**Webinar: Applied Hydrodynamic Modelling – Part 2**
**Wednesday 21st September 2022**

| Question | Answer |
|---|---|
| In Nepal highly erosive foody nature of stream and river, we have a lack of bedload and soil erosion data only few rivers have daily suspended data. In the limited data condition how can we select the best sediment model from your suggestion. | This will depend on whether the system is sediment limited or if you can assume full sediment capacity. Let's assume that you have a signficant amount of sediment. We can use an equilbrium condition such that the amount of sediment entering the model is a function of the flow that passes over the boundary location. You may then want to complete a series of sensitivity tests on this condition to see how the model responds to your assumptions. Likewise you can experiment with differing bed load models/erosion models etc. |
| What's the command to run hydrodynamic model in TUFLOW? | If you need help please send through an email to support@tuflow.com and we can get you up and running. |
| I see you'll be presenting on hydrodynamic modelling to inform on structural design. Can you describe in detail the difference between the NLSWE vs the Navier Stokes equation in terms of results and limitations?<br>CFD is becoming mainstream for structural design with lots of studies available to validate inputs and results (eg...US FHA have been publishing work in this space for ~20 years now). I thought the NLSWE is inadequate from a numeric perspective for informing on pressure gradients and lifting forces which are important considerations when assessing structures. | live answered |
| Please correct floods in the place of foody | |
| Thank you | |
| May I request the primary download link of TUFLOW model | The model that Mitch and Zac are speaking about today is TUFLOW FV. You can downoad via https://www.tuflow.com/downloads/. However, if you're interested in TUFLOW please send us an email via support@tuflow.com and we can assist you with some getting started steps, example models and free tutorials etc. |
| What is total model area? looks like the cell size is quite large even for the detailed model. The the bathymetry is quite flat? | The area is very flat. The model is over 100km long. I'll open this question at the end so that Mitchell Baum can further discuss. |
| I'm highly interested in the "salt mass retention on dry cells" method to produce a more realistic estimation of the salinity over time. Have you published any paper on this method showing the assumptions and calibration results Mitchell showed during the presentation? | Thanks for your interest. We haven't published a paper on these updates to the TUFLOW FV code yet but it'd be great to get this work out there. These code developments should make their way into an upcoming release of TUFLOW FV. |
| For the Coorong model, is the model mostly run in 2D or 3D? Is there much of a difference between the run time and/or results. | This initial modelling was done in 2D. In a study Mitchell Baum is currently working on they are using 3D. This will be needed to better understand temperature and salinity gradients both horizontally and vertically. I'll also open this up to Mitch B at the end for discussion. |
| It also has a nice brewery right at the bend! | Indeed :) |

**Webinar: Applied Hydrodynamic Modelling – Part 2**
**Wednesday 21st September 2022**

| Question | Answer |
|---|---|
| Interesting helical velocities from the model output, how have you validated these and are they cell size independent? | adding to this, how do you have confidence these results are equivalent to CFD if the shallow water equation solves eddy viscosity and not turbulence?<br>The shallow water equations are accounting for sub-grid scale turbulence (horizontal and vertical) via the use of eddy viscosity models including Smagorinksy, k-epsilon and/k-omega models. We are not stating that these methods will be equivilant to CFD. Rather they capture the flow behaviour sufficiently well for the problem at hand, i.e. forces on a broadscale structure. If you then need to look at very local forces around the piers etc. CFD can be utilised. However, when compared to the forces from the Brisbane River these were deemed to be minor for the purposes of Zac's study. |
| Interesting helical velocities from the model output, how have you validated these and are they cell size independent? | At this bend there's wasn't any detailed velocity measurements to compare to (but are always keen to calibrate where possible). We have extensively validated at lab scale using ubend and riprap models for velocity measurements. If you are interested we'd be happy to provide these to you via support@tuflow.com. For Brisbane River calibration was completed to measured water levels and observed headlosses throughout the Brisbane River. The cell size question for 3D assessments is a good question and worth investigating via sensitivity testing. We expect that we should see cell size convergence based on other testing we've completed. Keep the questions coming they are good :) |
| Thanks Mitchell - i'm pretty sure it is shallow enough (and usually windy enough) to be vertically mixed - though given the high salinity gradients there is a potential for stratification to occur. | Good pickup on the stratification potential with this system. The baroclinicity, particularly moving into the South Lagoon can make for some interesting dynamics during inflow/outflow events. We've looked into this to some extent with 3D simulations, but certainly something to dig into further. |
| Is it a FV or FE scheme? | TUFLOW FV uses a finite volume scheme. |
| These streamlines are so informative! An MRI of the flow! | |
| It looks like the cell sizes sected for the river bend are quite coarse compared to the structure assessed. How were finer eddies captured that would likely impact it? (ie there would be horse-shoe eddies between the piles that would highly influence shear stresses and velocites) | live answered (also see response to CFD for structural assessments below) |
| Bill warned with the release of HPC and SGS that the software is highly forgiving for modeller errors. Is it appropriate to endorse hydrodynamic modelling (with limited validation) for structural design when other tools and methods are available? | The selection of modelling level or appropriate tools for design is always going to be a function of project time and budget. CFD modelling is generally going to be out of range for most structural engineering applications - plus we reasonably avoid this by using the Morison equation (standard in many coastal structure applications) and previously determined and validated drag and added mass coefficients. See DNV-RP-C205 for further discussion |
| Thank you all, that's a very very practical presentation. Is FV for 3d analysis and regular TUFLOW for 2d only? | TUFLOW FV can be either 2D or 3D mode. TUFLOW FV is more geared up for environmental assessments, sediment transport, water quality etc. TUFLOW Classic/HPC is mainly for flooding and urban draining work. |

**Webinar: Applied Hydrodynamic Modelling – Part 2**
**Wednesday 21st September 2022**

| Question | Answer |
|---|---|
| Wait is this modelling not adequate for design purposes? Why is this being presented as an equal to CFD for structural assessments? | The selection of modelling level or appropriate tools for design is always going to be a function of project time and budget. CFD modelling is generally going to be out of range for most structural engineering applications - plus we reasonably avoid this by using the Morison equation (standard in many coastal structure applications) and previously determined and validated drag and added mass coefficients. See DNV-RP-C205 for further discussion<br><br>A non-hydrostatic CFD model will be required for analysing flow-structure interaction e.g. vortex shedding from the piles but that is another level of complexity that will not necessarily be required to develop a robust structural design solution.<br><br>Can a single CFD model represent spatial scales >1000m to effectively simulate the formation of helical flows and also spatial scales <0.1m to simulate flow-structure interaction and at the same time simulate a complete flood event over temporal scales lasting days? As computers get faster we expect CFD simulations at this scale to be possible, however for the scale of this study it is impractical and would likely effect project delivery. |
| Through model, is it possible to represent the spatiality of flow velocity when the flows are superelevated in an outer bend? | Yes, this is what these models are designed to do. The example images that Zac showed had superelevation on the outside of the bend driving a secondary circulation around the bend. This was both horizontal and vertically varing over time. |
| does it possible to change the direction of water and to build?? in river | Apologies, I'm not sure what is meant by this question. |