Webinar: The importance of 2D cell size for accurate hydraulic modelling
Wednesday, 18 November, 2020
Presenters: Chris Huxley, Bill Syme

Question	Answer(s)
Question: Raised that 0.5m grid cell size needed to accurately predict velicity and hence hazard. How do you get a large area modelled with a reasonable accuracy.	Answer(s): Using a Quadtree mesh in combination with Sub-Grid Sampling (SGS) will very much allow you to have your 0.5m cell size in the area of interest or where there are large changes in topography causing changes in hazard (in areas of little change in topography/roughness (eg. rural floodplain) you will be able to use much larger cells and reduce memory and run times. Access to high end GPU cards will also allow you to realistically turnover simulations for very large models.
with remapping the results (levels/depths) for Tuflow modelling using SGS, the change to the grid size (e.g. 4m grid size to 1m SGS sample size) causes some discontinuity in the flood extent. any advice? which results do you use for afflux? (remapped or the original results)	This may occur where you have significant topography changes within the 2D cell as you are taking a single computed water level per cell and interpolating over a finer resolution. For more explanation see https://wiki.tuflow.com/index.php?title=TUFLOW_Remapping I would focus in the 4m afflux output as this ties in with the 4m computational grid. However, if needed, you may need to reduce the cell size around the area of afflux assessment using a Quadtree mesh. Thank you Bill, I found it necessary to compare the remapped grids with the modelling grids and make sure that some wet cells are not removed by the remapping tool.
are the different grid sizes resampled from the same grid?	Yes. You feed the same DEM and other topographic data sets (TINs, breaklines, etc) regardless of cell size. To change the cell size is simply changing one number - there is no need to rework or copy any of the input data sets.
The channel bottom seems to be flat and smooth, is n't this could be one of the reasons that you could go upto 100 m grid cell?	Yes, relative to the surrounding hills the floodplain where the flood progresses is flat, but i does vary. We see the same effect with more variable topography, but clearly there becomes a limit as to how big your cell size can be!
Does the difference in cell directions and river flow direction effect the convergence result.	Not with Sub-Grid Sampling (SGS) on. We're seeing amazing convergence for a much large range of cell sizes. There does become a limit as Chris will explain with the Brisbane River model at the end of the presentation. You can also find more benchmarking tests about the grid rotation sensitively using SGS from one of our recent publications: https://www.tuflow.com/media/5022/2020-mesh-orientation-and-cell-size-sensitivity-in-2d-swe-solvers-kitts-et-al-iahr-river-flow-delft.pdf
HI Bill! do you have any recommendations on the required accuracy for convergence? Plus or minus delta WL? Q? Timing?	This will depend on the purpose and scale of the model. For a high level regional planning model that is not being used to set flood planning levels and hazards, maybe within 10% of the maximum depths in the main flow paths. But for a detailed flood planning model maybe only a few% of the maximum flow depth. The inaccuracies associated with hydrologic inflows and other inputs are often a much bigger cause for uncertainty. The other big advantage is that all your initial simulations during model development can be carried out quickly using a coarse resolution in the knowledge that for final production simulations you can use a finer resolution without an unacceptable change in results.
Do you have any example with online DEM (SRTM 30m or ASTER 30m)	Yes, we've seen similar outcomes using SRTM. But, the issue with these datasets is their vertical accuracy which can make your results inaccurate.

Question	Answer(s)
i'm sorry to ask question out of the topic. why when i'm choosing different computation interval it's given different result that very significant?	That shouldn't happen. Are you using TUFLOW? TUFLOW HPC uses a 4th order time integration which won't unacceptably change results with changes in timestepping. TUFLOW Classic is similar. It also could be an indication of instability. For example, if a model aleady has instability at 1D/2D boundary cells, the maximun water level caused by the flactuation can be influences by small changes in simulation timesteps. It might be worthwhile to output the time serise of the modelling results at the location where the difference happened with small output inverval to check if there is any instability.
what is the source elevation model for urban terrain with buildings? How was such detailed DTM generated?	*live answered*
Why the number of cells were just halfed when the cell size was doubled? (i.e. in 3m to 6m example)	*live answered*
My question - what is the standard on accuracy for convergence	Please see previous ansered question.
Hi Chris, in the second case did you use SGS?	live answered
I find it is different standard when applied to urban versus rural flooding (where accuracy is not that much needed)- so are any guidelines to assist us modellers	Yes, you will definitely benefit much more from SGS in urban areas as urban areas have lots of sudden changes in topography. You'll also need, as a rule, a finer cell size for urban areas over rural.
Isn't it necessary to use less than 2m cell size for urban modelling?	Not necessarily. Excellent, nicely calibrated urban models of up to 5 or even 10m cell sizes have been produced. SGS will greatly assist with justifying a large cell size.
Regarding the Brisbane Rive model, did you test 2 m and 5 m grid sizes with SGS?	*live answered*
Hi, I am wondering if you have experience in grid resolution for intertidal floodplains? Or flooding levels much less than 1 meter. In the dam failure example, the flood water levels are over a meter, hence a coarser resolution can achieve connectivity.	Yes, the effect is consistently the same.
SGS gives lower flood levels than flat cell approach. Would it be an issue to rerun a calibrated flat cell model in SGS?	Yes, if there is substantial topographic variations within your cells and you ahve too caorse a cell size you'll possibly see a significant change in results and the model may need recalibrating.
Hi Bill, thank you for your answer. So, we have almost 10 online DEMs available. How can I chose the grid size. For example, if I am using 30m DEM then is it necessary to provide 30m x 30m grid size?	No, the mdoel cell size is independent of the DEM size. But there is little point in using a 2D cell size smaller than the DEM size. Always use the finest and most accurate DEM if you have a choice.
for riverine model why we need to go for high resolution where I dont have any dam breaching effect?	Hi Vimal - can you please provide more details.
In Case 3 that you used SGS, what was the sub grid sampling cell size?	*live answered*

Question	Answer(s)
I've done modeling on the Rio Grande River in Texas, US. Generally, the floodplain is 2.5 miles wide. The low flow channel is approximately 500 ft wide. It migrates considerably. The existing LIDAR data is at 1 m resolution. I modeled scale bollards in the floodplain that are 6 ft x 6 ft (actual size is 6 in by 6in). These bollards have a spacing of 8.5 ft between them. I use a cel size of 25 ft x 25 ft in the floodplain and reduce down to 2 ft x 2 ft when I get to the bollards but, naturally, this resulted in a large number of cells. Correct? I should have mentioned that Depth is up to 50 ft in the channel and 14-16 ft in the overbank area. Perhaps the question should have been: "Would using SGS be a better way to model this?	*live answered*
Chris, Something you aluded to during your presentation - when you are dealing with very deep flow. Is there any limit to cell size with respect to the flow depth? Is there a specific point at which the Shallow Water Equations will not apply?	*live answered*
Thanks Bill. I think its really important to provide some guidance on 'accuracy'. I read a lot of reports with qualitative descriptors of model results 'excellent', 'looks good' etc. with no evidence to back up the description. My view is that model comparisons need some sort of benchmark for comparison.	Absolutely agree - benchmarking and demonstration of cell size convergence are really important, especially if no calibration has been carried out. BTW, nice appearance on "Big Weather" :)
Can TuFLOW used to model river channel morphology.? If yes what should be cell size to be used?	Our 2D/3D flexible mesh solver, TUFLOW FV, does this nicely. During 2021 we plan to build in TUFLOW FV's sediment transport and morphologic module into TUFLOW HPC/Classic.
Hi Chris, whats the impact of solver ie FEM vs FVM vs VoF on cell size in hydraulic modelling?	*live answered*
What presentation is Chris referring to that Greg gave last month?	https://www.youtube.com/watch?v=gbd7XZ88gck
Apart from calculation speed – How does Tuflow Results compare to HEC-RAS2D results? Have cell size changes comparrisions been done between the two software packages?	*live answered*
Bill- what are your recommendations on what models to use for 2D hydraulic modelling for rural areas?	I expected that- TUFLOW is the best live answered
Whats the role of numerical solver (numerical diffusion) in the case high resolution model? when I use diffusion wave approximation, water levels are over estimated , rather getting a low values. Is there any impact from resolution of grid? Please comment on this ?	Never ever use diffusion wave unless the water is moving very slowly and steadily everywhere within the model. The diffusive wave equation is missing several key physical terms.
Could TUFLOW used to model the Dam Break Analysis and determined the downstream cross-section?	live answered Yes, absolutely. There is also now a non-Newtonian option if needed.
and is it can calculate the flood arrival time?	live answered
Would an upper limit to cell size be determined by timestep and courant?	Not for explicit schemes like TUFLOW HPC - the timestepping is variable and is controlled by three numbers (Courant, Celerity and Diffusion). For TUFLOW Classic, an implicit scheme, yes there would be an uppler limit determined by timestep and courant.
What is a recommended grid size to accurately model flows within the urban road network, given that they comprise the main surface flood flow path in urban areas?	*live answered*

Question	Answer(s)
Has Tuflow has been applied for flood forecasting operational models	*live answered*
Questions from a novice: What's the difference between TUFLOW classic and HPC? And what does HPC stand for?	HPC - Heavily Parallelised Computations (plus a play on High Performance Computing). HPC uses a 2nd order spatial and 4th order time explicit finite volume solution. Classic is an ADI (alternating direction implicit) semi-implicit solution using matrices. Very different mathematics for the same equations!
For Moreton Bay what should be mesh size?	*live answered*
What is the best way to do a convergence test with cell / mesh refinement regions?	*live answered*
The maximum convergence is 3 when we use fullSt.Venant equation (HEC-RAS 2D). What if I see the convergance more than 3 for 10 cells Vs 100 cells (2% from total cells). Should I say my model is unstable?	live answered Are you referring to the Max Courant Number limit of 3? If this is specific to HEC-RAS, there are a number of ways to check stability. If you're referring to Courant Number criteria in the adaptive timesteps, I generally wouldn't exceed 3 - usually I keep it closer to 1 or 2. Feel free to e-mail me at info@surfacewater.biz and we'll sort out any HEC-RAS questions.
For salinity model how to chose dispersion co efficent , e.g in Brisbane River Estuary?	*live answered*
Most of us use online DEMs (SRTM, TANDEMX, GTOPO30). What kind of resolution is really required when vertical accuracies are not so promising?	Depends on the capture technique. Satellite is typically the least accurate, Lidar much more accurate and field survey the most accurate. Also be VERY careful of the effects of vegetation and other obstructions on accuracy. And where there's water you need a hydrographic survey - you can't rely on Lidar or satellite.
Hi Chris, Thanks for answering my question - so previously, when the Smargorinsky Turbulance was applied, you had to keep your cell size to about the same as the flow depth. But now Bill has a new formulation for turbulence - so this doesn't apply? Can you tell me from which Tuflow release this was made? (e.g. from 2017 or 2019, etc....)?	*live answered*
Could Tuflow used to model syphon?	Possibly as a 1D operational or discharge matrix structure - if you email support@tuflow.com with details we can let you know.
if i want velocity profile in vertical direction for sediment calculation, how can I get it with only one cell i.e. 30 m depth cell u are talking about	*live answered*
I have recently been introduced flexible mesh modelling. Is it possible to give some comments between flexible mesh and QuadTree?	*live answered*