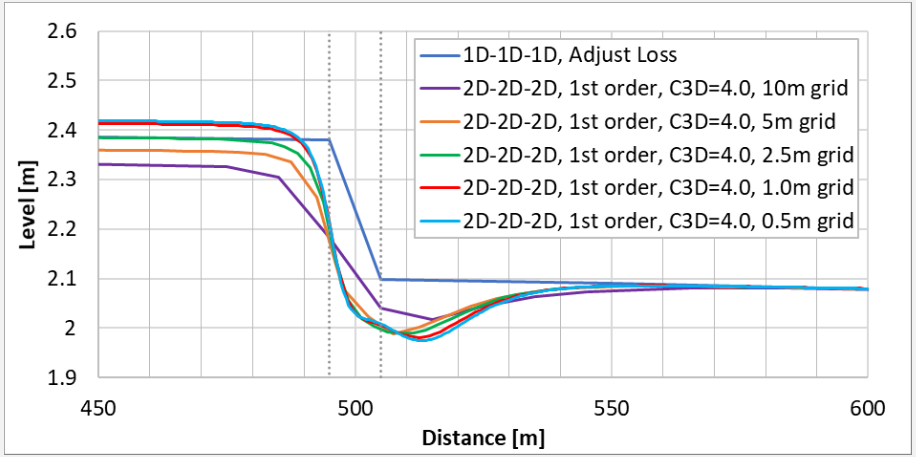
Constrictions – Benchmark Test 2D-2D-2D – 1st Order Check



Not all 2D solvers are the same!

- Repeat runs using TUFLOW HPC's 1st order option
- Numerically diffusive – causes artificial energy losses
- Reduced Manning's n or turbulence needed

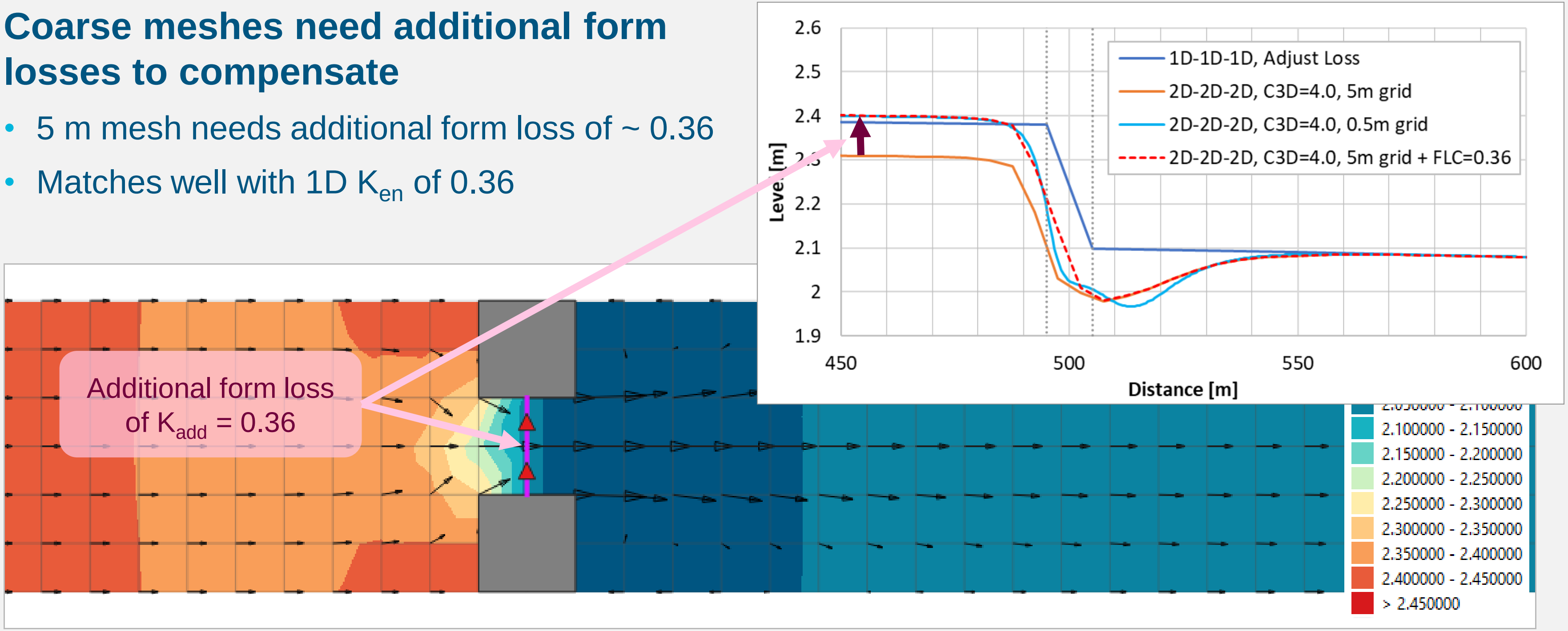




Constrictions – Benchmark Test
2D-2D-2D Coarse Mesh with Additional Form Loss

Coarse meshes need additional form losses to compensate

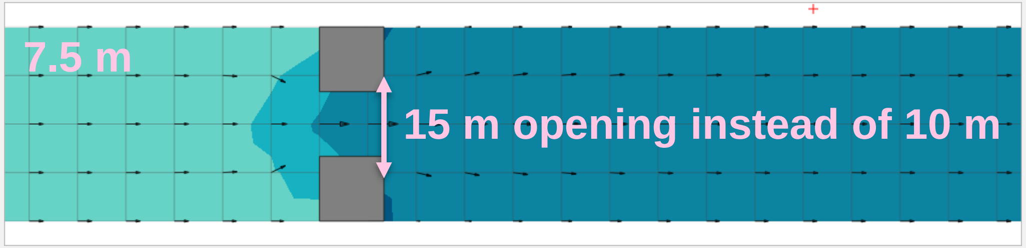
- 5 m mesh needs additional form loss of ~ 0.36
- Matches well with 1D K_{en} of 0.36

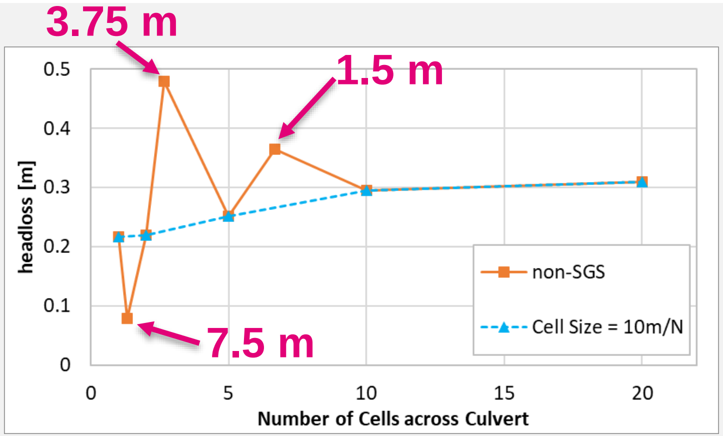


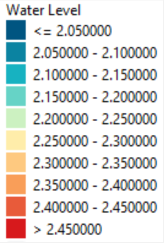
Constrictions – Benchmark Test
2D-2D-2D Coarse Mesh Mismatch

What if the 2D cell size mismatches opening?

- Common issue for fixed grid solvers
- Poor reproduction of afflux





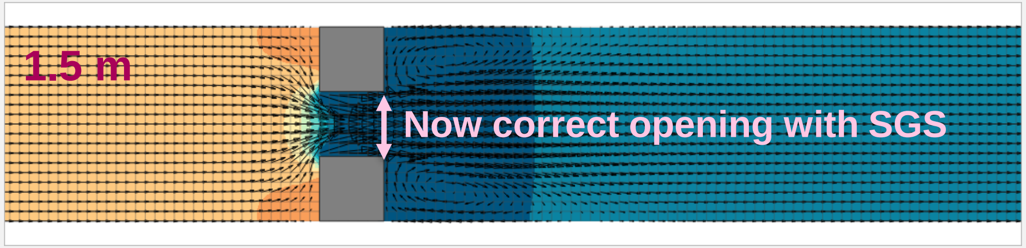


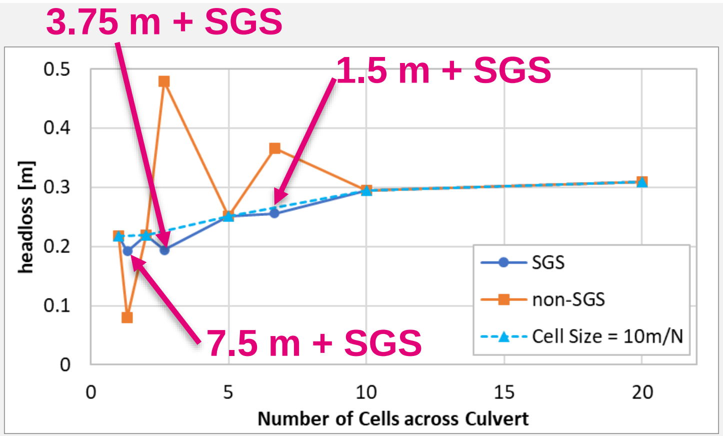
TUFLOW

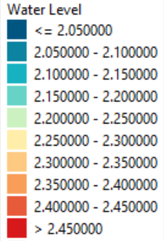
Constrictions – Benchmark Test
2D-2D-2D Coarse Mesh Mismatch with SGS

What if the 2D cell size mismatches opening?

- Common issue for fixed grid solvers
- Sub-Grid Sampling (SGS) helps enormously







TUFLOW

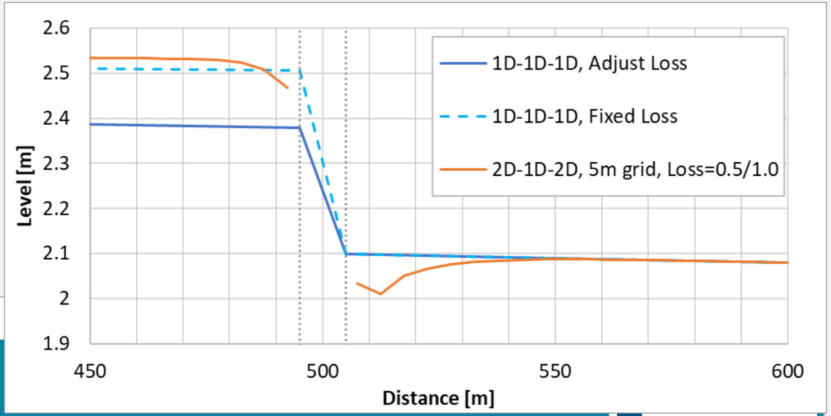
Constrictions – Benchmark Test
2D-1D-2D – Unadjusted Loss Coefficients

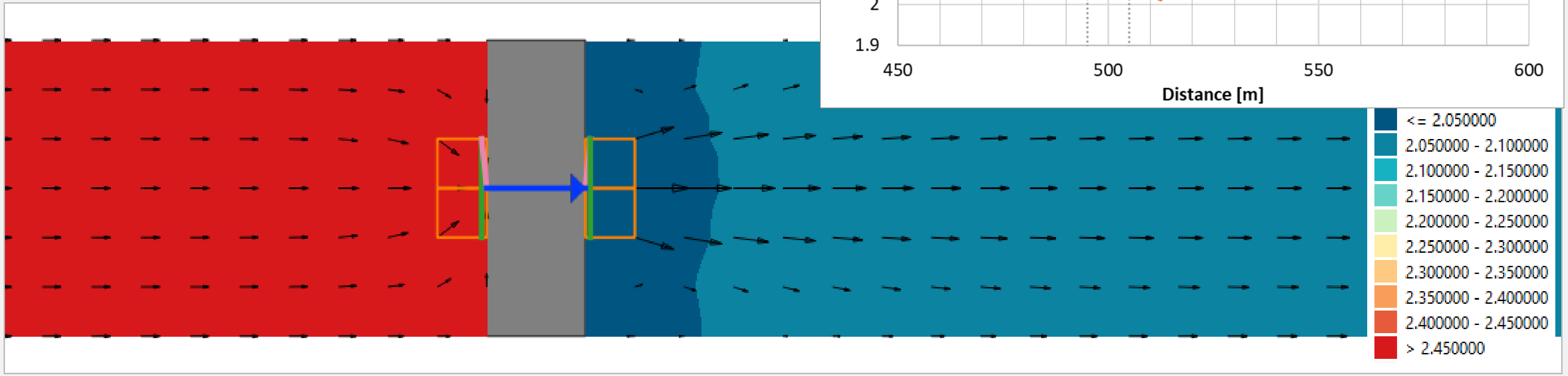
Let's insert a 1D constriction

- Source transfer of water in/out of 1D element (SX Link in TUFLOW)

Overpredicts afflux

- As expected due to duplication of losses

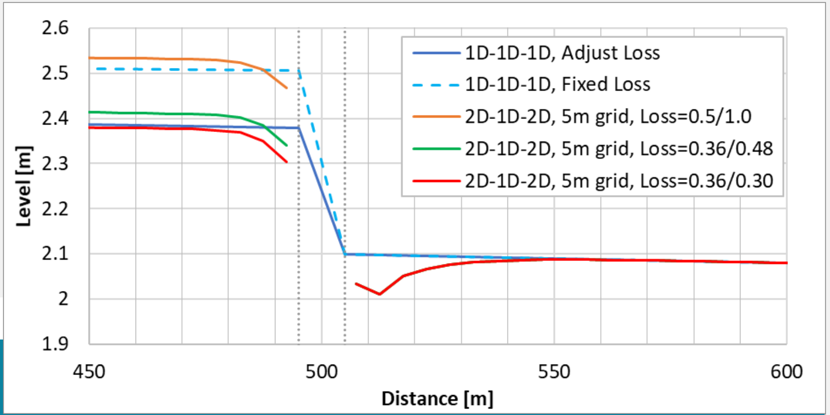


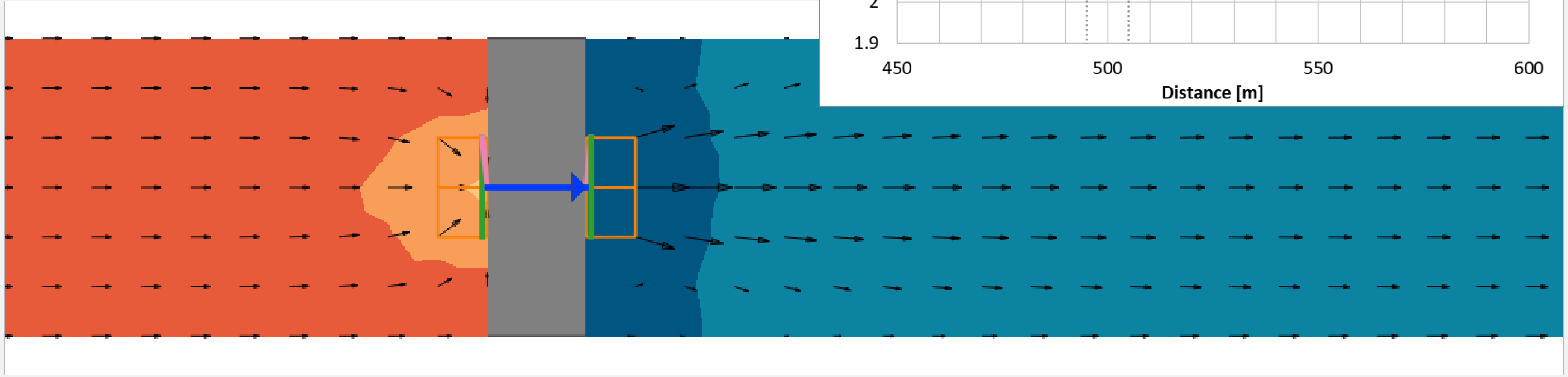


Constrictions – Benchmark Test
2D-1D-2D – Adjust Loss Coefficients

Reduce 1D form losses to account for duplication

- Try 1D adjusted losses (0.36 / 0.48)
  - Still too high an afflux
- Reduce exit form loss
  - 0.36 / **0.30** provides good match



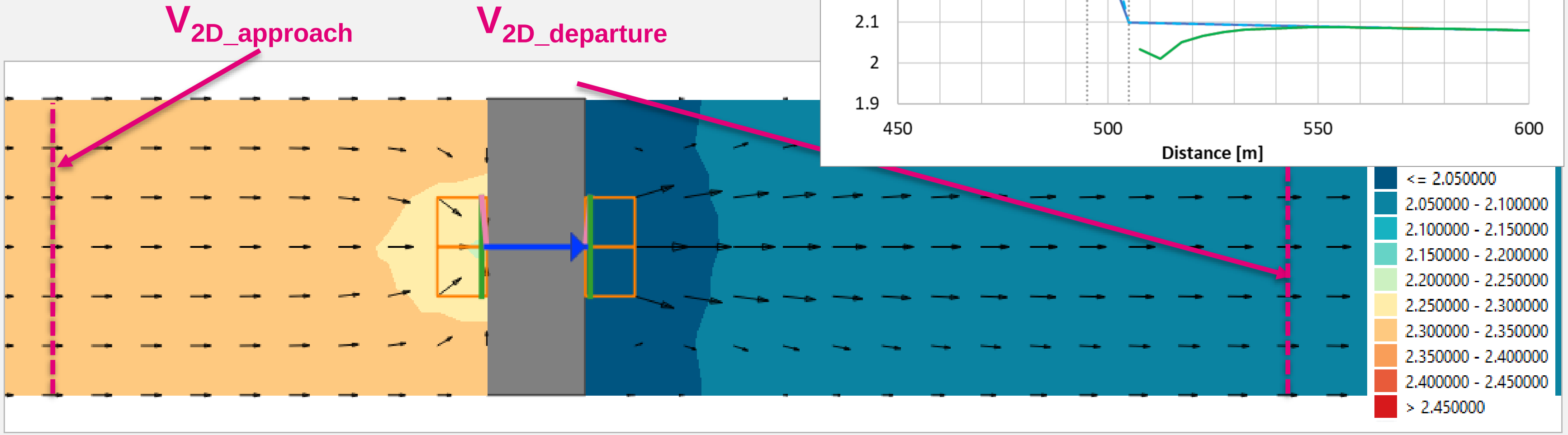


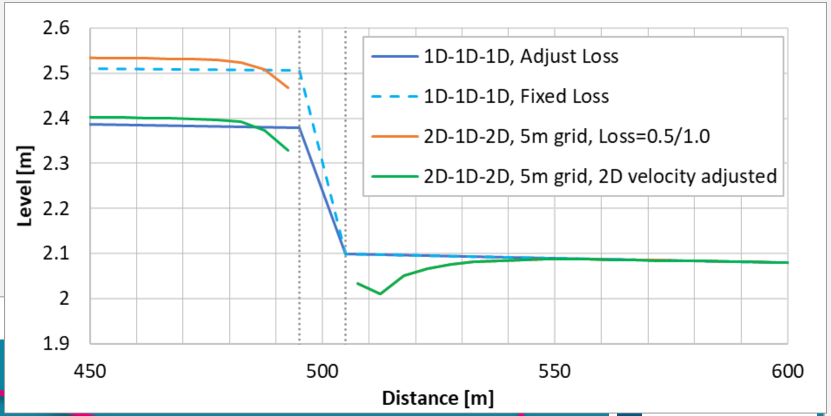
TUFLOW

Constrictions – Benchmark Test
2D-1D-2D – 2D Velocity Adjustment

What if we adjust losses according 2D approach/departure velocities?

- Feature under development in TUFLOW HPC
- Solves issue of what adjustment is needed on 1D





Modelling Hydraulic Structures

Piers using Fine Mesh

Piers
Cause a Constriction

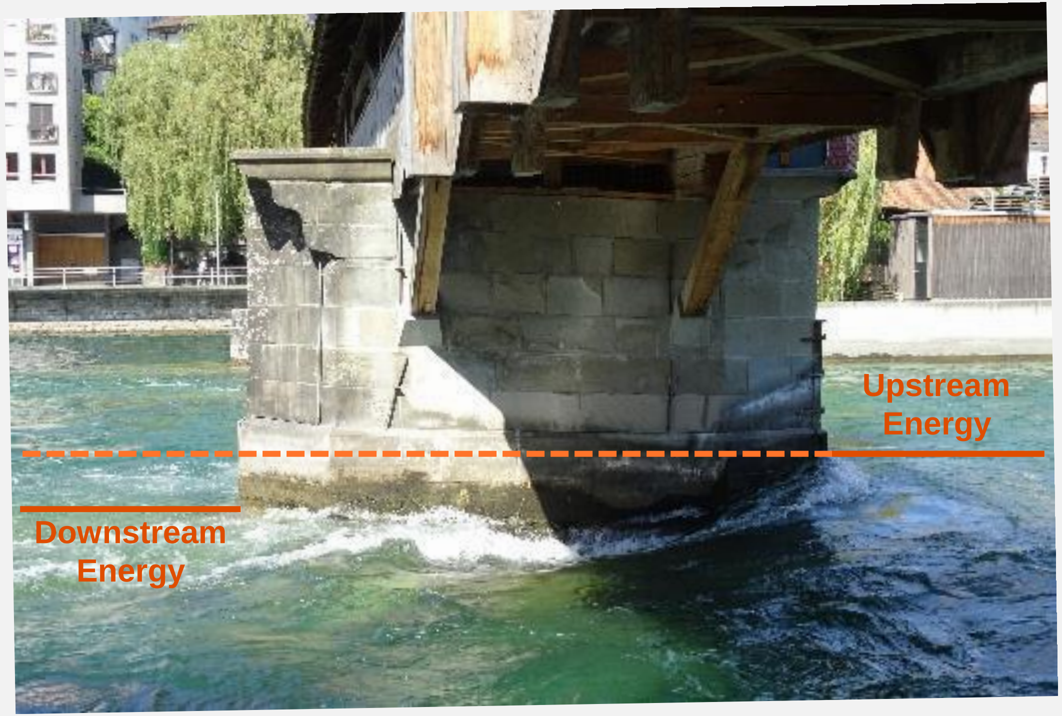
1D approach

- Apply form loss (e.g. K_p HBW)

2D fine mesh

2D using form losses

- Apply form loss
- But how to apply 1D K_p values?
  - Across whole waterway, or
  - Factor up by waterway cell ratio



Piers using a Fine Mesh Flume Benchmark Test

Flume test by Kimura et al (2005)

- 20 cm wide, 10 m long flume
- 4 cm by 4 cm square pier
- Q = 415 cm^3/s
- Depth = 1.14 cm
- Slope = 1/1000
- Manning's n estimated to be 0.0088

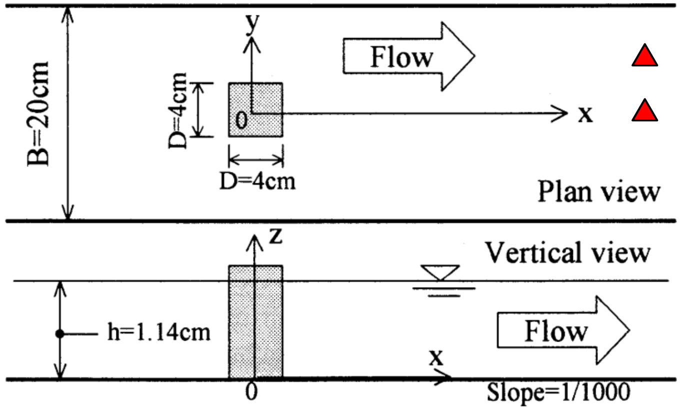


図-2 流れ場の模式図

Gauge 2: x=24cm, y=6cm

Gauge 1: x=24cm, y=0cm

Piers using a Fine Mesh
Flume Benchmark – Fine Mesh

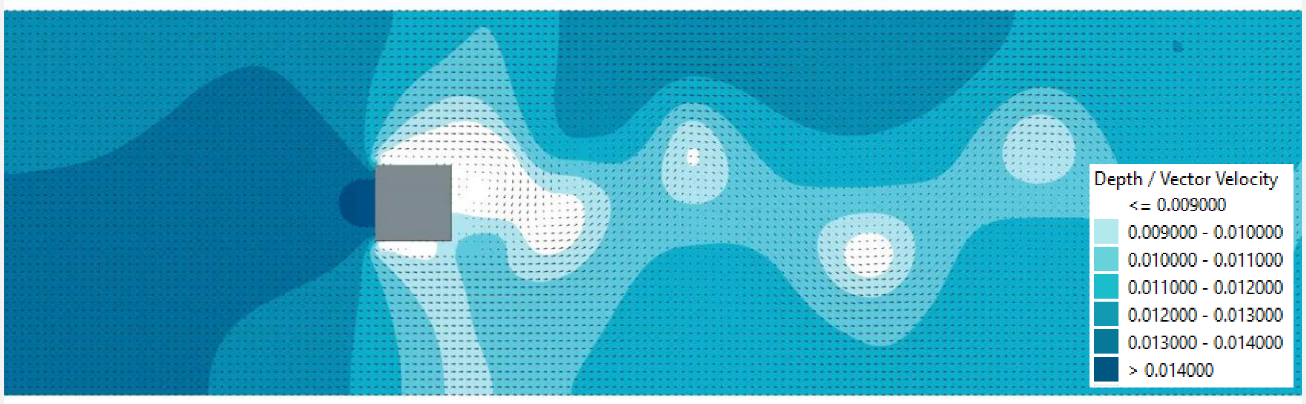
Primary parameter to test is turbulence eddy viscosity coefficient

- Wu turbulence model (isotropic)

Varied Wu parameter from 0.2 to 7.0

- TUFLOW 2020 default is 7.0

Cell size = 4mm



Piers using a Fine Mesh
Flume Benchmark – Fine Mesh

Primary parameter to test is turbulence eddy viscosity coefficient

- Wu turbulence model (isotropic)

Varied Wu parameter from 0.2 to 7.0

- TUFLOW 2020 default is 7.0

Cell size = 4mm

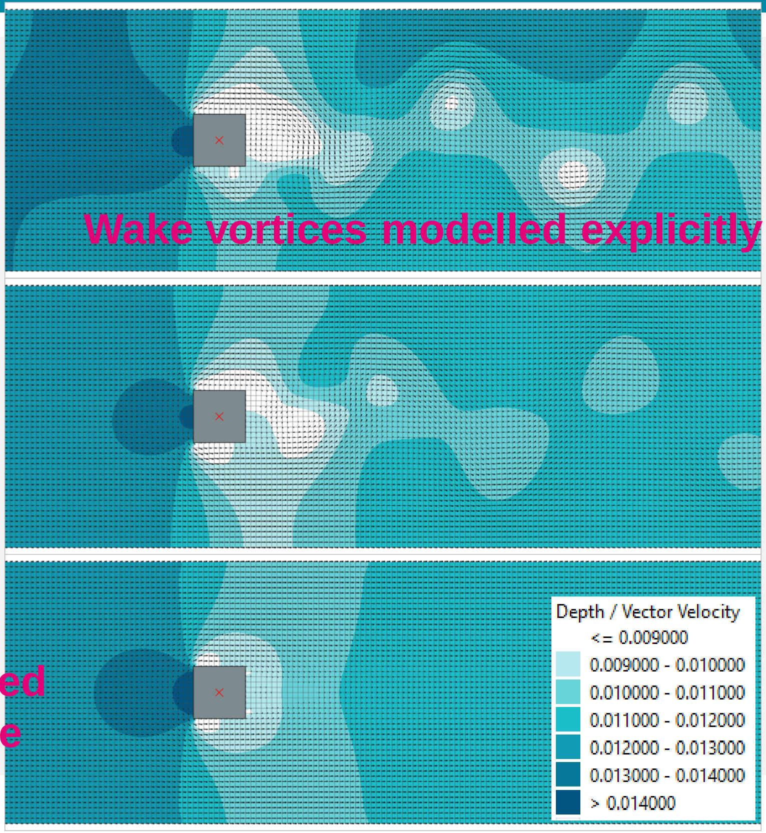
C3D = 0.2
(Kimura et al 2005)

Wake vortices modelled explicitly

C3D = 1.0

C3D = 7.0

Wake vortices modelled as sub-grid turbulence

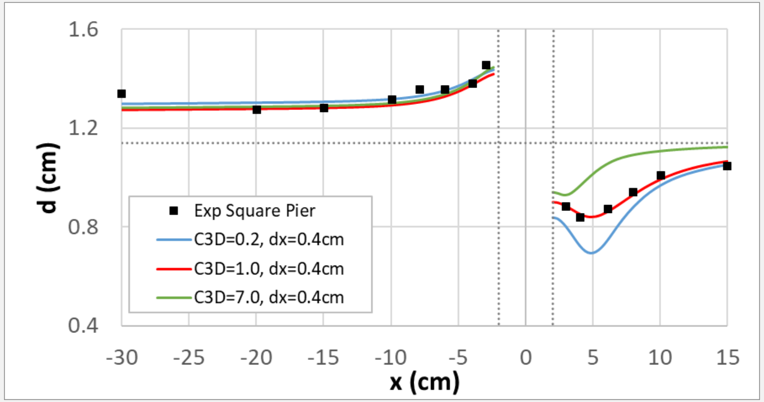


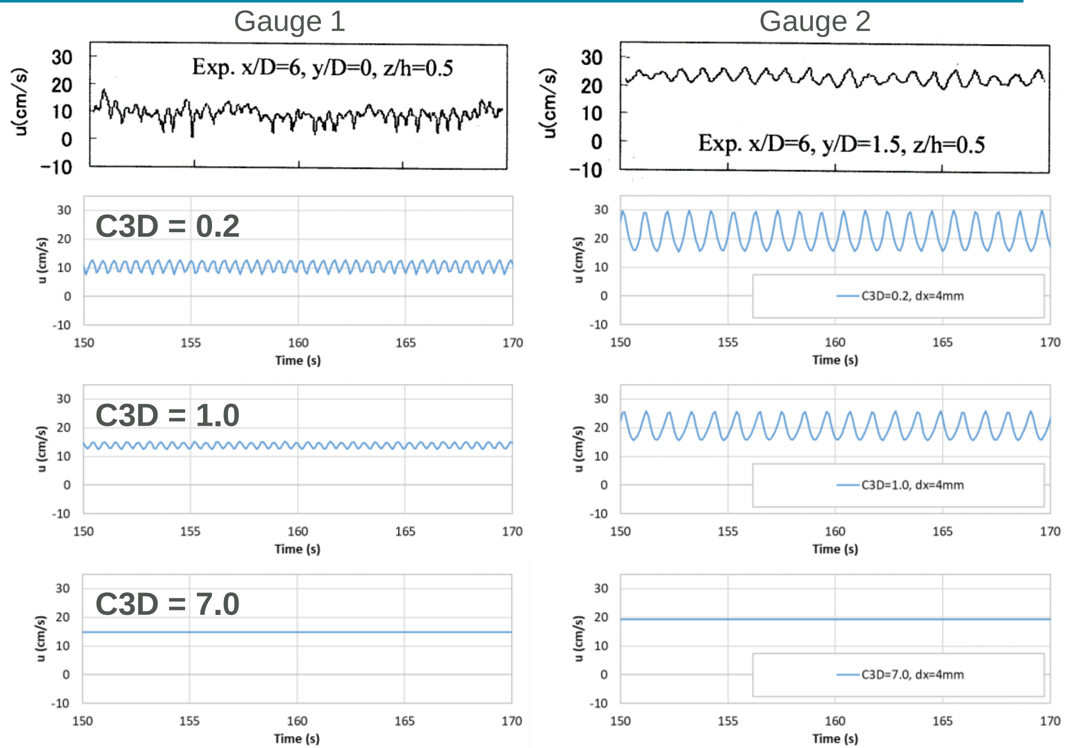
TUFLOW

Piers using a Fine Mesh
Flume Benchmark – Fine Mesh Observations

Primary parameter to test is turbulence eddy viscosity coefficient

- Tested Wu parameter 0.2 to 7.0
- Good match to measured afflux in all cases
- Lower eddy viscosity needed to reproduce wake vortices

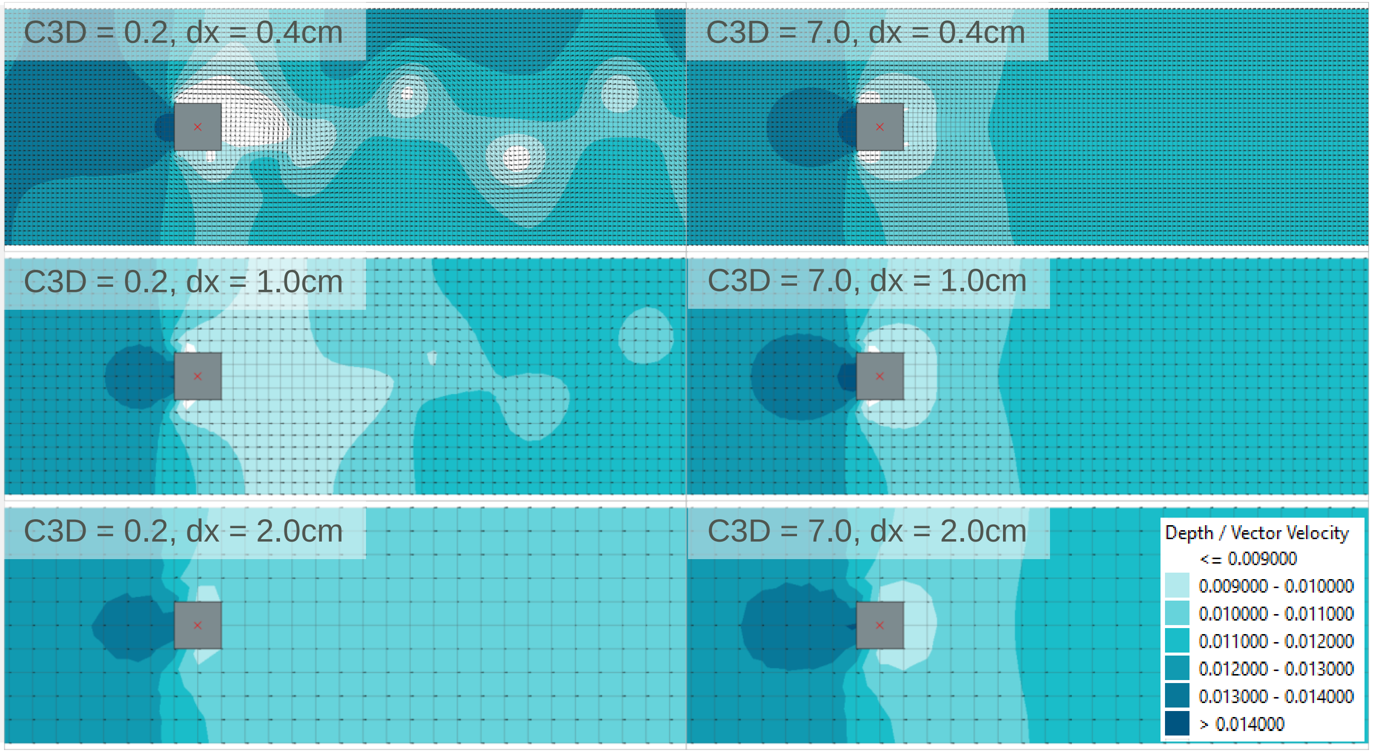




Piers using a Fine Mesh
Flume Benchmark – Cell Size Results Convergence

Cell size results convergence test

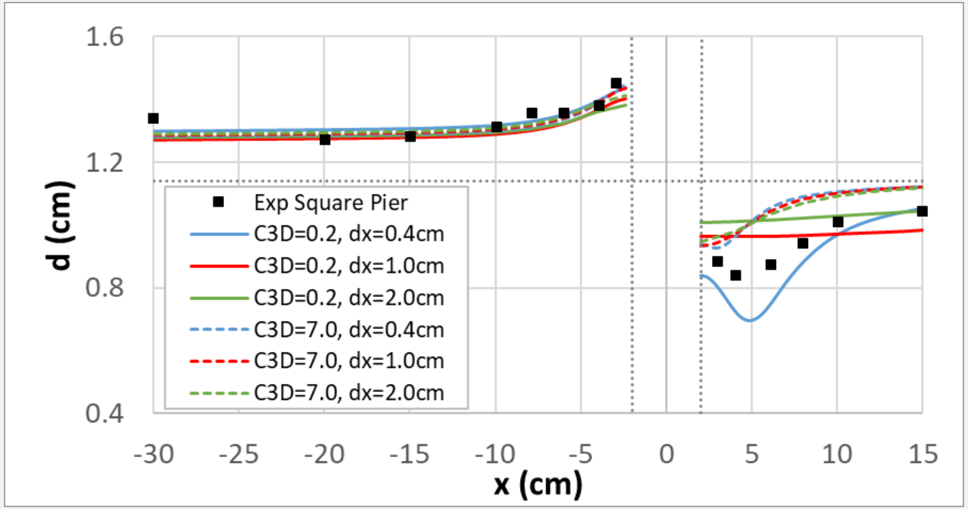
- Increased cell size from
  - 0.4 cm
    (10 cells across pier), to
  - 2.0 cm
    (2 cells across pier)



Piers using a Fine Mesh
Flume Benchmark – Cell Size Results Convergence

Increasing cell sizes

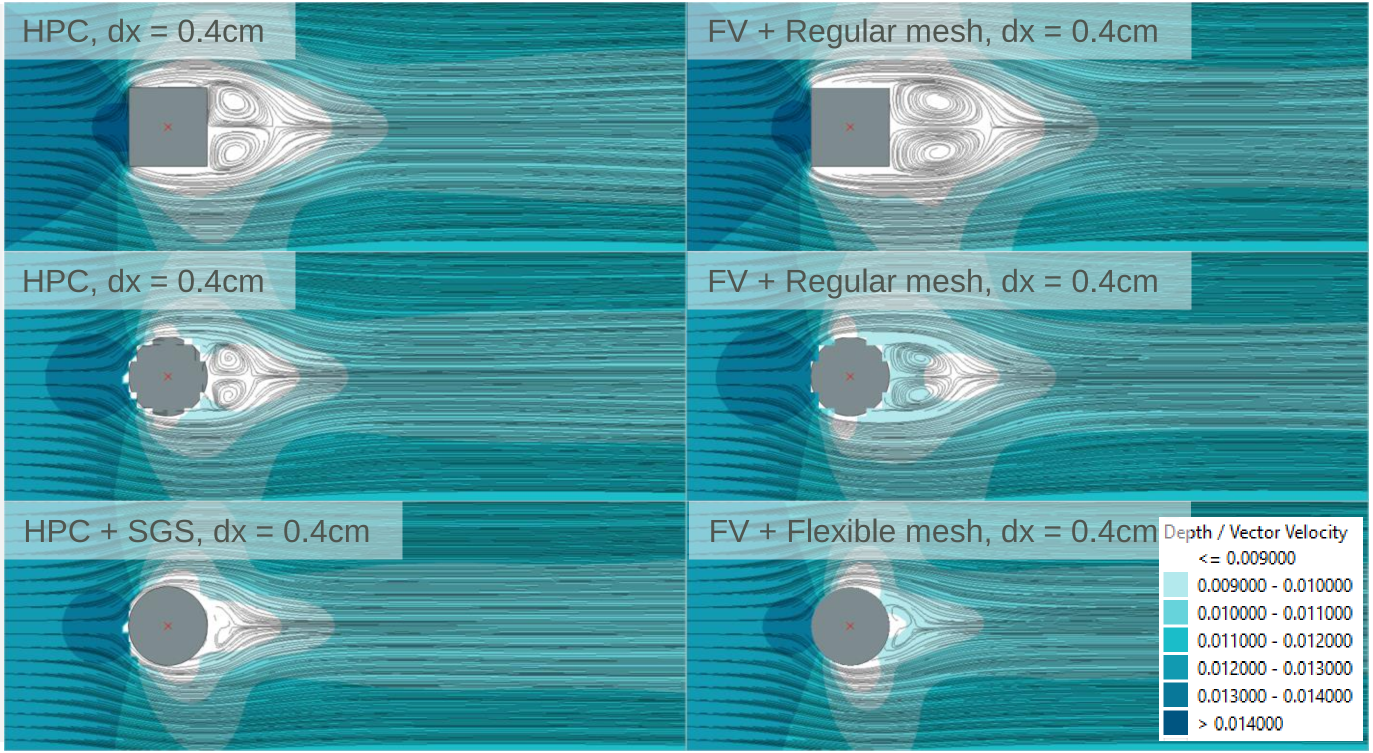
- Still provides reasonable reproduction of afflux
- Loses ability to simulate downstream eddy structure



Piers using a Fine Mesh
Circular Piers

Circular pier cell sizes

- Most of the real-world piers are hydraulically “smooth”



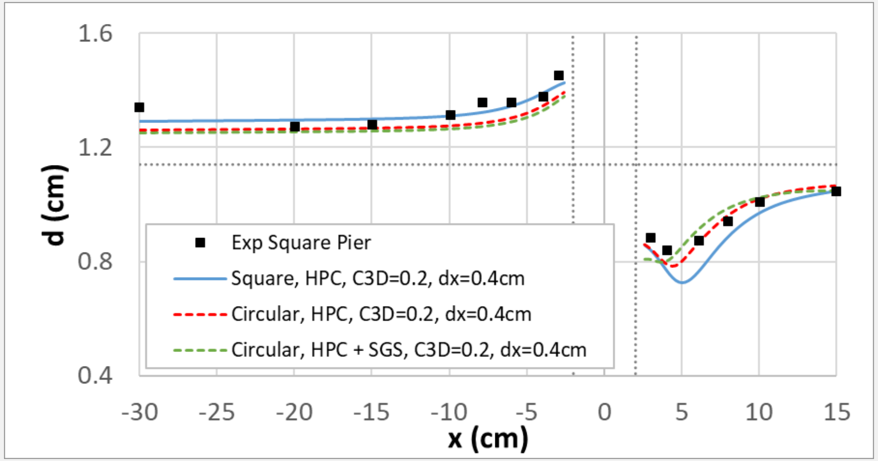
TUFLOW

Piers using a Fine Mesh
Circular Piers

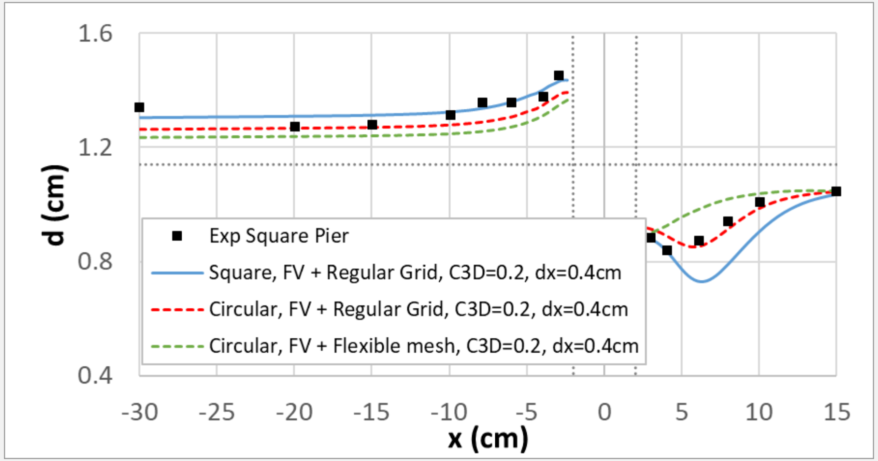
Circular pier cell sizes

- High resolution flexible mesh is needed to represent hydraulically “smooth” pier
- Regular grid may overestimate the head loss, even with SGS

HPC



FV



Modelling Hydraulic Structures

Piers using Form Losses

Piers using Form Losses
Case Study

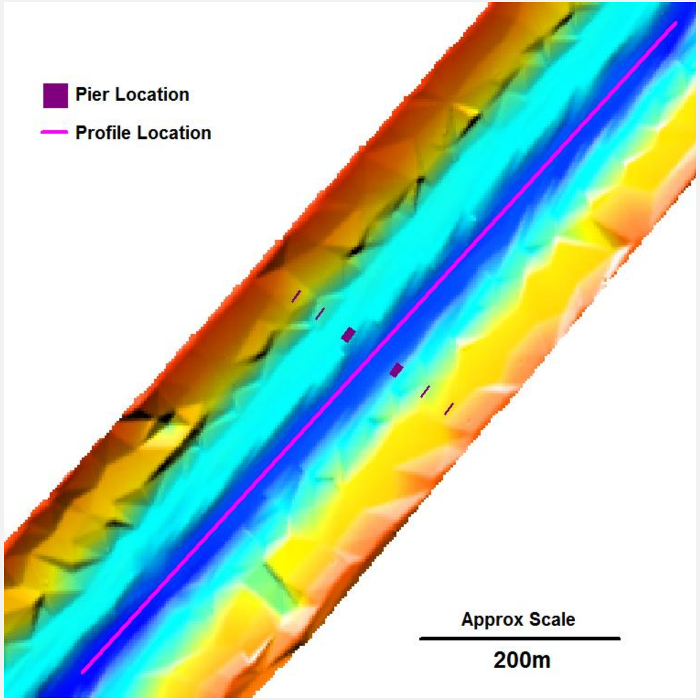
Case Study: Jingling Bridge, China

Pier widths sub-cell or less than several cells

Options

- Try blocking cells with piers (quick to do)
- Apply 1D form loss (e.g. HBW K_p) across waterway
- Apply form losses individually to pier cell(s)

Q: Should we reduce the cell flow area by the pier area when applying form losses?



Piers using Form Losses
Apply Pier Blockage?

Apply pier blockage?

- 2D solution uses the velocity at the cell
- If pier blockage applied, 2D velocity slightly higher, therefore, slightly higher afflux
- Clarify the basis for K_p
  - For HBW, Kp assumes area of the piers is not used in the determination of the velocity
  - Therefore, should not apply blockages from piers
  - Or, reduce Kp to allow for the higher velocities caused by applying pier blockages

A_1 = total water area at section I - sq ft.

If piers are present in the constriction, these are ignored in the determination of A_{n2}. The velocity V_{n2} does not represent an experimentally measured velocity but rather a reference velocity readily computed for both model and field structures.

For practical purposes, the backwater is simply the product of K^*, the backwater coefficient, which was determined experimentally, and the velocity head $V^2_{n2}/2g$. The expression

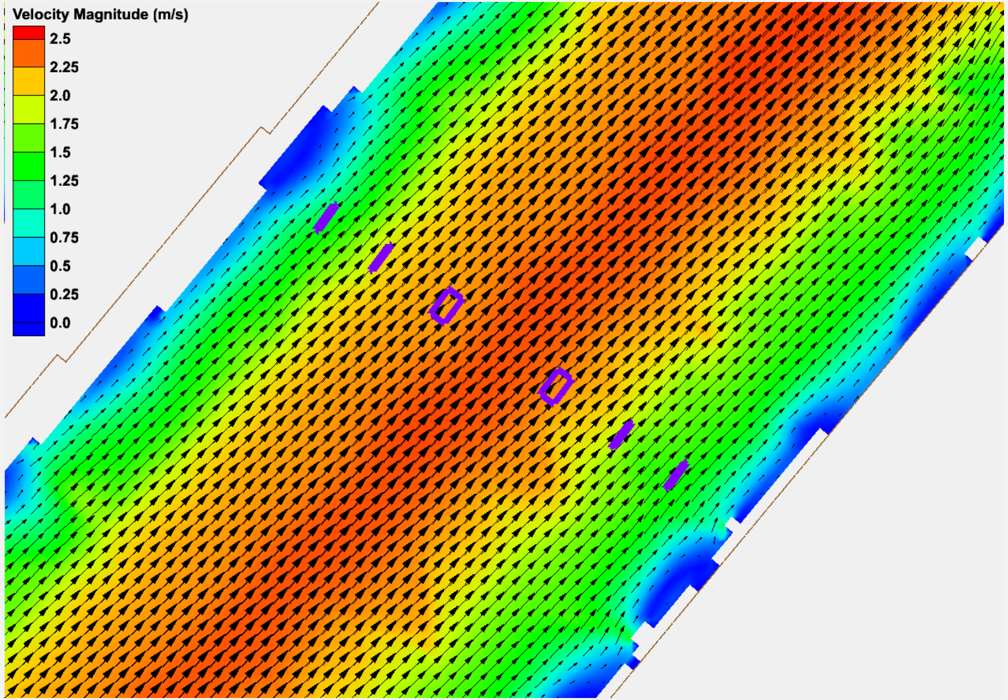
$$\alpha\left[\left(\frac{A_{n2}}{A_4}\right)^2 - \left(\frac{A_{n2}}{A_1}\right)^2\right]$$

- Applying pier blockage is usually slightly conservative (which may not be a bad thing!)

https://wiki.tuflow.com/index.php?title=TUFLOW_FAQ#What_form_loss_coefficient_.28FLC.29_values_should_I_use_for_2d_lfcsh_bridge.3F

Piers using Form Losses
Case Study Velocity Field

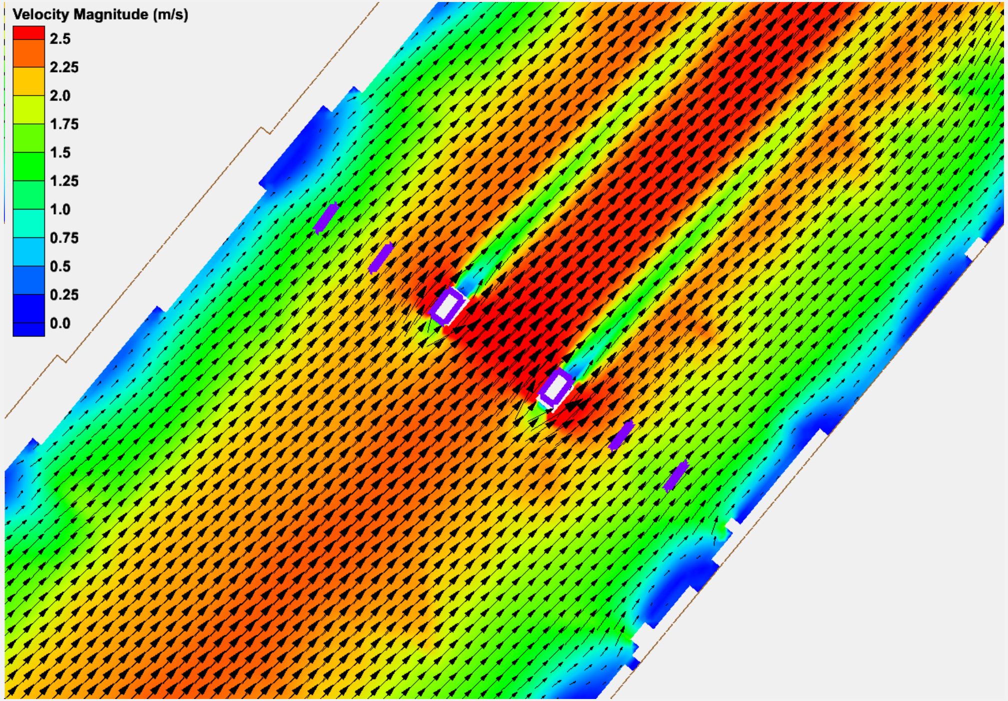
No bridge velocity field



Piers using Form Losses
Whole Cells Blocked Out

Pier cells blocked out (no SGS)

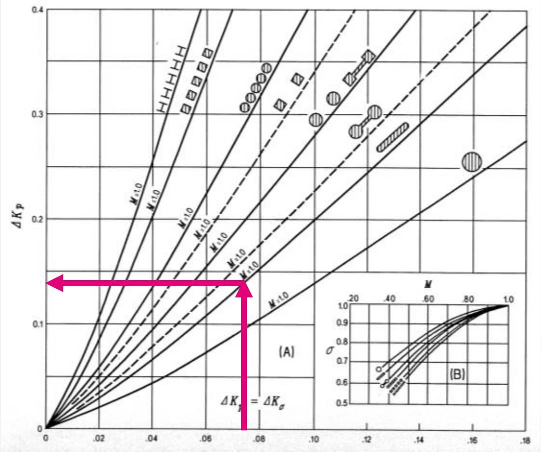
Always problematic for fixed grid or coarse flexible mesh?

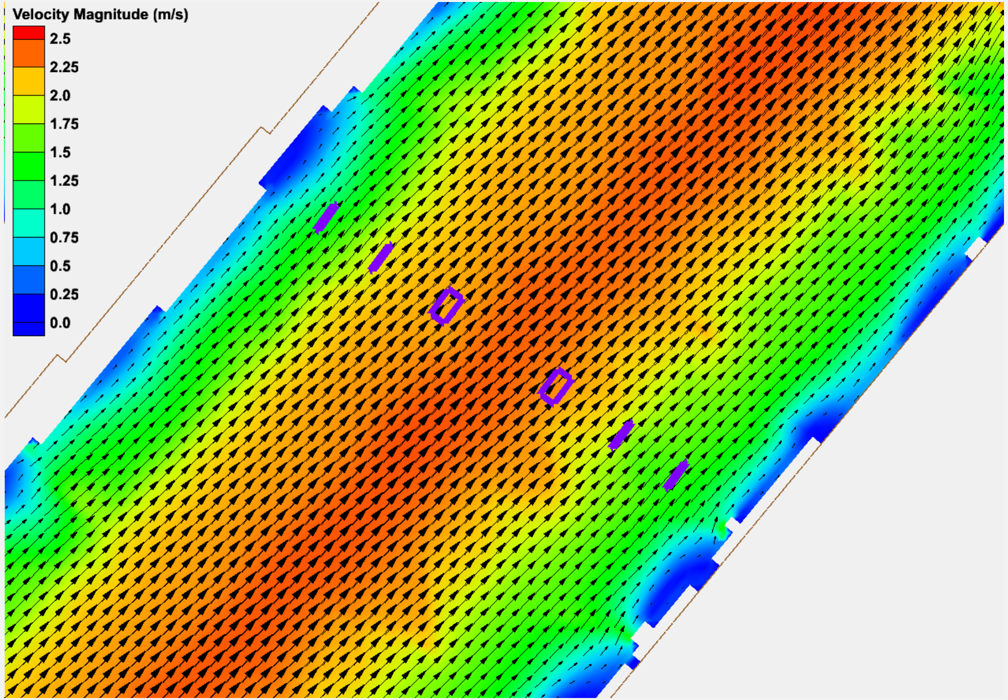


Piers using Form Losses
Apply HBW Kp Pier Form Loss

HBW analysis: K_p = 0.14 (J = 0.073)

- Must apply across entire waterway
- Presence of piers does not show in velocity field

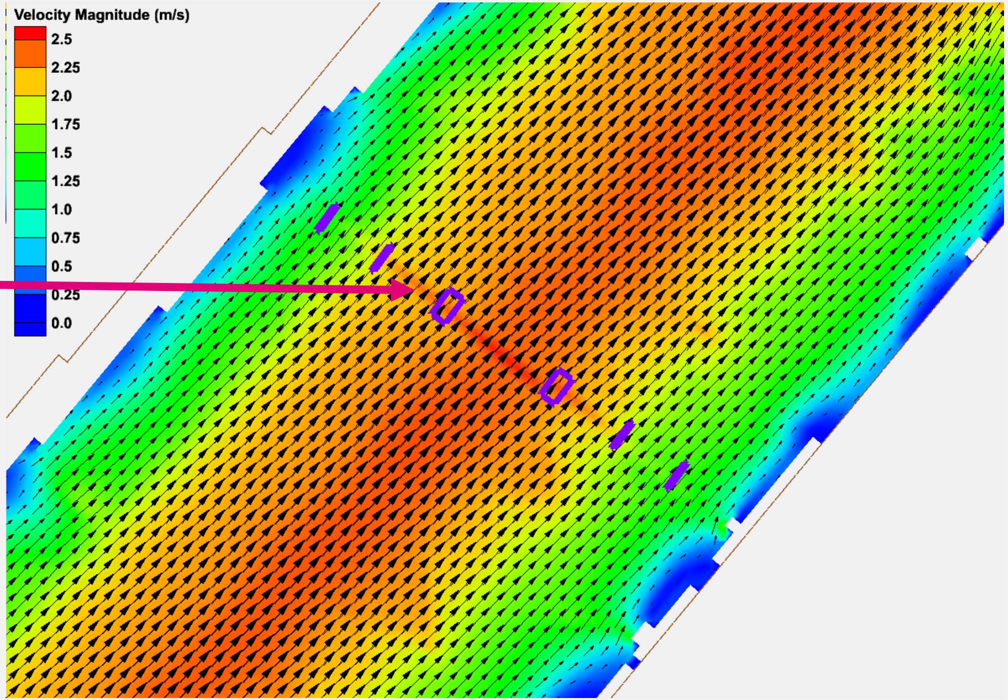




Piers using Form Losses
Apply HBW Kp Pier Form Loss + Blockage

K_p = 0.14 and pier blockage = 7.3%

- Must apply across entire waterway
- Presence of piers does not show in velocity field
- Blockage causes line of slightly higher velocities across entire waterway



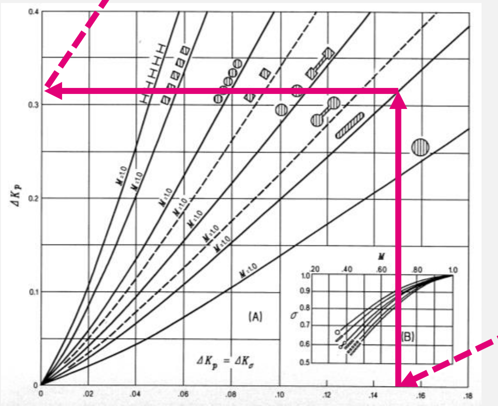
Piers using Form Losses
Form Loss Pier Cell(s) Only

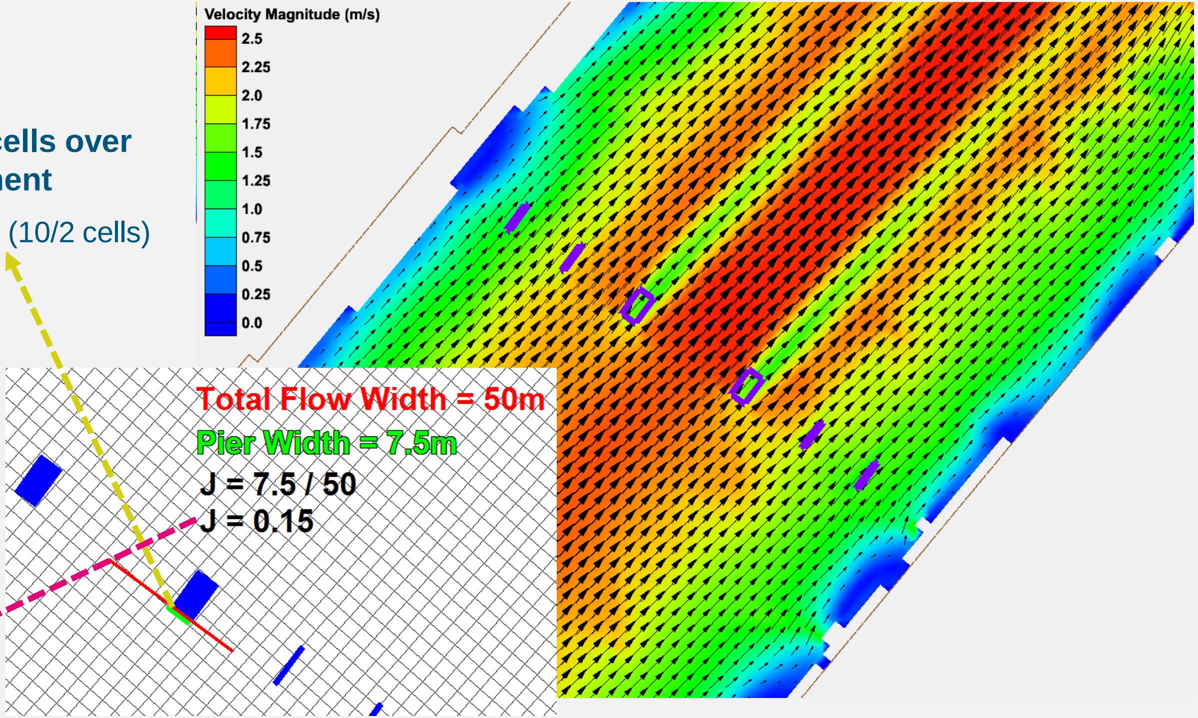
K_p individually calculated per pier

- Large pier $K_p = 0.32$

K_p factored up by ratio of number cells over number of pier cell(s) for pier segment

- Large pier factored K_p from 0.32 to 1.6 (10/2 cells)

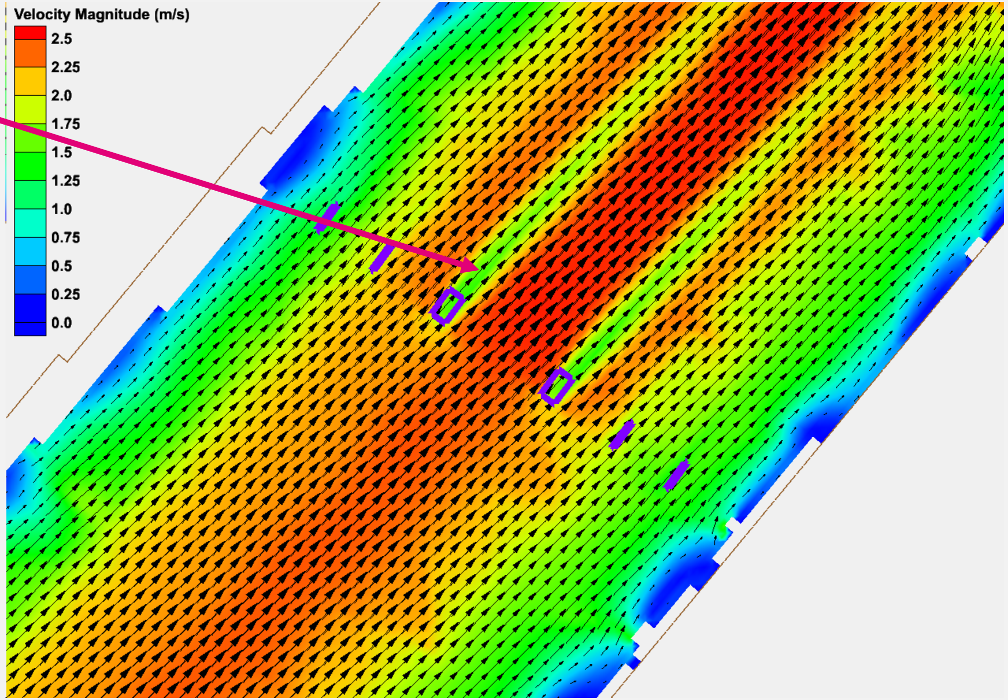




Piers using Form Losses
Form Loss Pier Cell(s) Only

Apply factored up K_p to pier cells

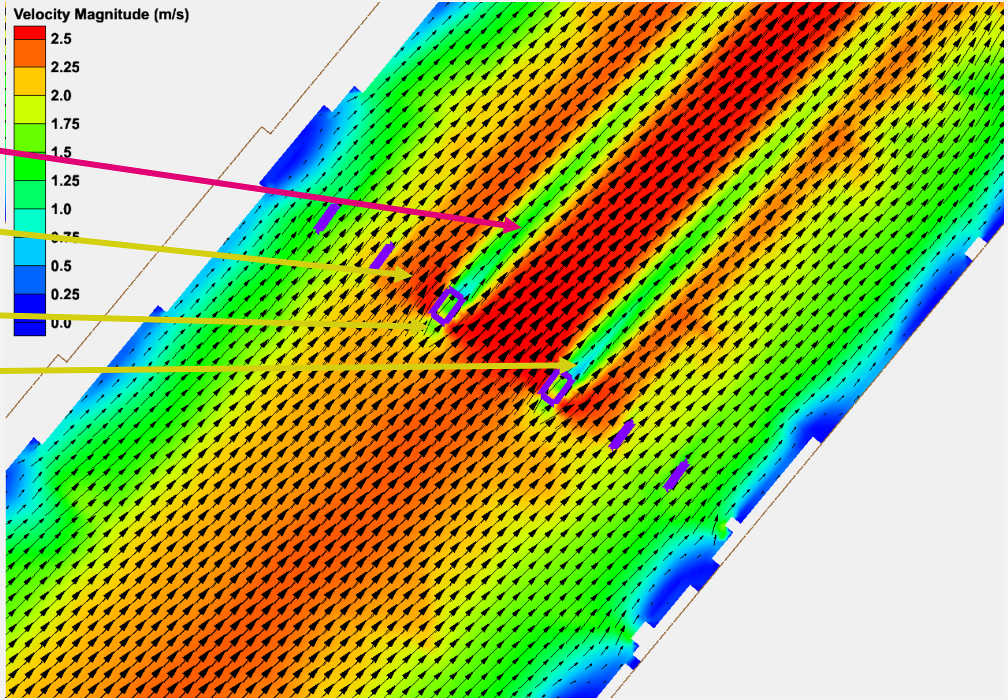
- Presence of piers shows in velocity field
- Effect of individual piers can be assessed



Piers using Form Losses
Form Loss Pier Cell(s) Only + Blockage

Apply factored up K_p to pier cells plus, pier blockage

- Presence of piers shows in velocity field
- Higher (more accurate) velocities between piers
- Surcharging in front of piers
- Set down behind piers
- Effect of individual piers can be assessed



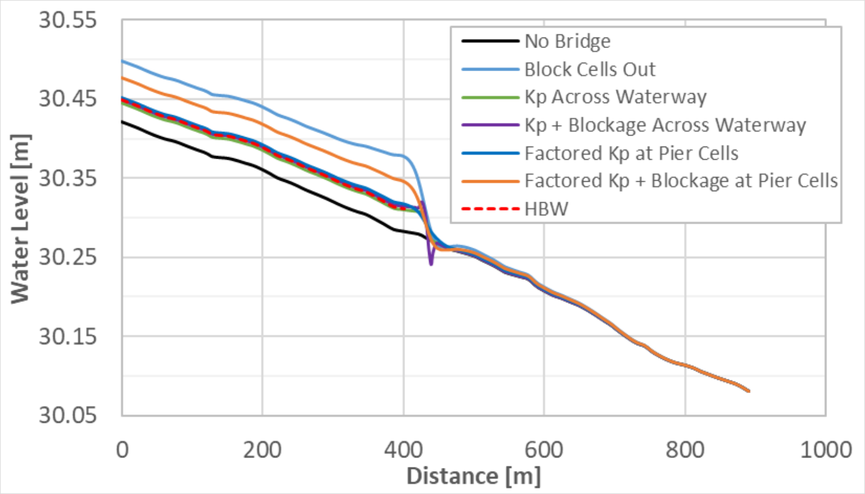
TUFLOW

Piers using Form Losses
Case Study – Afflux Comparison

Desktop check

- Average V_s ~ 2.0 m/s gives an afflux ~ 29 mm

Approach to Pier Losses	Afflux (mm) (Compared to No Bridge)	
	TUFLOW Classic (2011-09)	TUFLOW HPC (2020-10)
HBW Desktop Analysis	29	
Block whole cells out	51	78
Kp Across Waterway	28	25
Kp + Blockage Across Waterway	33	29
Factored Kp at Pier Cells	33	31
Factored Kp + Blockage at Pier Cells	40	57



Modelling Hydraulic Structures

Decks

Bridge Decks – TMR/TUFLOW Investigation Objectives

Improve bridge deck representation in flood models

- Reduce uncertainty of hydraulic model results and lead to better bridge designs

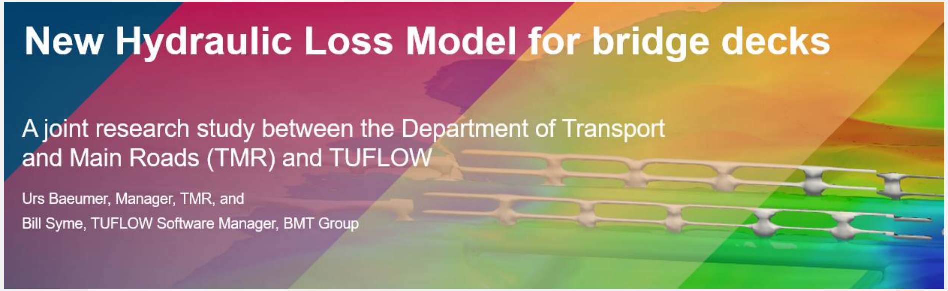
Benchmarking and comparison testing

- Measured data
- CFD modelling
- TUFLOW modelling

Joint research Qld TMR and TUFLOW

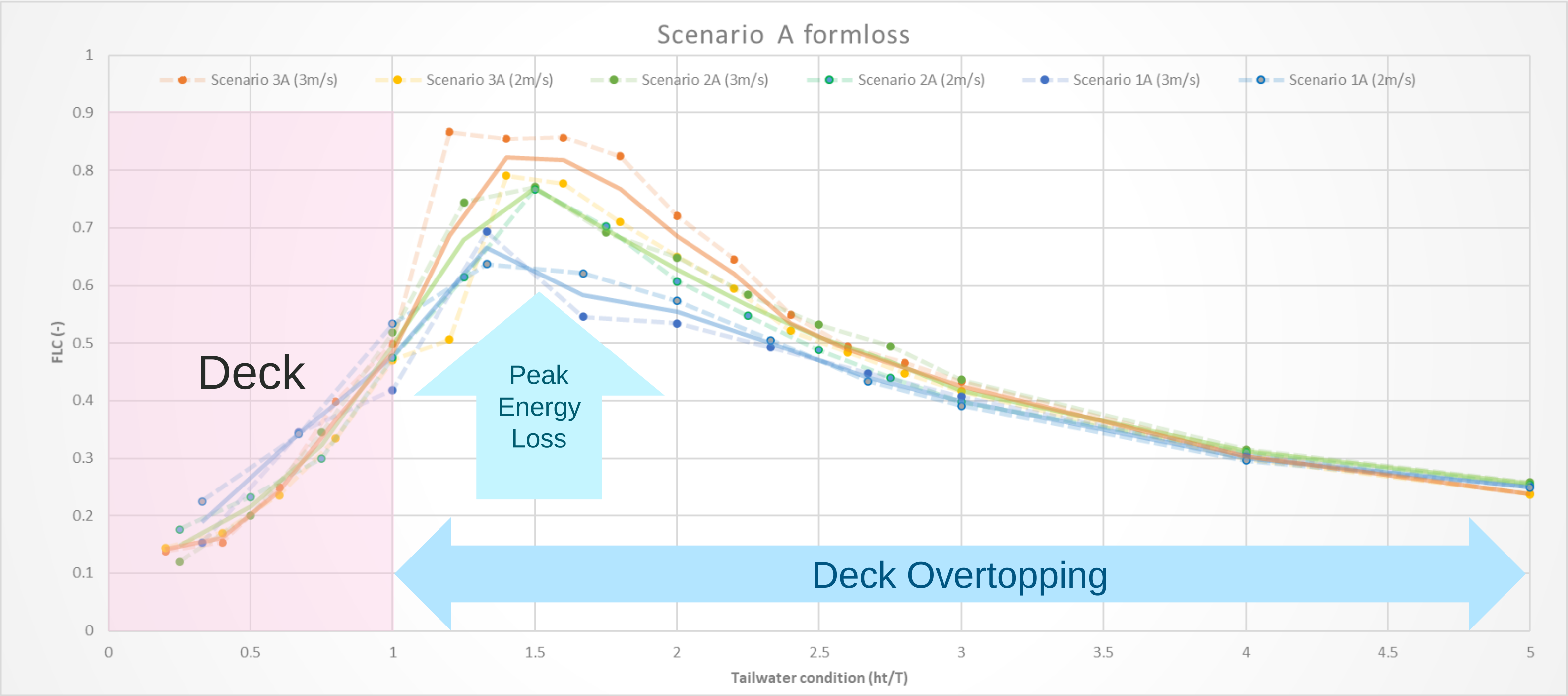
Slides courtesy of:
Urs Baeumer
Manager – Hydraulics and Flooding
Transport and Main Roads Dept, Qld Gov

Presentation scheduled for TMR Tech Forum, Feb 2022



TUFLOW

Bridge Decks – TMR/TUFLOW Investigation
Preliminary CFD Modelling of Standard Decks

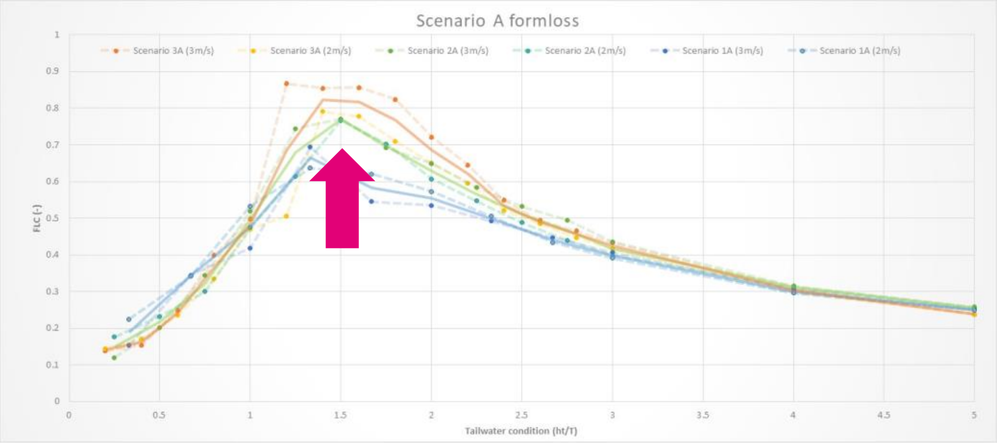


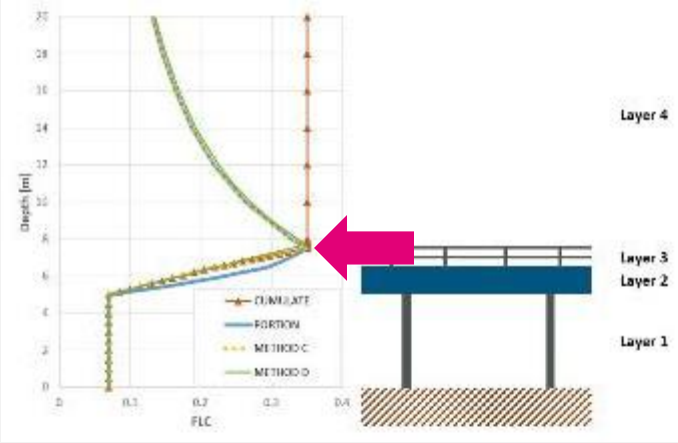
TUFLOW

Bridge Decks – TMR/TUFLOW Investigation
New Approaches in 2D and 3D

Benchmarking

- New methods for modelling bridges in 2D using TUFLOW HPC
- New 3D layer blockage and form drag feature in TUFLOW FV
- Improve representation for pressure flow and submergence





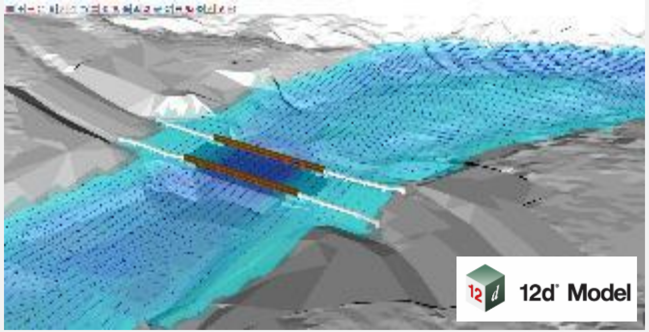
TUFLOW

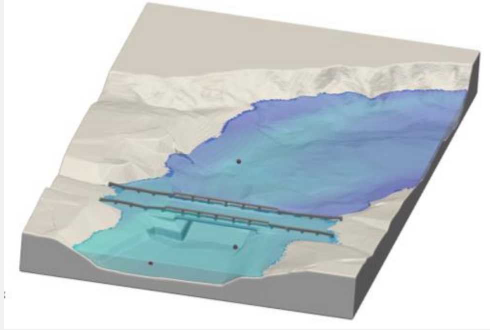
Bridge Decks – TMR/TUFLOW Investigation
Gordon Rd Bridge – Field Data Gauges

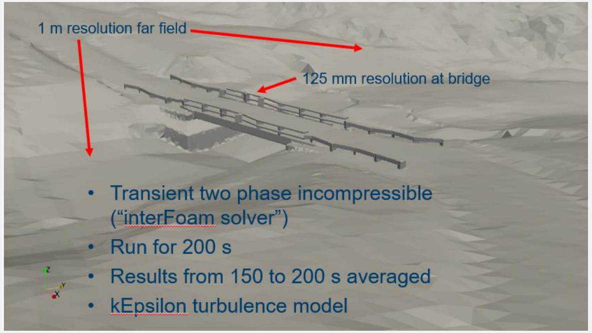
Images sourced: Aquamonix Installation Report AIS-AQ16692, MBRC

TUFLOW

Bridge Decks – TMR/TUFLOW Investigation
Gordon Rd Bridge – Modelling







TUFLOW

FLOW-3D (CFD)

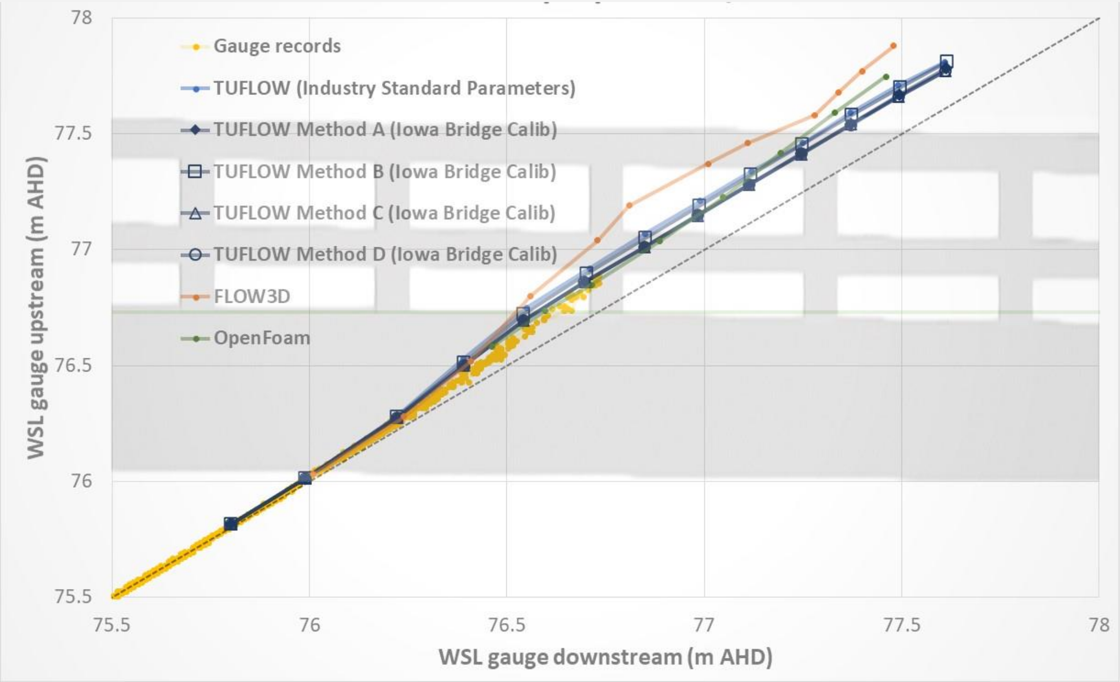
OpenFoam (CFD)

TUFLOW

Bridge Decks – TMR/TUFLOW Investigation
Gordon Rd Bridge Modelling – Early Results

Early Results

- From first event good to reasonable match between gauges, FLOW-3D, OpenFoam and TUFLOW
- Good match using same TUFLOW calibration parameters as derived for Iowa Bridge, USA
- Hoping(!) for significant overtopping flood events this summer
- So far increased confidence in ability of TUFLOW and CFD modelling to reproduce reality



TUFLOW

Modelling Hydraulic Structures

Conclusions

Conclusions
Constrictions

2D models contract and expand flow lines

- Implicitly models form losses if using complete version of the 2D equations

Not all losses are represented

- Coarse meshes will not reproduce losses to same degree as fine meshes
- 3D (vertical) and sub-cell, fine-scale losses not represented
- Need ability to add (minor) form losses (benchmark and calibrate)

1st Order 2D schemes diffusive and overpredict losses – benchmark your software!

Linking 1D structures into 2D

- Useful when the structure is small relative to the 2D cell size
- Large structures (relative to 2D cell size) may duplicate (over predict) losses
- May need to reduce entrance/exit losses (benchmark and calibrate)

Conclusions
Piers

Piers

- Fine mesh, preferably flexible mesh can reproduce affluxes
  - Sub-grid sampling (SGS) helps for fixed grid meshes
  - Quality sub-cell turbulence scheme needed for modelling wakes
- Using form losses best approach for coarse meshes
  - Produces accurate affluxes
  - Can represent piers individually by treating each pier separate K_p analysis and factoring up
  - Careful applying pier blockage to flow area – K_p values may assume this is not the case

Conclusions
Decks

TMR/TUFLOW Investigation

- Provide improved approaches and guidelines for representing decks
- Benchmarked against
  - CFD modelling
  - Field measurements

Finally, as always, benchmark, cross-check and understand your results

- A simple *“What proportion of the $V^2/2g$ has been lost?”* check will often do!

Thank you