Next Generation Urban Surface Water Modelling Capabilities

Chris Huxley
Principal Engineer, TUFLOW, BMT, Brisbane, Australia
E-mail: chris.huxley@bmtglobal.com

Bill Syme
Senior Principal Engineer, TUFLOW, BMT, Brisbane, Australia
E-mail: bill.syme@bmtglobal.com

TUFLOW's 2020 release includes three major development items that are set to redefine how urban stormwater modelling is done in the future, namely sub-grid topography sampling, a cell size independent sub-grid turbulence scheme and quadtree mesh refinement. This paper summarises a range of benchmark tests using the three new features. It highlights the limitations associated with traditional approaches, and how the three new features overcome those constraints. Collectively these features will result in faster simulation speeds and improved result accuracy. It is expected this will translate to improved hydraulic assessments for future infrastructure design, flood impact assessments, flood studies, climate change adaptation work and flood risk management planning.

1. INTRODUCTION

Computational hardware and software constraints have historically limited the ability to accurately model stormwater inundation of urban areas using two-dimensional (2D) schemes in fine detail. To accurately represent 2D free-surface flow along roads, narrow flow paths and pit inlet flow capture, 2D cell sizes of less than one metre are typically required. This has historically presented two major challenges:

1. Excessively long simulation times; and
2. Exceedance of minimum cell size limits associated with the 2D shallow water equations used within 2D software.

Since the late 2000's Graphical Processing Units (GPU) have been used to vastly accelerate model simulation times allowing higher resolution hydrodynamic models than what was possible with conventional Central Processing Unit (CPU) hardware. GPU accelerated software provides tools for rapidly assessing drainage problems at scales previously not feasible, whilst allowing time for the many iterations inherent in the concept and detailed design process of large and often complex urban drainage systems.

New, as of 2020, three complimentary computational techniques have been derived and implemented into TUFLOW to further the evolution of 1D/2D stormwater inundation modelling in urban areas. These techniques have been named:

- Sub-grid topography sampling;
- Cell size independent sub-grid turbulence scheme; and
- Quadtree mesh refinement.

Combined, these new features greatly improve the quality of the modelling, reducing simulation time and memory footprint whilst also improving accuracy. This paper discusses the new computational techniques, including why they're beneficial to urban stormwater modelling situations.

2. METHODOLOGY

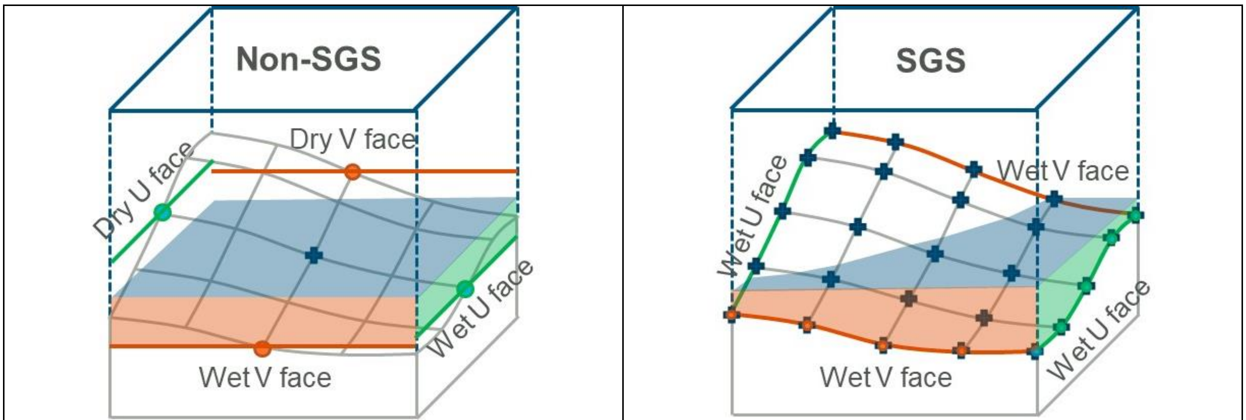
There are numerous numerical schemes solving the one-dimensional (1D) St Venant equations and the 2D depth-averaged incompressible Navier-Stokes equations. For slow moving, benign flows, most schemes should benchmark well to theory and calibrate well to historical events as the dominant term is the surface-friction term (e.g. Manning's equation). Where schemes start to differ is in the ability to represent more complex hydraulics, as present in urban situations. Benchmarking of 2D schemes to demonstrate their ability to accurately reproduce these complex hydraulic phenomena is essential.

This paper uses benchmarking to measure benefits associated with the above-mentioned technology advancements.

2.1. Sub-Grid Sampling (SGS)

The 2D depth averaged Navier Stokes equations, known as the Shallow Water Equations (SWE), are commonly simulated on either a uniform regular (cartesian) mesh of square cells, or an irregular mesh comprised of cells of varying shape and area, typically triangles and quadrilaterals (Lane, 1998). Traditionally, the approach to specifying the cells' terrain elevations is to take either the elevation at the cell centroid or the average elevation within the cell. The resulting mesh is a series of flat-bottomed cells with linear relationships between water surface elevations and cell water volume (cell water depth multiplied by cell area). Furthermore, connections between adjacent cells and the cell faces are rectangular in shape, with linear relationships between water surface elevation and the face flux area used to convey flow. The traditional approach is schematised in Figure 1(a).

Sub-Grid Sampling (SGS) is a new approach to the treatment/interpretation of topography for 2D SWE models. SGS extracts sub-grid data from an underlying digital elevation model (DEM) (which is typically at a finer resolution than the model grid resolution) to develop a non-linear relationship between the water surface elevation and the cell's volume to describe the cells' storage capacity. SGS also generates a non-linear relationship between the water surface elevation and the cell face area and cell width (or wetted perimeter) to improve the representation of the fluxes across the cell faces as flow is conveyed throughout the model domain. The SGS approach still computes a single water level for each cell, but the computations to determine the cell volume and cell face fluxes utilise the higher resolution terrain data. Figure 1(b) provides a schematised presentation of SGS.



(a) Traditional Approach (single elevation per cell centre and cell face)	(b) Sub-Grid Sampling (SGS) (sub-grid non-linear elevation / volume and face wetted perimeter relationship)

Figure 1 2D Mesh Terrain Sampling Options

The traditional approach has historically been adequate if the mesh resolution is sufficiently fine to represent the hydraulically significant terrain features. In some cases, particularly in an urban setting, this may however require high resolution modelling, which can become impractical both in terms of computational memory and time. SGS assists in overcoming these challenges as it can achieve equivalent and, in some cases, superior accuracy using larger cells (lower resolution) compared to the traditional approach. This means a model using the new SGS approach can be designed to run faster.

SGS has been successfully benchmarked to a wide range of hydraulic scenarios, with substantial benefits noted. Two of the scenarios that are directly relevant to urban stormwater modelling are:

- Theoretical solution to a rectangular channel.
- Flume test of flow around a smooth bend.

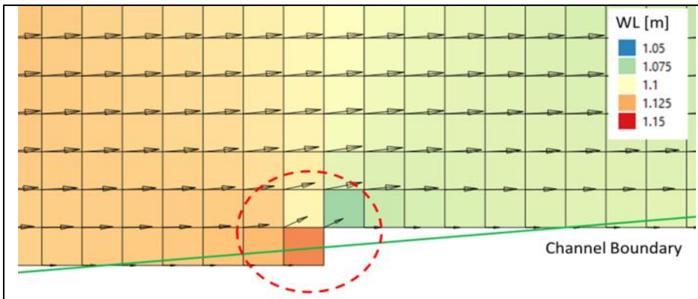
They are discussed in this paper in the following sections.

2.1.1. Theoretical Rectangular Channel

This theoretical scenario is a 1,000m long uniform channel with a 100m wide rectangular section. The test scenario model represents uniform flow conditions that should reproduce Manning's equation for bed resistance. The computational mesh was rotated to provide various degrees of misalignment with the flow direction, and the simulations compared to the theoretical water level and energy slope derived from Manning's equation.

Traditional Single Elevation Per Cell Result

The theoretical rectangular channel scenario demonstrates how a regular mesh or poorly designed irregular mesh using the traditional single elevation per cell (Figure 1(a)) fails to reproduce simple uniform flow hydraulics when there is a sharp change in elevation at a wet/dry interface that is not aligned to the mesh. This is caused by a flow disturbance created where the water flows into dry or inactive cells along the wet/dry boundary as shown in Figure 2.



Example of artefact velocities near wet/dry boundary for traditional single elevation per cell approach

Figure 2 Wet/Dry Interface Flow Disturbance

Figure 3 shows energy (dashed lines) and water surface (solid lines) along the channel for different orientations of the mesh. The results conform exactly with Manning's equation when the mesh is perfectly aligned with the rectangular channel (0° scenario). However, as the mesh is rotated the results do not reproduce Manning's equation with an elevated water surface (greater energy loss) occurring. The distorted, non-uniform, velocity field leads to a jagged saw-tooth effect at the wet-dry interface that obstructs the flow field. This leads to artificial energy losses and the undesirable elevated water surface, greater than the target result. This numerical artifact and associates result inaccuracy has been observed in regular mesh solvers by others prior to this paper (e.g. Hardy et al., 1999).

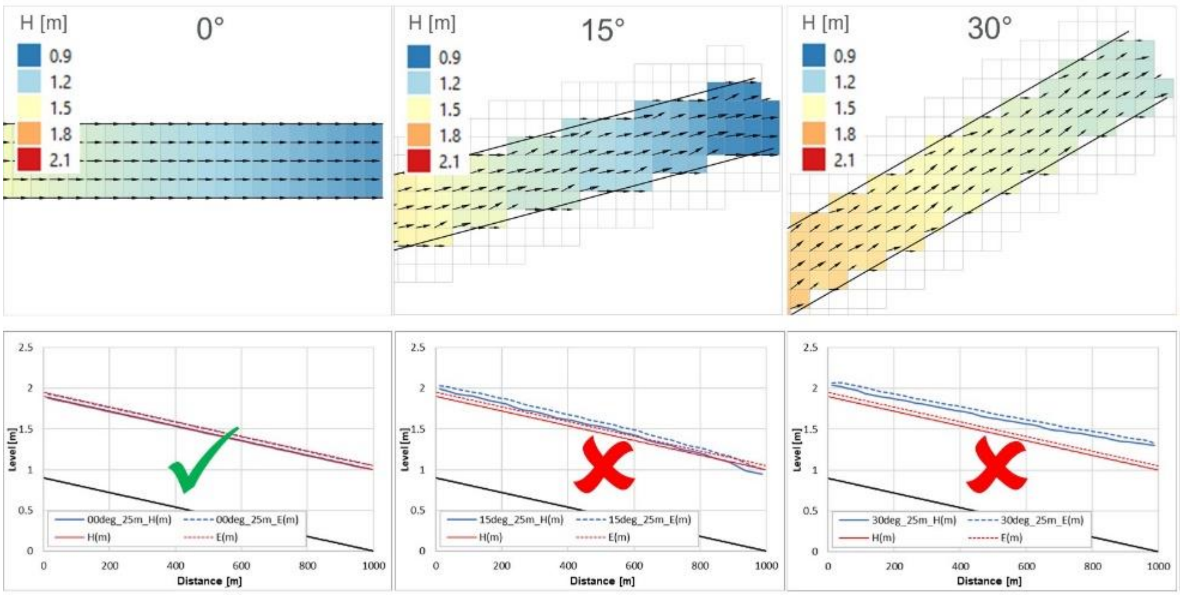


Figure 3 Rectangular Uniform Channel Benchmark Result: Traditional Single Elevation Per Cell Approach

Sub-Grid Sampling Approach Result

SGS was applied to identical model scenarios as those shown above. The results are presented in Figure 4. Irrespective of the mesh rotation, the model using SGS produced results conforming to Manning's equation. Interrelated to this, there is no distortion of the velocity field along the wet/dry interface with the velocity vectors aligned parallel to the rectangular channel for all tests.

These results indicate that a regular mesh 2D solver with SGS can accurately reproduce hydraulic flows at any mesh orientation (i.e. the 2D solution's results are not dependent on mesh orientation). This finding is particularly relevant for stormwater modelling practitioners since linear concrete stormwater drainage channels are a common feature in many urban situations.

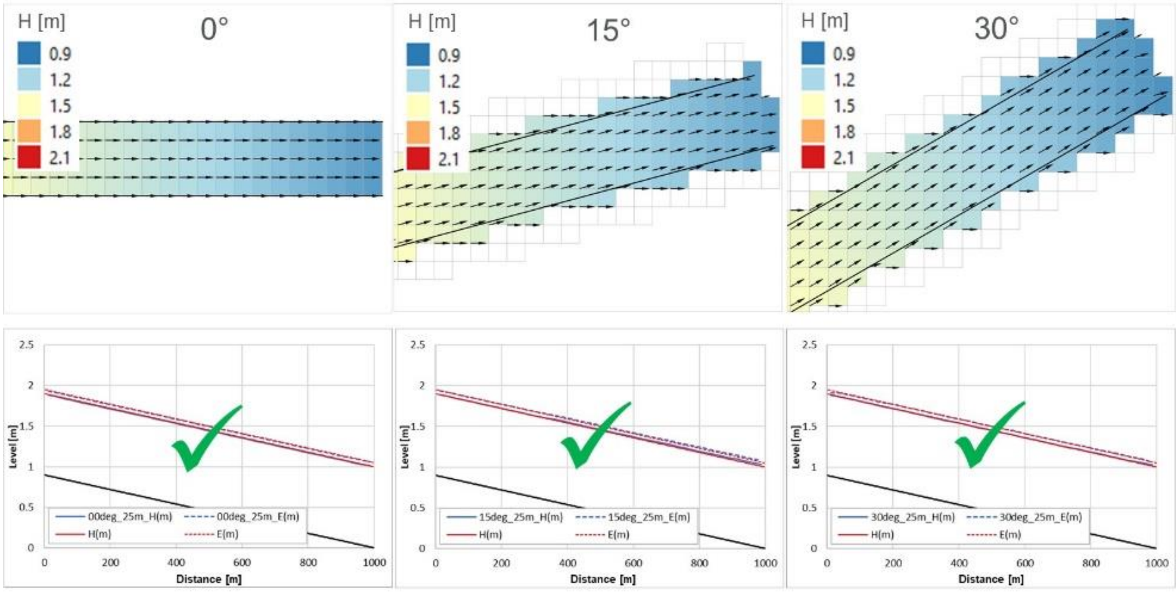
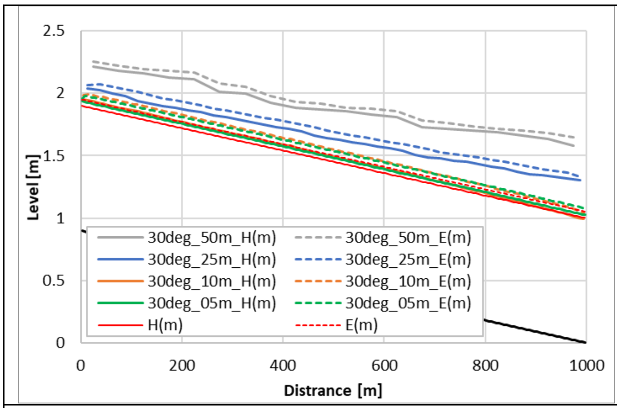


Figure 4 Rectangular Uniform Channel Benchmark Result: Sub-Grid Sampling Approach

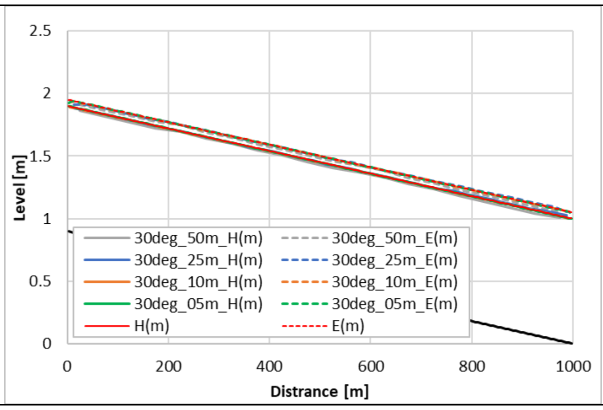
Impact of Cell Size

For the traditional single elevation per cell approach the worst-case scenario was found to be the 30° rotation scenario. We have used this worst-case scenario to test for cell size convergence. The testing aims to determine what cell resolution in the traditional approach is required to negate the above-mentioned undesirable saw-tooth loss effect. It was expected that with increasing coarseness (greater cell size) the results will have poorer agreement with Manning's equation. This expectation corresponds with 2D modelling guidelines that emphasise the need to have enough cells across the primary flow paths (Australian Rainfall and Runoff, 2012).

The 30° rotation test scenario was simulated for a range of different cell sizes: 5m, 10m, 25m and 50m. Results for the traditional and SGS approach are shown in Figure 5. Results for the traditional approach confirmed expectations that the coarser the mesh, the poorer the result compared the Manning's equation. Grid sizes had to drop to below 10m before the traditional approach produced a similar result to Manning's. However, the SGS results indicate not only is SGS insensitive to mesh rotation, it also exhibits cell size convergence at a much larger cell size compared to the traditional approach. In this scenario, SGS is able to achieve a similar result at a cell size of 50m, compared to the <10m cell size required using the traditional approach. This has significant implications for stormwater modelling practitioners. It means models using SGS will be able to use larger cell size compared to the traditional approach. This reduces the model's cell count, in turn reducing the simulation time.



(a) Traditional Approach
(single elevation per cell centre and cell face)



(b) Sub-Grid Sampling (SGS) Approach

Figure 5 Rectangular Uniform Channel Benchmark Result: Cell Size Convergence

2.1.2. U-Bend Flume

The U-Bend flume test (De Vriend, 1978) comprises a uniform U-bend channel with a shallow rectangular cross-section, shown in Figure 6. During the flume testing, surface water elevations along the inner bank, centre and outer bank were measured, demonstrating super-elevation effects on the outside of the U-bend. Water levels at the inside, centreline and outside were measured every 1m in longitudinal length or at 15° intervals around the bend. The flow applied was 0.189 m^3/s with a downstream water depth of 0.18m. For modelling purposes, a Manning's n value of 0.0125 was used, based on the published roughness height of 0.001 to 0.0005 m.

Traditional Single Elevation Per Cell Result

Figure 7 presents the results for the traditional single elevation per cell approach. They demonstrate the influence of the same jagged saw-tooth wet-dry boundary loss effect as the rectangular channel benchmarking identified. As a result, the modelled water levels were higher than the measured data

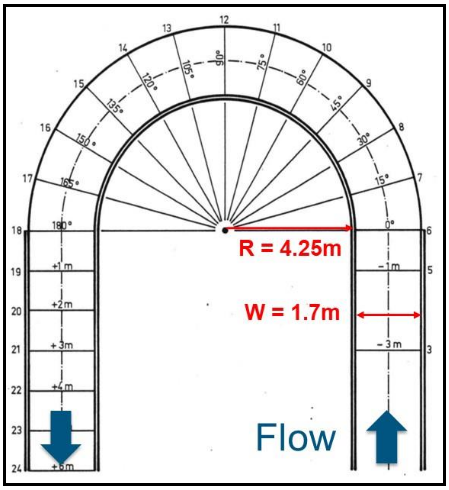
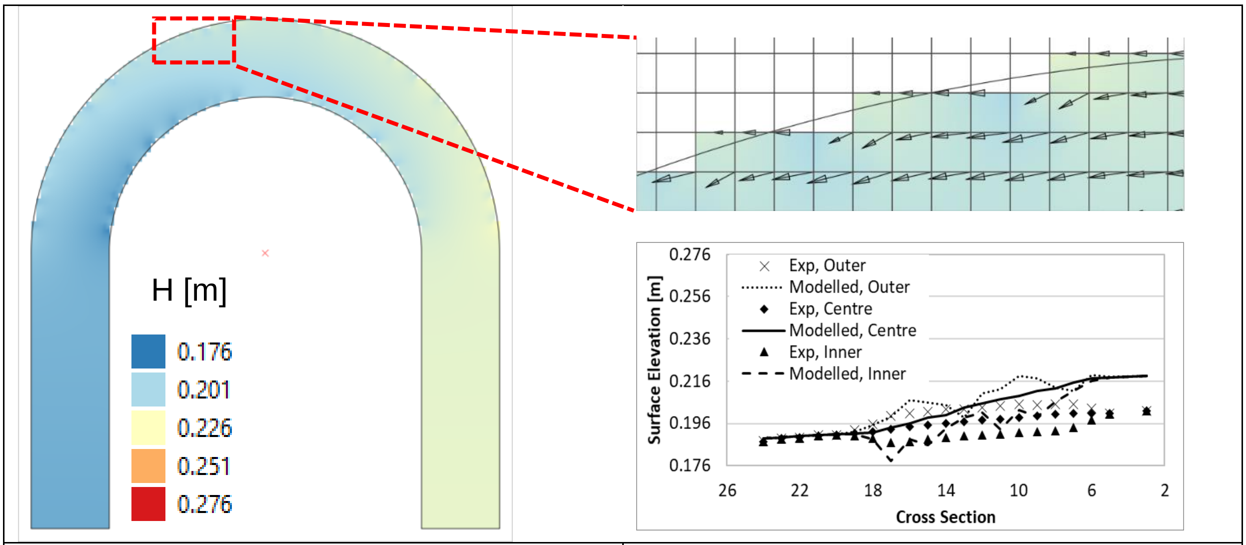


Figure 6 U-Bend Flume

Sub-Grid Sampling Approach Result

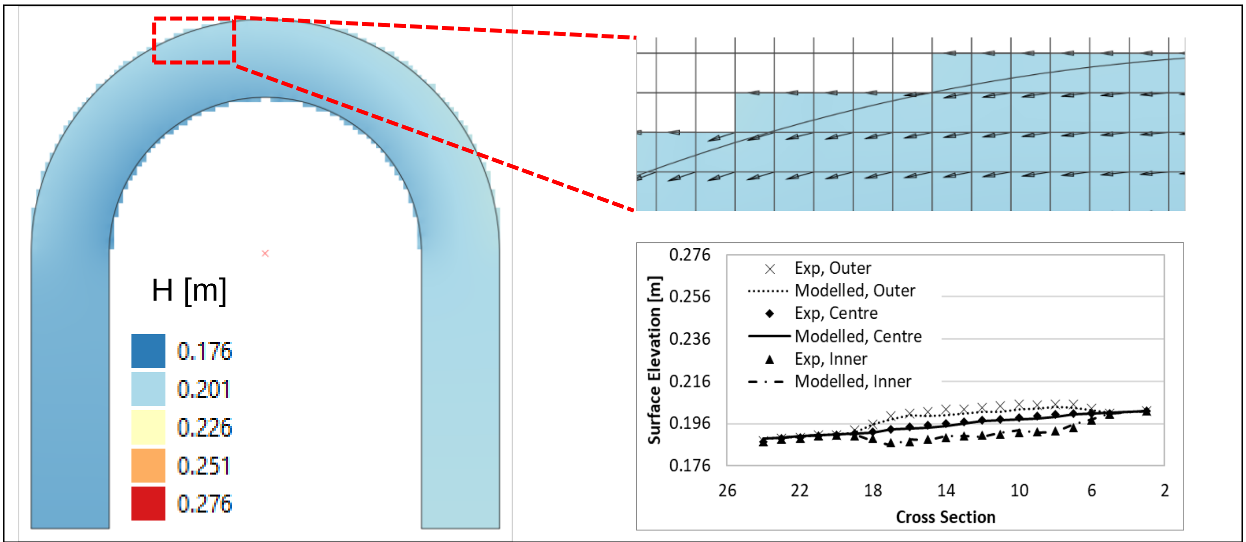
Figure 8 presents the results for the SGS approach. The impact of the saw-tooth wet-dry boundary around the U-Bend is non-existent due to the wet-dry boundary being represented by partially wet cells and flows conveyed through partially wet cell faces. This in turn results in a smooth flow field and modelled water level results that are in good agreement with the measured data.

These results further support the positive SGS findings demonstrated by the rectangular channel testing in Section 2.1.1.



(a) Mapped water surface elevation	(b) Simulated vs Observed Results

Figure 7 U-Bend Flume Benchmark Result: Traditional Single Elevation Per Cell Approach



(a) Mapped water surface elevation	(b) Simulated vs Observed Results

Figure 8 U-Bend Flume Benchmark Result: Sub-Grid Sampling Approach

2.2. Cell Size Independent Sub-Grid Turbulence Scheme

As technology and data collection methods have improved over the past 5-10 years 2D model cell sizes have reduced as practitioners attempt to obtain higher resolution and more accurate results. This is particularly the case in urban stormwater situations. The representation of sub-grid-scale turbulence (often referred to as eddy viscosity) in the 2D shallow water equations (SWE) has been an increasingly concerning issue in 2D modelling as cell sizes have reduced.

It is well known that as element size reduces, the traditional and commonly used Smagorinsky eddy viscosity approach becomes invalid. The Smagorinsky approach, intended for large eddy simulation scales in coastal models, fails because it is proportional to element surface area and therefore tends to a zero-turbulence state as the element size reduces. The deficiencies of using Smagorinsky, especially once the cell size is smaller than the depth, has historically been mitigated in TUFLOW by using an additional constant component. The default setting for calculation of eddy viscosity in

TUFLOW has been to calculate the turbulence component as the addition of the calculated Smagorinsky value and a constant value (rather than one or the other). The inclusion of the constant component ensures some turbulence is accounted for as cell sizes become very small. However, our research and benchmarking over the last two years has shown that the constant coefficient value is highly dependent on model cell size, varying by several orders of magnitude from flume scale to large river scale. This constraint makes it difficult to recommend a default value for the constant component as the value is cell size dependent.
As 2D solvers of any persuasion are increasingly expected to model smaller and smaller cell sizes, it has been increasingly important to develop a cell size independent approach to sub-grid turbulence averaged in the vertical for 2D schemes. This issue becomes particularly critical for models that use a mesh with varying cell sizes (i.e. Quadtree – Section 2.3).

To solve this issue, Greg Collecutt and Shuang Gao from our team completed an extensive two year research project investigating sub-grid turbulence across a wide range of model cell sizes / scales. The aim of their work was to identify a SWE compatible scheme that is cell size independent. The following sections briefly summarise some of their research and testing. It also states the implications for urban stormwater modelling.

2.2.1. Benchmark Tests

Three benchmark scenarios were chosen for the sub-grid turbulence scheme testing. Each included high quality recorded datasets. Importantly all test scenarios contain significant energy losses due to sudden changes in flow direction and velocity. They also span a wide range of spatial scale requiring a large range in cell size.

1. **Angled Flume Bend (Malone and Parr, 2008)**
   Malone and Parr (2008) investigated the head losses associated with flow around sharp bends in a rectangular channel. Their test was at laboratory scale with a flume width of 150mm and a comparable flow depth. Figure 9 shows flume details and recorded head loss results for each configuration as a function of upstream velocity head (computed from the tabulated data in their report).

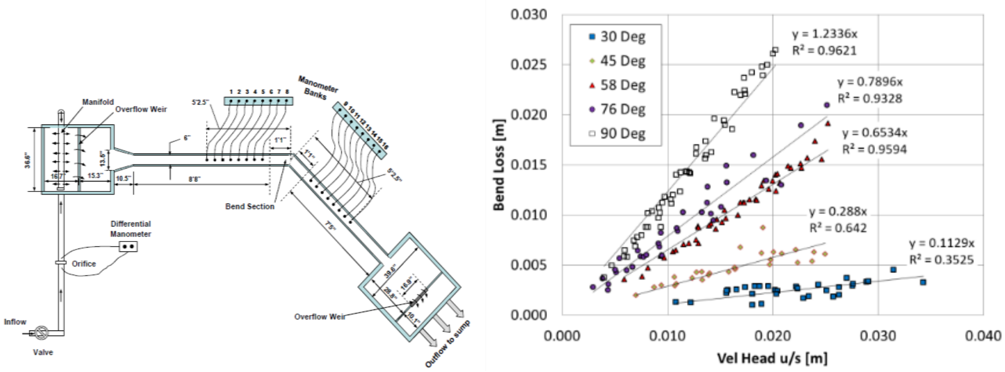


Figure 9 Angled Flume Bend

2. **Dam Breach (Soares-Frazão & Zech 2002)**
   Soares-Frazão & Zech (2002) completed testing of a dam break scenario in a 3.6m wide flume, shown in Figure 10. An oblique obstruction, representative of a building, was placed 3.44m downstream of the dam failure gate. The channel was constructed of smooth concrete and reported to have a Manning's bed friction coefficient of 0.01. Water level and velocity information was recorded at six locations, five surrounding the obstruction and one in the upstream reservoir. Regions of super-critical flow, sub-critical flow and a hydraulic jump were observed in the measured data.

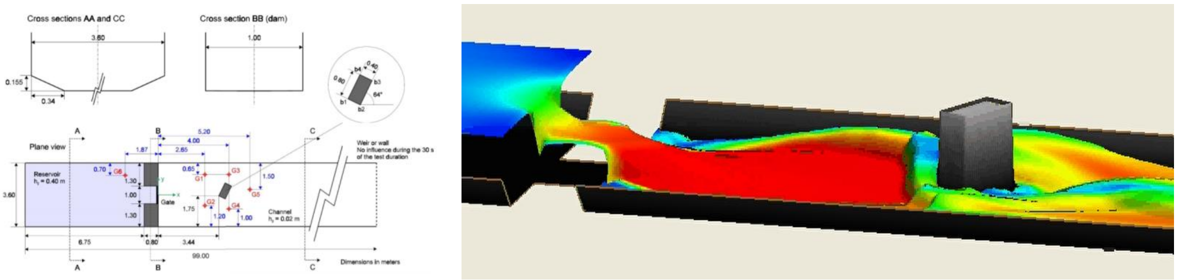


Figure 10 Dam Breach Flume

3. **Brisbane River Flood**
   The third benchmark scenario is a real-world scale test. The Brisbane River (Australia) is several hundred metres wide. During the 2011 flood event it was running at approximately bank-full capacity. The volume flowrate at the flood peak was derived by ADCP current profiling at approximately 9,000 m^3/s.

   Numerous water levels were measured at along the rivers length during the event. The high volume of recorded data associated with this flood make it a suitable dataset for model validation. An 11km length of the river spanning the centre of Brisbane City was used for this test. This section of the river was selected due to the presence of two near 180-degree bends making it ideal or turbulence tests. The head loss along this section of river was measured to be 4.0m at the peak of the flood (for a downstream level of 2.7m AHD).

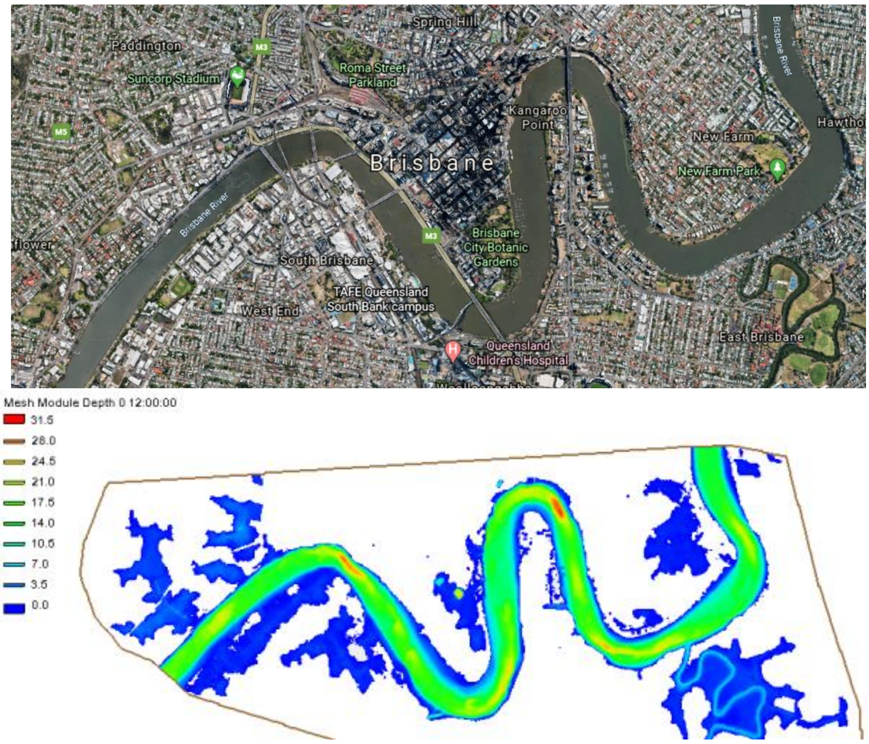


Figure 11 Brisbane River

2.2.2. Methodology and Results

The above three test scenarios were modelled using five different sub-grid turbulence schemes: Smagorinski, Constant, Wu 2D, Wu 3D and Prandtl. Note, other commonly used schemes in CFD modelling were also investigated (such and k-omega and k-epsilon) though found to be unsuitable for 2D SWE schemes.

The research adopted a three-step process:

1. The above listed test scenarios were modelled using the five different sub-grid turbulence schemes. Sub-grid turbulence coefficients associated with each scheme were adjusted to achieve a model result matching the recorded data. Example results from the model testing for the angled flume bend test are shown in Figure 12.
2. The optimum sub-grid turbulence coefficient values for all test scenarios were tabulated, shown in Table 1.
3. The tabulated results were reviewed to identify a sub-grid turbulence scheme capable of producing accurate results across all spatial scales using the smallest possible range in the sub-grid turbulence coefficient value (a scale independent sub-grid turbulence scheme)

Table 1 provides a summary of the turbulence testing results. As an outcome of this research Wu 3D has been selected as the sub-grid turbulence scheme for TUFLOW Heavily Parallelised Compute (HPC) engine.

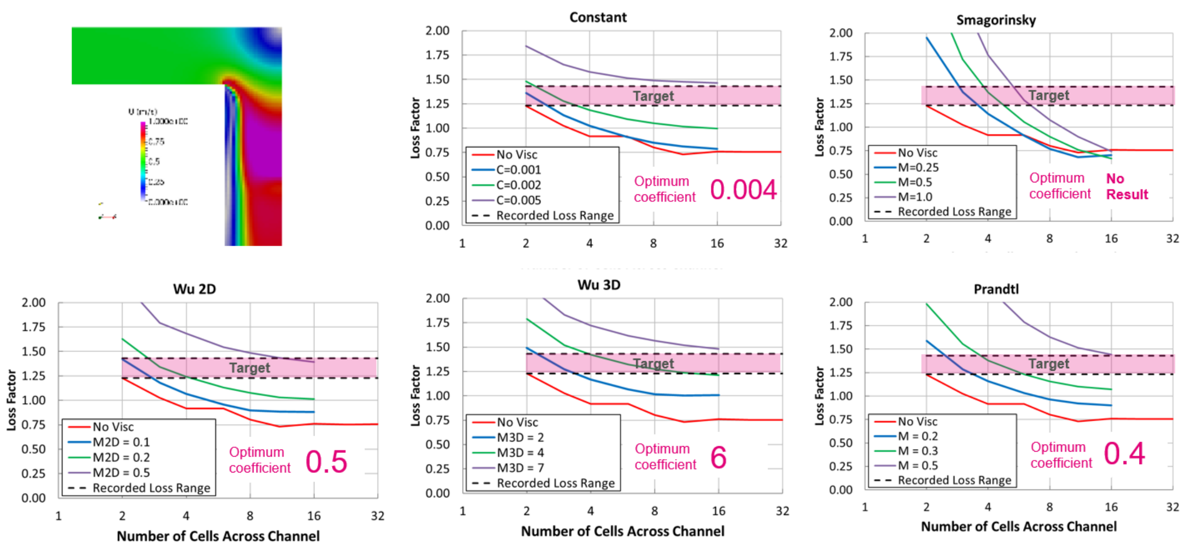


Figure 12 Result Example: 90º Angle Flume Bend Test

Table 1 Turbulence Test Result Summary

Test	Turbulence Model				
	Smagorinski	**Constant**	**Wu 2D**	**Wu 3D**	**Prandtl**
Angled Flume Bend (centimetre scale)	No optimum result	0.004	0.5	6	0.4
Dam Breach (metre scale)	No optimum result	0.01	0.5	3	0.5
Brisbane River (100s of metre scale)	No optimum result	10	4	7	1.0
Commentary	At small cell sizes relative to the water depth this scheme reduces to 0 loss. This is not ideal.	The large range in coefficient value across the range of model scales is not ideal. It is not possible to define a one size fits all default coefficient value for all cell sizes.		This scheme achieves a good result with minor variation in coefficient required. This is the preferred scheme and has been selected as the new TUFLOW HPC default	This scheme achieves a good result with minor variation in coefficient required. It does however have a larger memory footprint compared to Wu 3D

As shown in Table 1, the Wu 3D turbulence model meets the objective of the cell size independent turbulence testing. As such it has been selected and the new default for the 2020 TUFLOW release. Using it TUFLOW modellers can now model at all scales from sub-centimetre cells for a flume to tens of metres for a large river using the same default turbulence parameters with confidence. They can vary cell size downwards without seeing significant changes in results due to limitations associated with turbulence scheme assumptions, especially where the flows are complex, and cell sizes are less than flow depths. This is relevant for stormwater modelling situations where there is a need for smaller cell sizes and where engineered drainage systems often create hydraulically challenging and sometimes highly turbulent situations. Further to this, this new world-first 2D SWE development will allow modellers to vary cell sizes within a single model using quadtree mesh refinement (see Section 2.3) without any cell size dependencies. To our knowledge, no other SWE software has been able to achieve this.

2.3. Quadtree Mesh Refinement

Another major TUFLOW development for 2020 that is relevant to stormwater modelling is quadtree. A quadtree mesh is constructed by dividing a cell into four cells, with these cells able to be divided into four, and so on, allowing modellers to use larger cells in areas of flat terrain (e.g. large flat floodplains, parks) and smaller cells where the terrain is variable or along primary flow paths (e.g. river channels, road gutters, open channels). Of the three features discussed in this paper this is perhaps the most intuitive. The benefits of quadtree include:

1. Improved hydraulic computational delineation where most needed,
2. Smaller memory footprint on the GPU card, and
3. Often a reduced total cell count, typically leading to faster simulations by a factor of 2 to 5.

Figure 13 provides a demonstration of a quadtree mesh in an urban setting. Four levels of nesting have been applied to scale down from a 16m cell size to 2m within the road easements and main stormwater drainage channels.

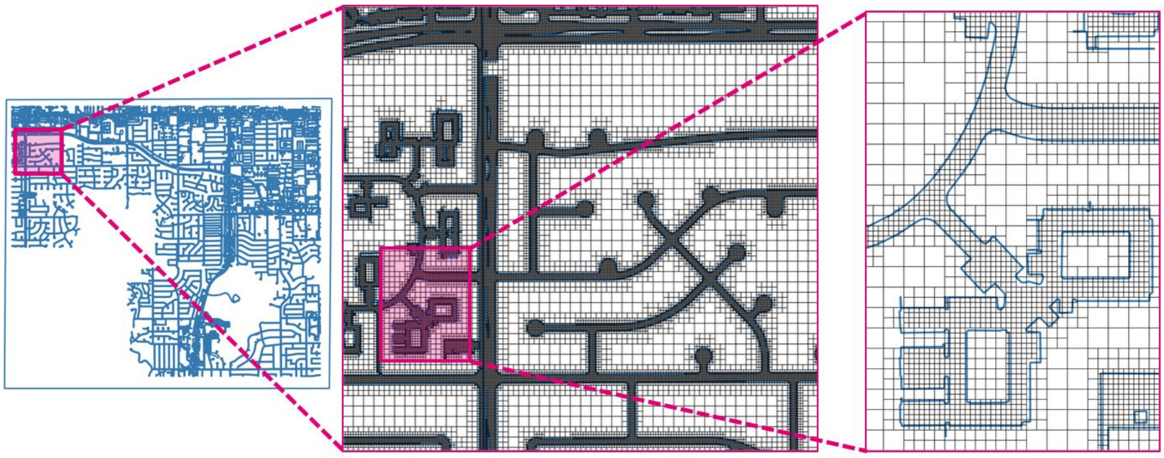


Figure 13 TUFLOW Quadtree Mesh

The implementation of quadtree takes minutes from a single domain fixed grid model. In its simplest form, such as the example above, a GIS file containing regions with the desired level of nesting is added to the model. In an urban stormwater setting road easement GIS files are perfect for this task. TUFLOW applies the finest level nesting (2m or Level 4 in the above example) and automatically graduates out to the coarsest parent level (16m or Level 1 above). All other boundary and topography inputs are allocated to the quadtree mesh automatically without the need for any manual intervention or manipulation.

2.3.1. Quadtree Testing

Extensive hydraulic benchmarking and testing has been completed. The results are consistent with TUFLOW HPC and were previously presented at the TUFLOW release national workshop tour. They have not been reproduced here. More interesting to the stormwater practitioner is testing to identify how the quadtree influences simulation speed.

A 1D/2D direct rainfall model of Innisfail, QLD using a 12-hour ARR1987 rainfall event has been used for this comparison. This assessment builds on previously published research by Huxley (2017). Quadtree has been added to the previous work. The model extent is shown in Figure 14. The quadtree version of the model was implemented using three levels of nesting. The council road easement GIS dataset was used to define the finest level nesting (shown below). The model was run using a range of cell sizes and two mesh approaches to assess the impact these assumptions have on simulation time. Results are summarised in Table 2.

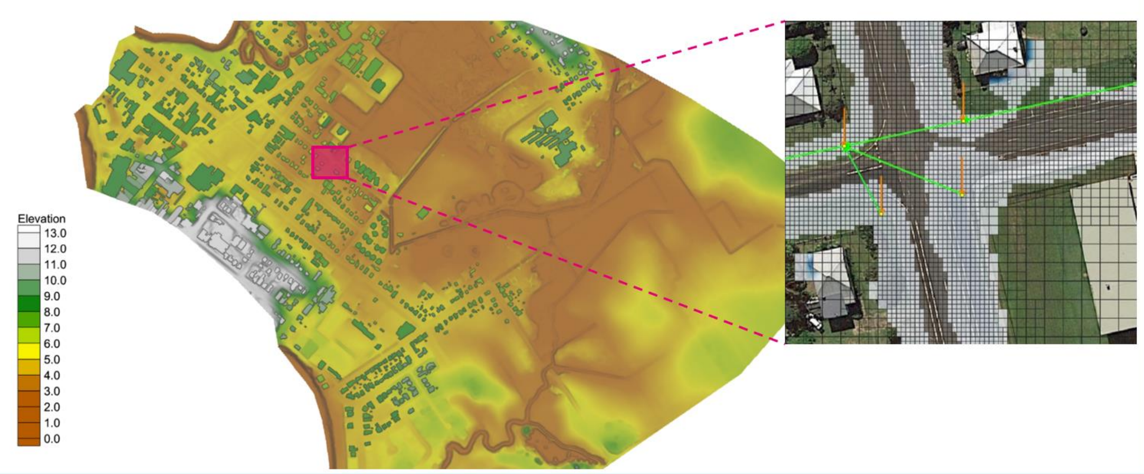


Figure 14 Quadtree Test Model

Table 2 Quadtree Result Summary

Single domain model			Quadtree mesh			Simulation speed-up (Quadtree / single domain)
Cell size	**Cell count**	**Simulation time (hh:mm)**	**Cell size**	**Cell count**	**Simulation time (hh:mm)**	
5m	125,000	0:04	20m / 10m / 5m	27,000	0:03	1.3
2m	750,000	0:18	10m / 5m / 2.5m	109,000	0:05	3.6
1m	3,100,000	1:47	5m / 2.5m 1.25m	407,000	0:26	4.1
0.5m	12,500,000	13:18	2m / 1m / 0.5m	2,540,000	3:47	3.5
GPU card used during testing = NVIDIA TITAN Xp						

The test results highlight a few key findings:

- The upgrade from Central Processing Unit (CPU) to Graphics Processing Unit (GPU) for the single domain version of the model translates to faster simulation times. The speed-up ranges from approximately 20 times faster for the 5m cell resolution model to over 80 times faster for the 0.5m cell resolution model.
- Quadtree mesh refinement using three levels of nesting reduced the number of cells with the model by approximately 80%.
- Quadtree mesh refinement using three levels of nesting to achieve comparable modelled extents and peak levels reduced the simulation runtime significantly. The quadtree models ran on average 3 to 4 times faster than the single domain model using the finest resolution globally across a single domain.
- The ratio between the simulation speed-up and cell count reductions are not one-to-one. On a like for like basis the quadtree solver is slower than the HPC solver (e.g. The 1m model reduces the number of cells by 87% though only reduces the simulation time by 76%). Irrespective of that, intelligent mesh design will achieve the simulation speed benefits shown in this paper. The key message on this topic is to start by identifying the target fine resolution portions of a model and work upwards from there. This is preferable to indiscriminately

working down from an existing single domain model and the probable outcome will most likely translate to a model with more cells than necessary.

3. CONCLUSION

1D/2D stormwater modelling tools have progressively evolved over the past three decades. New technology developments released this year by TUFLOW include:

- Sub-grid topography sampling
- Cell size independent sub-grid turbulence scheme
- Quadtree mesh refinement

This suite represents one of the most ground-breaking advancements in recent times. The new technology will redefine stormwater modelling. The substantial benefits relating to simulation speed and increased result accuracy have significant implications for the stormwater industry. They will translate to improvements in the accuracy of hydraulic assessments for future infrastructure design, flood impact assessments, flood studies, climate change adaptation work and flood risk management planning.

4. ACKNOWLEDGMENTS

This work was conducted in collaboration with Greg Collecutt, Shuang Gao, Bill Syme and Phillip Ryan, all of whom are senior members of the TUFLOW Software Team.

5. REFERENCES

Australian Rainfall and Runoff (2012), *Project 15, Two Dimensional Modelling in Urban and Rural Floodplains, Stage 1&2 Report*, P15/S1/009, November 2012,

De Vriend, H.J. and Koch, F.G., (1978) *Flow of water in a curved open channel*. Delft Hydraulics Laboratory/Delft University of Technology, TOW-Report, R657, VII/M141S-III.

Hardy, R. J., Bates, P. D., and Anderson, M. G., (1999), *The importance of spatial resolution in hydraulic models for floodplain environments*, Journal of Hydrology, 216, 124-136.

Huxley, C., Syme, W., (2016) *TUFLOW GPU – Best Practice Advice for Hydrologic and Hydraulic Model Simulations.* Hydrology and Water Resources Symposium 2016, Queenstown.

Huxley, C., Syme, W., (2017) *Rapid and Accurate Stormwater Drainage Assessments Using GPU Technology.* Stormwater Queensland Conference, Brisbane.

Lane, S., (1998), *Hydraulic modelling in hydrology and geomorphology: A review of high-resolution approaches*, Hydrological Processes, 12(8), 1131-1150.

Malone T., Parr, D., (2008) *Bend Losses in Rectangular Culverts.* Kansas Department of Transport

Soares-Frazão, S. and Zech, Y. (2002). *Dambreak flow experiment: the isolated building test case. Technical Report*, WP3 Flood Propagation, IMPACT (Investigation of Extreme Flood Processes and Uncertainty), EC Research Project No. EVG1CT2001-0037.