Webinar: 1D, 2D and 3D Hydraulic Modelling of Bridges *Wednesday 16th November 2022*

#	Question	Answer
1	The TUFLOW paper "Modelling Bridge Piers and Afflux in TUFLOW" discusses bridge modelling approaches using blockages and/or Form Loss Coefficients. Is it prudent to assume blockage based on piers AND FLCs simultaneously? Presumably that would be a conservative assumption? Cheers!	The answer to this depends on the derivationof the FLC values being used. If the FLC is based on the "structure velocity" then yes, you should include blockage factors (that act to reduce the flow area in the calculations, subsequently increasing the velocity value to better represent the true value through the structure). The FLC presented in the CFD modelling section were based on the structure velocity. If the FLC being used is based on the approach flow velocity, then do not include the blockage factors. This should only be done for FLC derived for a somewhat similar geometry. Note, the pier form loss from Hydraulics of Bridge Waterways (FHA, 1978) is derived based on the background velocity in the absence of piers. Therefore, blockage is not needed if you're using a form loss value from this reference, although it is often included to be conservative.
2	Is it possible to generate a profile plot of the Water Surface Elevation (WSE) withing a culvert passing through a roadway in 2D using TUFLOW? I'd been trying to generate the same in HEC RAS, and it allows me only for 1D modelling, but doesn't shows the location of the hydraulic jump. My aim in such modelling is to get a profile plot of WSE and to be able to locate the location of hydraulic jump within the culvert (if there is one).	In the TUFLOW 1D solver (ESTRY), the water level is stored at the upstream and downstream ends of the culvert. It will report the flow regime of the culvert including whether a hydraulic jump is happening, but it doesn't report the location of the hydraulic jump nor how the water surface changes inside a culvert. To get a result in such detail, it's best to use 2D SWE model or 3D CFD with sufficient resolution. TUFLOW 1D can also resolve hydraulic jumps in the sense that the flow can transition from supercritical to subcritical. To have a better idea of where the jump maybe occurring you'll need to discretise the culvert into a series of short 1D segments.
		Hi Shuang, thanks for your response. the notion of discretising the culvert does interests me. While, it really does takes me to the second part of the 2D HEC RAS model that I've been trying to model the location of hydraulic jump (Resolution of 0.5 m). Still no help. So, if you could provide more information as to what software would be the best in my case? And what resolution would be appropriate.
		In general, you'll need (1) sufficient mesh resolution, (2) appropriate turbulence model, and (3) a second or higher order spatial scheme to reproduce a hydraulic jump. But it really depends on the hydraulic condition of the culvert. Can you please send a query to support@tuflow.com with more details about the model and the flow conditions so that we can further assist?
3	I mostly complete dam breach modelling, where the peak flows are relatively large with a lot of momentum behind them. In my modelling, I've historically chosen the 'easy' path to assume a bridge isn't designed to convey AEP flows anywhere near equivalent to the peaks of my dam break and effectively remove them from my 2D model domain as flow obstructions. Is this a reasonable practice, or should I expend that additional effort to model the bridge within the model?	This really depends on how critical your bridge affuxes are for your investigation. If a bridge's afflux has the potential to produce meaningfully different results in your area of interest, it might be worth including the bridge. However, if the results are not overly sensitive to the bridge, your approach is fine.
		The necessary model input depends on the scale of the feature relative to the flows and the objective of the modelling. In some cases, bridges will have a significant impact on flow, in others it may not. You will need to use your judgement regarding what is necessary. Adding bridges of a significant size via the addition of form losses is a relatively easy exercise. Because of this, if I were responsible for the modelling I would define a structure size above which I would represent the features in the model. I would make sure to report the data exclusions in my reporting so my assumption are clear.
		As a general comment, if your depth over the top of the bridge is greater than the depth of flow under the bridge (highly likely for a dambreak flow!) then the bridge is likely to be drowned out at the peak and it is fine to exclude. However, if the bridge crossing is a signficant blockage to flow at the peak then it would be of benefit to include.
		I would suggest that if a bridge is over-topped by a factor of 2 or more (i.e. water depth is more than 2x height to the top of the guard rails) then it probably will have only a small impact on results. But to be safe (and defendable) I would recommend including it or demonstrate through desktop calcs or sensitivity testing in the model whether it can be excluded.
4	What conservative assumptions should I make when I do not know the pier layout/design of the bridge I want to model in TUFLOW?	I would suggest that you make a conservative assumption about your bridge design/layout and then assign form loss coefficients to that design in accordance with available guidelines.
		Conservative is a relative term. Applying form losses that are too small will be conservative downstream, not upstream of the bridge. The opposite is true if form losses greater than reality are applied (conservative upstream, not downstream). Whatever assumption you choose, make sure it is reported so future users of the model or study are aware of the data gap.
		Thank you Urs and Chris. I definitely need to strengthen my knowledge about general bridge design.
10	What does the 'interFoam solver' mean or stand for in the 3D modelling slide?	The "interFoam" solver is part of the OpenFoam CFD modelling suite of applications.
11	What baseline FLC values do you recommend to adopt for bridge decks and guard rails in absence of meaningful data to attempt calibration?	Greg will present the recommended value based on the joint study with TMR in the second half of the presentation.
		We suggest you use 0.28 as the peak form loss coefficient if using the new 2d_bg layer in the upcoming TUFLOW release. And document this as an assumption in your report, as it is for a "middle" value of hb/T = 4.
12	I have run the models by enforcing again in the Geometry Data and the results for both the cases are significantly different. My question is what is the correct procedure to create a geometry to get the correct results?	We recommend you represent your embankments using a 2d_zsh geometry modifier and the pier and deck features (3D loss features) via 2d_lfcsh (2D layered flow constriction / form loss) inputs. Use the various 2d_check layers or DEM_Z to confirm the modifications are as intended. Any issues, please email support@tuflow.com.

13	Is it safe to assume that TMR knows what ZTMR is and how it's calculated?	The hazard category request came from a DTMR staff member. The DTMR staff wrote, "we'd like to define a road closure hazard (in accordance with our road drainage manual (TMR July 2015)) as follows …". It looks like the current DTMR drainage manual was published in 2019. We'll have to check if the hazard criteria has changed since 2015. The updated version of the guideline is here: https://www.tmr.qld.gov.au/business-industry/Technical-standards-publications/Road-drainage-manual
		Awesome, thanks - I'll have a flick thru the 2019 manual and check if the classification is updated as per their guidelines. In the meantime I guess I'll show the ZTMR hazard outputs in the deliverable for the time being.
14	The interpretation of the type of pier to obtain the Kp value can be highly interpretative where pier shapes depart from those on the Figure 4.10 for determining Kp. I think its quite vital to have further hydraulic experiments to determine a greater number of pier form loss curves for varying shapes, where so many new bridges built today may have non-standard piers including	Agree, and yes it would be nice for some more research and testing on unusual pier shapes to be carried out to. CFD can potentially play an important role in this space to inform 2D models in the future, as can high-resolution 2D/3D SWE solvers based on our benchmarking of pier losses to flume measurements as presented in the 2021 webinar on structure losses (see https://www.tuflow.com/library/webinars/#structures).
15	Would you please inform line (2d_Ifcsh_L) or polygon (2d_Ifcsh_R) work well for big (bridge width>cell size) bridge?	We normally use polygons (2d_Ifcsh_R) for bridges when bridge width in direction of flow is greater than the cell size. This is also the approach we used for the Gordon Rd bridge modelling Greg is about to present. The main difference of using polygons vs lines is how the form loss value is distributed. TUFLOW 2d_Ifcsh polygon evenly distribute the form loss inside the polygon. However, you may argue that the loss might happen mainly near the front and tail ends of the bridge. I believe this is an area that warrants further investigation.
16	Would BMT ever consider physical modelling to expand the set of pier loss curves available?	We don't have inhouse facilities nor the expertise to carry out physical modelling. However, we are always keen to partner with other organisations to help with on-going R&D such as this.
17	With increasing computational power (and with view to the current slides on 1D struggling with a appropriate representation) - why would a 1D solution ever be preferred over a 2D solution nowadays?	The recommended modelling approach typically depends on the study objectives. For example, we developed a 1D only model of the Brisbane river catchment that could run in 5 minutes back in 2013 for use in a Monte Carlo assessment (>11,000 simulations per assessment). Bridges were necessary in that model as the model was extremely well calibrated to an exhaustive data set. A 2D model was also developed for detained mapping and impact assessments, however, it was totally unsuitable because of the long run times to be used for deriving design flows via the Monte Carlo assessments. For conventional 2D flood studies, we do expect we'll see 2D bridge modelling becoming the dominant assessment approach.
18	What was the scale of the physical model - was it a distorted model with different scales vertical to horizontal ? It is usual with distorted physical models that the model roughness must be substantially increased	Good question. They didn't want to use a distorted scale for that reason. The scale was 1:30 undistorted.
19	TUFLOW model can be used high turbulent river in mountainous region	Yes, it's well suited to that type of application.
		TUFLOW HPC uses a finite volume scheme that is accurate for both subcritical and supercritical flow. Further, the use of sub-grid-sampling has proven to be able to compute excellent flow properties where narrow flow paths are not well resolved by the mesh.
20	For the simplified 2D CFD modelling research - what was the model downstream extent - how many bridge deck widths ?	The downstream length was set to 100m to minimise downstream boundary influences
22	For the Eudlo Creek example, was the 1D model coupled to a 2D floodplain model?	For the TUFLOW model it was coupled to a broad-scale 1D model that extended from well upstream of the site down to the Maroochy River. For the other 2D models, the inflow hydrographs and downstream water level hydrographs were extracted from the coupled model so that all models used the same flow conditions. There was good calibration data for three floods and flow gauging off the bridge for the 1983 flood. So, quite a good calibration data set.
23	How does width impact the loss coefficient?	I assume with width you mean deck thickness? The thicker the deck, the higher the losses. This was shown in the values for different hB/T ratio
		If you mean the width of the bridge in the direction of flow, this is something that needs further research. Preliminary models have indicated the results are not particularly sensitive to bridge width, but keep and ear out for any further research that we do.
24	The maximum K value from the 2D/3D/Field data (about 0.3) is a lot smaller than the 1.5625 bridge deck submergence factor used in TUFLOW 1D for B and BB bridges and derived from experimental data in Hydraulics of Bridge Waterways HDS 1	The 1.5625 is derived from the 0.8 orifice flow contraction recommended by Hydraulics of Bridge Waterways for pressure flow. It is not applicable for downstream controlled submerged deck flow. We have a nice Wiki page on this which support@tuflow.com can help you with.
		Agree with above answer - it was for "Pressure flow case 1" with supercritical flow.
25	Is there some good resources for this sort of modelling online? I often still struggle just getting the TUFLOW plug in working well and knowing what result output types I'm going to need for it later. Maybe I should have been looking at AWS YouTube channel rather than TUFLOW' s?	You can find the TUFLOW tutorial models here: https://wiki.tuflow.com/index.php?title=Tutorial_Introduction We also have TUFLOW eLearning courses, and I recall the "QGIS for TUFLOW" course includes how to use the plugin and it's free. There are also numerous example and demo models available via the Wiki that cover the primary aspects of TUFLOW modelling. You can also always email support@tuflow.com for any questions in regard to the tuts, example and demo models, or even if you're using the Free mode.
26	Future research involving bridges with piers would also be useful. As would research into the losses where piers are aligned / misaligned with downstream piers.	Agree, we plan to do some more research on it in the future.

27	In 2D or 3D, do you input any loss coeffs?	For 2D solvers that use a 2nd order spatial solution, include the inertia and turbulence terms, and have been benchmarked to demonstrate their adherence to flume and real-world measurements, these solvers will inherently model most losses associated with flow formation changes such as through a bridge. Sometimes small amounts of additional form losses might be applied to respresent the form losses not well resolved, such as small eddies and changes in the vertical. Losses in the vertical can be inherently modelled in good 3D solvers. Simpler 2D solvers should not be used as bridges are a complex 3D hydraulic problem that they are not suited to.
28	Could you discuss how to handle the ineffective flow areas upstream and downstream of the bridge, specially when the bridge overtops for 1D and 2D?	For when the bridge and embankments are not overtopped, if modelling in 1D then in order to get a better estimation of the approach and departure flow velocities for adjusting the bridge contraction and expansion losses the Effective Area approach should be adopted (see TUFLOW manual Section 5.10.4) as using A_e produces the mainstream velocity. Alternatively, the ineffective areas of your cross-section can be removed from the conveyance calculations by using a negative M (material) ID - the storage of the ineffective areas are still used for the mass balance calculations. For 2D, there is no need to specify ineffective areas and this inherently modelled by a 2D solver. For higher flows where the bridge/embankments are overtopped, it would be awkward to do this in 1D (possible but difficult if trying to have the same cross-sections for both scenarios). In this case 2D would be much more preferable and will do a much better job of computing the correct flow velocity at the bridge without any need for the user to define ineffective flow areas.
29	Can you explain key difference between upstream-controlled and downstream-controlled?	Downstream control means a change in d/s water level will cause a change in u/s water level. Upstream control means the upstream water level is insensitive to the downstream water level and usually indicates the occurrence of supercritical flow.
30	This has been very informative. Will there be a paper summarising this 2D/3D/CFD bridge deck study (is this what Greg is presenting in a few weeks at HWRS in Brisbane)?	Yes. And it will be published in the proceedings and we'll host on www.tuflow.com once publicly available.
31	Hi there was a mention on DTMR wanting to adopt this model, which model was it in particular? was it all the 1D, 2D, 3-D model? was it the 3-D model?	The findings from this research is to provide better guidelines for 2D modelling of bridge decks. The 2022 release of TUFLOW includes a new 2d_bridge layer that is based on these findings so that they can be easily implemented.
32	Is it still considered good practice to carry out a 1D bridge check / calibration (e.g. in HEC RAS) when using a 2D model?	As Bill discussed, 1D models can miss losses caused by changes in direction in the approach flow and/or exit flow and will struggle with situations like the Eudlo Creek example where the flow immediately downstream of the bridge is forced sideways. The best review is to compare the 2D results with any available calibration data, however calibration data is rarely available so comparisons with desktop calculations using Hydraulics of Bridge Waterways is always good practice (bearing in mind it isn't set up for unusual bridge crossings or bridges at tight bends) and comparison with other results from other software is always useful - if they agree, great, if not, then further examination is warranted. If you ever have any observations you'd like to share or questions about your comparisons, please email support@tuflow.com as we're always keen to see these. I think comparisons are less needed these days as our knowledge and 2D modelling has improved greatly, but whether you need to carry out a comparison should be more driven by whether the bridge has a significant influence on the hydraulics in your area of interest. For example, if it's well away from your area of interest it would not be needed.
33	Thanks everyone for the great presentation. I am looking to model the future floods and droughts within the Sydney area. I am struggling to find Future Climate downscaled data for variables such as Temperature, Precipitation, Wind speed and relative humidity or which ever one available preferably for a time period of 2020-2100	SILO may help: https://www.longpaddock.qld.gov.au/silo/