TUFLOW FV
New Release Notes

Release Build
2023.1
www.tuflow.com

[TUFLOW FV User Manual]
[TUFLOW FV Science Manual]
[TUFLOW FV Wiki]
[TUFLOW Tutorials]
[support@tuflow.com]

Document Updates and Important Notices

January 31st, 2023: TUFLOW FV 2023.0 is a major release that includes the inaugural TUFLOW FV Water Quality (WQ) Module offering plus a range of new functionality, enhancements, and minor bug fixes. For new modelling projects it is recommended to upgrade to Build 2023.0 to make use of this new functionality and improved workflows. For existing projects, while we have provided backward compatibility, it is recommended to assess potential changes by completing result comparisons between previous and new builds.

Both Windows and Linux operating system builds are available on the TUFLOW Download Page.

December 1st, 2023: The 2023.1 TUFLOW FV Build is a major release update that includes the addition of pathogen capabilities to the TUFLOW FV Water Quality Module (refer Section 2.6). Also addressed are several minor enhancements and bug fixes. Features and fixes related to the 2023.1 Release are provided in light green.

IMPORTANT: Culvert flux bug fixes and changes to automatic culvert loss adjustment defaults so that they align with TUFLOW Classic/HPC may potentially lead to changes in results if using culverts, particularly in intertidal environments.

Due to these bug fixes and enhancements, it is recommended that all 2023.0 users update to 2023.1.

Note: If running TUFLOW FV on GPU hardware the NVidia drivers may need to be updated. This is due to a change in the CUDA compiler version. If using TUFLOW FV on a NVidia GPU device it is recommended to update the NVidia drivers prior to using the TUFLOW FV 2023.1 Release.

Using the Release Notes and User Manuals

The new features documented within these Release Notes will be progressively added to our latest User and Science Manuals. In the interim we recommend using a combination of these Release Notes and latest manuals as described in Sections 10.1.2 to 10.1.5. Where any conflicts exist, these release notes take priority. If unsure or if you have any queries, please contact support@tuflow.com.

Versions

To run simulations using the Windows or Linux 2023.0 build requires payment of the 2022/2023 annual software maintenance fee and for the TUFLOW licence to have been updated (i.e. via RaC/RaU files).

For tutorial and demo models, no licence is required. For any licensing enquiries please contact sales@tuflow.com, or for general support support@tuflow.com. Use of the TUFLOW software suite is bound by the TUFLOW Products End User Licence Agreement.

Please see also further information regarding changes to our version numbering in Section 1.5.

Table of Contents

1 2023 Release Overview

The major announcement for the 2023 Release (Build 2023.0) is the unveiling of our new state-of-the-art Water Quality (WQ) Module. Also included are significant upgrades and enhancements to hydraulic structures, Imperial/US Customary Units support, Geographical Information System (GIS) integration and pre- and post-processing toolboxes.

1.1 Water Quality Module Overview

The TUFLOW FV Water Quality Module draws directly on world-leading environmental science to support 2D or 3D water quality simulation of lakes, reservoirs, estuaries, freshwater streams, urban waterways and wetlands, the coastal ocean and other environmental waterways.

Environmental water quality constituents including dissolved oxygen, silicate, inorganic and organic nutrients (particulate, dissolved, labile and refractory) and phytoplankton are simulated. The simulated environmental processes captured by the WQ Module broadly include sediment-water and atmosphere-water interface exchanges and internal processing. The latter includes full nutrient cycling and phytoplankton dynamics.

Set up and execution of the WQ Module uses well-established TUFLOW text file style command syntax, which allows for a high degree of customisation of water quality simulations. To support effective set up and execution, the WQ Module also operates within a flexible framework that:

- Integrates seamlessly with TUFLOW FV
- Allows for native simulation in the milligrams per litre (mg/L) units system
- Comes pre-packaged with library defaults and recommended ranges for all water quality parameters
- Produces a detailed log file for each simulation
- Is supported by an online user manual (see Section 2.6) and access to expert support from the TUFLOW team
- Comes packaged with a range of demonstration models that are free to run, including a model (complete with bespoke post processing tools) that demonstrates the mass conservation performance of the WQ Module (see Section 2.8), and
- Draws on the world-leading scientific code base developed at the University of Western Australia

More information regarding the WQ Module is provided in Section 2 of these release notes.

1.2 New Features and Enhancements

The 2023 Release also includes a range of user experience updates and minor enhancements including:

- Hydraulic structure updates
- Enhancements to Imperial/US Customary Units support
- Improved GIS integration including cell inflow, output points and linked zone structure support
- Upgrades to the TUFLOW FV Python and MATLAB Toolboxes
- New tutorial and example models

- Expanded wiki resources
- A new TUFLOW FV interface for Aquaveo SMS.

1.3 Upgrading to the 2023 Release

As always, it is recommended that when switching to a new build with an established model that test runs are carried out and comparisons made between the old and new builds. Easy ways to do this include subtracting simulation maxima, or comparing your preferred sets of statistics and reviewing any differences. If you have any queries on the comparison of output results or require clarification of any of the new features in these release notes, please email support@tuflow.com.

1.4 2023 Release Updates

Software updates to 2023 will be provided as further features become available and will be included in this documentation.

1.4.1 TUFLOW FV 2023.1

The 2023.1 TUFLOW FV Build is a major release update that includes the addition of pathogen capabilities to the TUFLOW FV Water Quality Module. Also addressed are several minor enhancements and bug fixes.

In 2023.1 the ability to simulate any number of pathogen groups (such as E. coli, Cryptosporidium or Enterococci, for example) has been added to the Water Quality Module. Pathogens can be configured to respond to natural mortality, light inactivation and settling. The simulation of attached pathogens is optional. This new enhancement will allow users to examine environmental issues such as the potential downstream environmental impact of wastewater discharges or unplanned overflows of sewage, for example. Interactions of these overflows with potentially sensitive receptors such as swimming and/or other primary contact areas can now be investigated. Refer to Section 2.6 for further details.

Other key updates include:

- Culvert enhancements and fixes (Sections 3.1.6, 3.1.10 and 3.1.11).
- Variable specification via the command line (Section 3.4.3).
- CPU parallel processing for GPU runs (Section 3.5.15).
- Significant upgrades to the TUFLOW FV Python Toolbox (Section 8.1).
- New Python astronomical tidal, ocean circulation and meterological boundary condition generation tools via the GetTools suite, with supporting tutorials (Sections 8.4.2, 8.4.3, and 8.4.4).
- New Example Models (Section 10.3.2).

1.5 Changes to TUFLOW FV 2023 Version Numbering

TUFLOW FV's versioning has been updated to be consistent with semantic versioning standards for Linux and Windows. The adoption of this system will improve planned integration with software repositories for installation with yum, apt-get, win-get, chocolately and other selected package managers for Linux and Windows deployment.

The version numbering system is X.Y.Z, where:

- X is the major version
- Y is the minor version, and
- Z is the patch version

For example, the 2023.0.0 release is the 2023 major version, 0 minor version and 0 patch version.

These release notes are prepared for the 2023 major version. Subsequent updates and enhancements to TUFLOW FV 2023 will use the 2023 major version and will increment the minor version number. For example, the next update of TUFLOW FV will be 2023.1.Z, 2023.2.Z (where Z is the patch number) and so on.

The patch version is a non-negative integer number used by the TUFLOW team internally to assign unique identifiers to each executable we compile. Unless specifically instructed by the TUFLOW team the patch number can be ignored and each version can simply be identified via the major and minor version numbers 2023.0, 2023.1 etc.

If you have any queries or feedback regarding our changes to version numbering please contact support@tuflow.com.

2 The New TUFLOW FV Water Quality Module

2.1 Features

The TUFLOW FV Water Quality (WQ) Module is a flexible module that enables the 2D or 3D Eulerian simulation of water quality processes in lakes, reservoirs, estuaries, freshwater streams, urban waterways and wetlands, the coastal ocean and similar environmental waterways. It draws on the power of the GPU enabled TUFLOW FV hydrodynamic and advection dispersion engines to support simulation of higher order transformative (i.e. non-conservative) environmental processes. Key features of the WQ Module include:

- Seamless integration with TUFLOW FV. The WQ Module is activated via a single command in the TUFLOW FV control file
- Ease of construction. The WQ Module is constructed and controlled via use of simple text based files that use the well-established TUFLOW nomenclature. Namelist (.nml) files are not used
- The ability to simulate in units of mg/L. This eliminates the need for pre- and post-processing of model inputs and outputs, supports an intuitive modelling workflow and allows for easy comparison of model predictions with laboratory measurements
- Access to a library of parameter defaults. This includes automatic access to a library of parameter values, for all parameters that are not user specified. This means that a user does not need to construct complex control files or search the literature for parameter values in order to set up and execute a water quality simulation. For example, a WQ Module simulation can be executed without requiring a user to (at least initially) specify any parameters whatsoever. Once running, a user can progressively overwrite library defaults with tailored parameters based on the review of model predictions. This workflow supports effective and efficient model set up and calibration
- Provision of model log files. Detailed feedback on a water quality simulation construction and simulation commentary is provided via production of a bespoke water quality output (text based) log file at runtime. This mirrors the TUFLOW log file approach and assists in model setup, debugging and calibration. This information also supports an efficient workflow in the execution of model scenarios
- Provision of an interactive online user manual. This manual describes the processes (and their linkages and supporting science) available within the WQ Module, simulation commands and construction requirements, model parameters (together with units, library defaults and typical ranges), model output variables and diagnostics (also with corresponding units) and demonstration of a full WQ Module mass conservation model of all processes. These features are designed to improve the efficiency and effectiveness of the modelling process, including recalibration and scenario execution
- Provision of pre-packaged demonstration models. The WQ Module comes packaged with a range of rapidly executing demonstration (tutorial) models that are free to run (e.g. Tutorial 09), and a model that demonstrates the mass conservation performance of the WQ Module. Both are available on the TUFLOW FV Downloads page. Post-processing tools that can be used in multiple environments are also provided for free with the mass conservation model

- Provision of wiki pages. Water quality modelling can be daunting for new users, so a suite of wiki pages (that is being continually expanded) is available to offer support. An example is the wiki page that describes the use of diagnostic variables in water quality modelling
- Provision of support. TUFLOW's software team supports and maintains the WQ Module in the same manner as other TUFLOW products at support@tuflow.com

2.2 Computed Variables and Simulated Processes

The WQ Module supports simulation of the following environmental water quality computed variables:

- Dissolved oxygen
- Silicate
- Inorganic nitrogen and phosphorus
- Particulate and dissolved, labile and refractory organic carbon, nitrogen and phosphorus
- Phytoplankton

The corresponding environmental processes captured by the WQ Module include

- Sediment-water interface exchange of dissolved oxygen, silicate and dissolved inorganic and organic nutrients
- Atmosphere-water interface exchange of dissolved oxygen and inorganic nutrients, with the latter including both dry (dust) and wet (precipitation) atmospheric deposition
- Inorganic nitrogen nitrification, denitrification, dissimilatory reduction of nitrate and anaerobic oxidation of ammonium
- Inorganic phosphorus adsorption (and settling) and desorption
- Organic matter settling, hydrolysis, mineralisation, activation, photolysis and breakdown, and
- Phytoplankton productivity, respiration, mortality, excretion, exudation and settling. One or more phytoplankton groups can be simulated, each with one of two, user selectable, nutrient storage mechanisms

2.3 Interaction with TUFLOW FV

TUFLOW FV's Hydrodynamic (HD) engine and Advection Dispersion (AD) Module provide the underlying forcing for the WQ Module calculations. Specifically:

- The HD engine computes water surface elevation and two- or three-dimensional water velocity, salinity, temperature and light fields. The temperature, salinity, light and optionally sediment fields are sent directly to the WQ Module for use as state variables in its calculations (e.g. to compute the dependence of phytoplankton growth on temperature and salinity). Velocity fields are sent to the AD Module
- The AD Module uses velocity fields to compute the temporal and spatial evolution of passive tracer fields, where each tracer represents a single water quality computed variable (e.g. dissolved oxygen). The AD Module treats these tracers as passive and conservative, i.e. it performs no transformative calculations on these tracers other than those associated with pure advection and dispersion

Once computed, the AD Module's passive tracer fields are sent to the WQ Module as concentrations. The WQ Module treats these concentrations as computed variables and modifies them on a model cell by cell basis via execution of its non-conservative water quality algorithms (which can also include dependence on temperature, salinity and light fields provided by the HD engine). Modified computed variables are returned to the AD Module (overwriting the fields previously sent to the WQ Module) for subsequent conservative calculations. This two-way exchange process is repeated for the duration of a simulation. This general arrangement is presented in Figure 2-1.

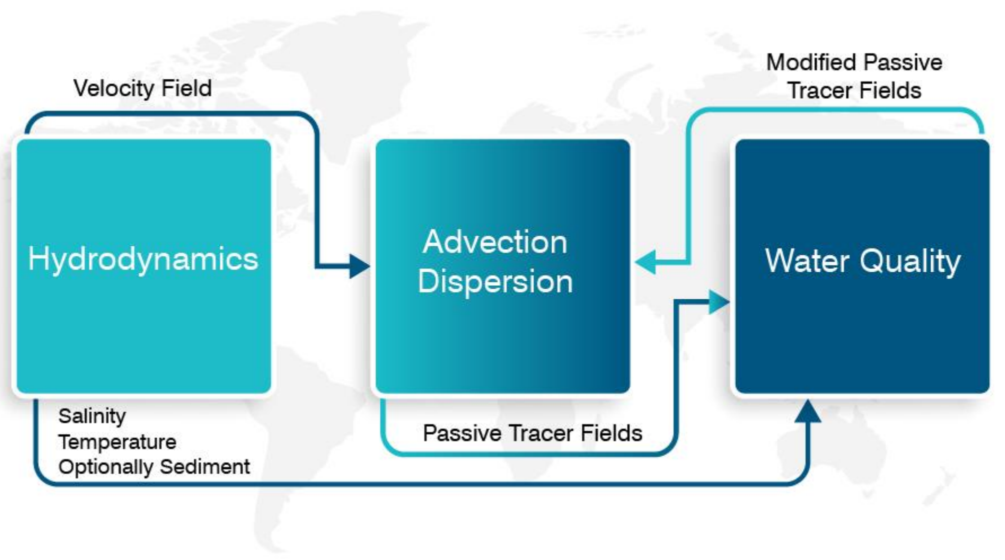


Figure 2-1 Interaction of TUFLOW FV's HD engine, AD and WQ Modules

The WQ Module is activated via specification of a single command in TUFLOW FV's control file, and can run on any simulation timestep. This timestep will typically be much larger than the hydrodynamic timestep, supporting efficient execution of simulations that span the larger timescales at which water quality transformations typically occur.

2.4 Architecture

The architecture of the WQ Module has been deliberately designed to assist users in overcoming some of the challenges and complexities of water quality modelling, and in doing so improve the efficiency and effectiveness with which numerical modelling can support environmental management. Importantly, the WQ Module's architecture provides a mechanism by which users can rapidly initiate and execute water quality simulations without (at least in early modelling stages) concerning themselves with the often time-consuming set up and parameterisation of simulated environmental processes.

The governing component of the WQ Module architecture is the user-selectable Simulation Class, which determines the simulated suite of water quality processes, and therefore the computed variables included within a simulation. The choice of Simulation Class is dependent on the issues being addressed by the WQ Module, in general alignment with the below.

Table 2-1 WQ Module Simulation Classes

Simulation Class	Example Uses
DO ("dissolved oxygen")	Simulation of rudimentary dissolved oxygen dynamics such as: • Desalination return water impact analyses • Seasonal reservoir oxygen analyses • Agricultural drain oxygen analyses
Inorganics	Simulation of relatively simple aquatic ecosystems that experience primary productivity such as: • Shallow urban lakes • Smaller estuaries • Coastal ocean zones that are relatively free of organic materials or sediments
Organics	Simulation of more complex aquatic ecosystems such as: • Impacted coastal embayments or nearshore zones • Impacted lakes • Larger estuaries • Wetlands • Aquaculture operations • Environments receiving point and/or diffuse source organic pollutants

General descriptions of each Simulation Class and its computed variables are provided following.

2.5 Simulation Classes and Computed Variables

2.5.1 DO

The intent of this Simulation Class is to provide a relatively simple entry point to commence water quality modelling. It might be used to examine basic oxygen dynamics across seasons in a water supply reservoir or a relatively newly constructed urban lake, or the potential impacts of dense desalination plant return waters on local coastal sediment oxygenation processes.

The computed variables available in this Simulation Class are:

- Dissolved oxygen

The environmental processes included in this Simulation Class are:

- Sediment-water interface exchange
- Atmosphere-water interface exchange

The computed variables and processes included in this Simulation Class are presented in Figure 2-2 in a conceptual diagram format.

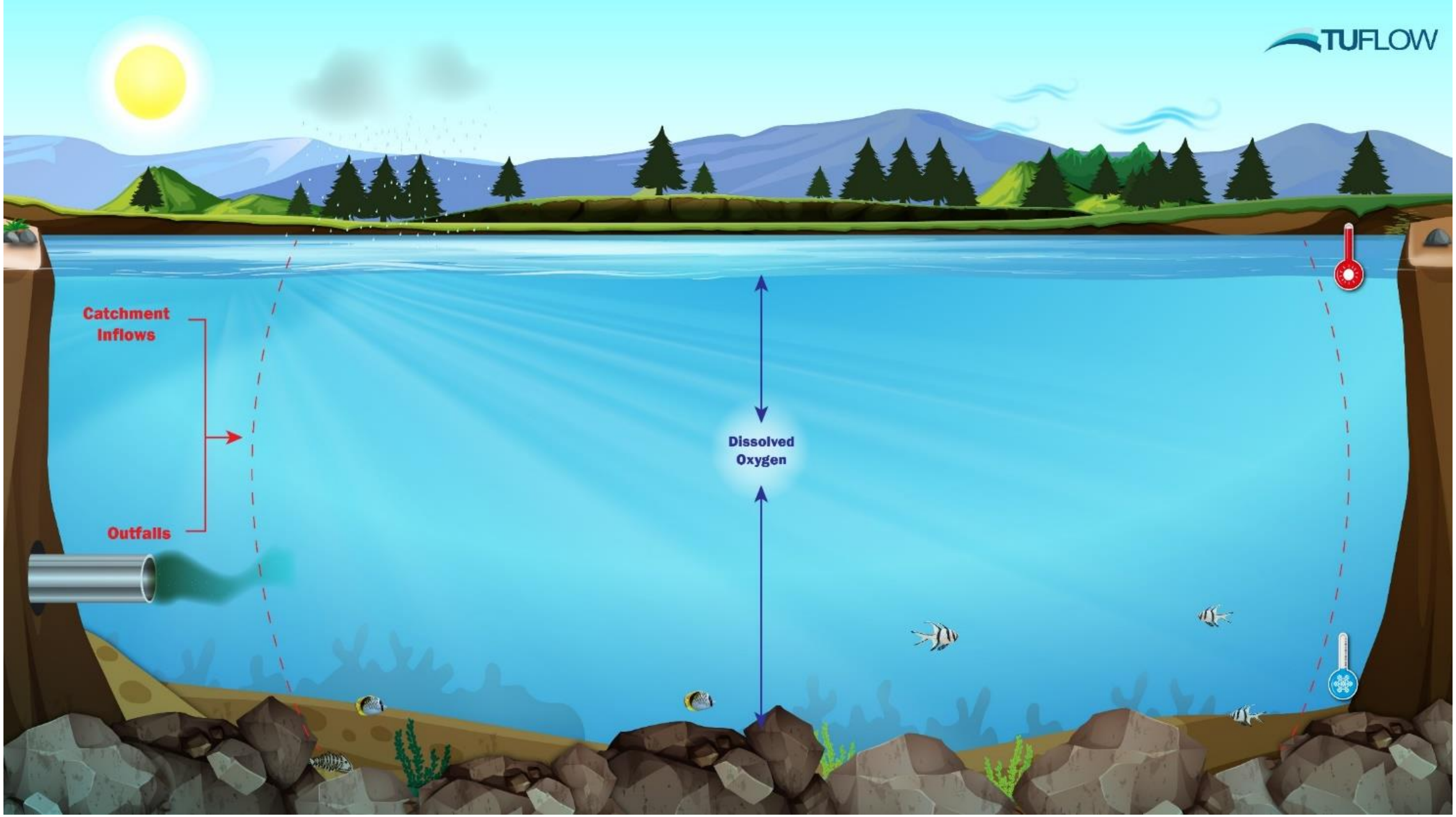


Figure 2-2 Conceptual diagram: DO Simulation Class

2.5.2 Inorganics

The intent of this Simulation Class is to provide a framework for simulating typical water quality conditions where organic matter does not play a significant role in ecosystem processes. For example, it might be used to examine basic inorganic nutrient processing in nearshore coastal environments that include relatively unimpacted sandy bed conditions, or mine voids that receive little catchment inflow. This simulation class could equally be applied to smaller, well flushed estuaries that receive relatively little organic matter, or to basic studies of lakes and water supply reservoirs. The latter cases (amongst others) might also use this simulation class as a stepping stone towards subsequent and more detailed simulations that include organic matter cycling.

The computed variables available in this Simulation Class are:

- Dissolved oxygen
- Silicate
- Ammonium and nitrate
- Free reactive phosphorus and its adsorbed equivalent
- Any number of phytoplankton groups

The environmental processes included in this Simulation Class are:

- Sediment-water interface exchange
- Atmosphere-water interface exchange
- Inorganic nitrogen nitrification, denitrification, dissimilatory reduction of nitrate to ammonium and anaerobic oxidation of ammonium
- Inorganic phosphorus adsorption (and settling) and desorption, and
- Phytoplankton productivity, respiration, mortality, excretion, exudation and settling

The computed variables and processes included in this Simulation Class are presented in Figure 2-3 in a conceptual diagram format.

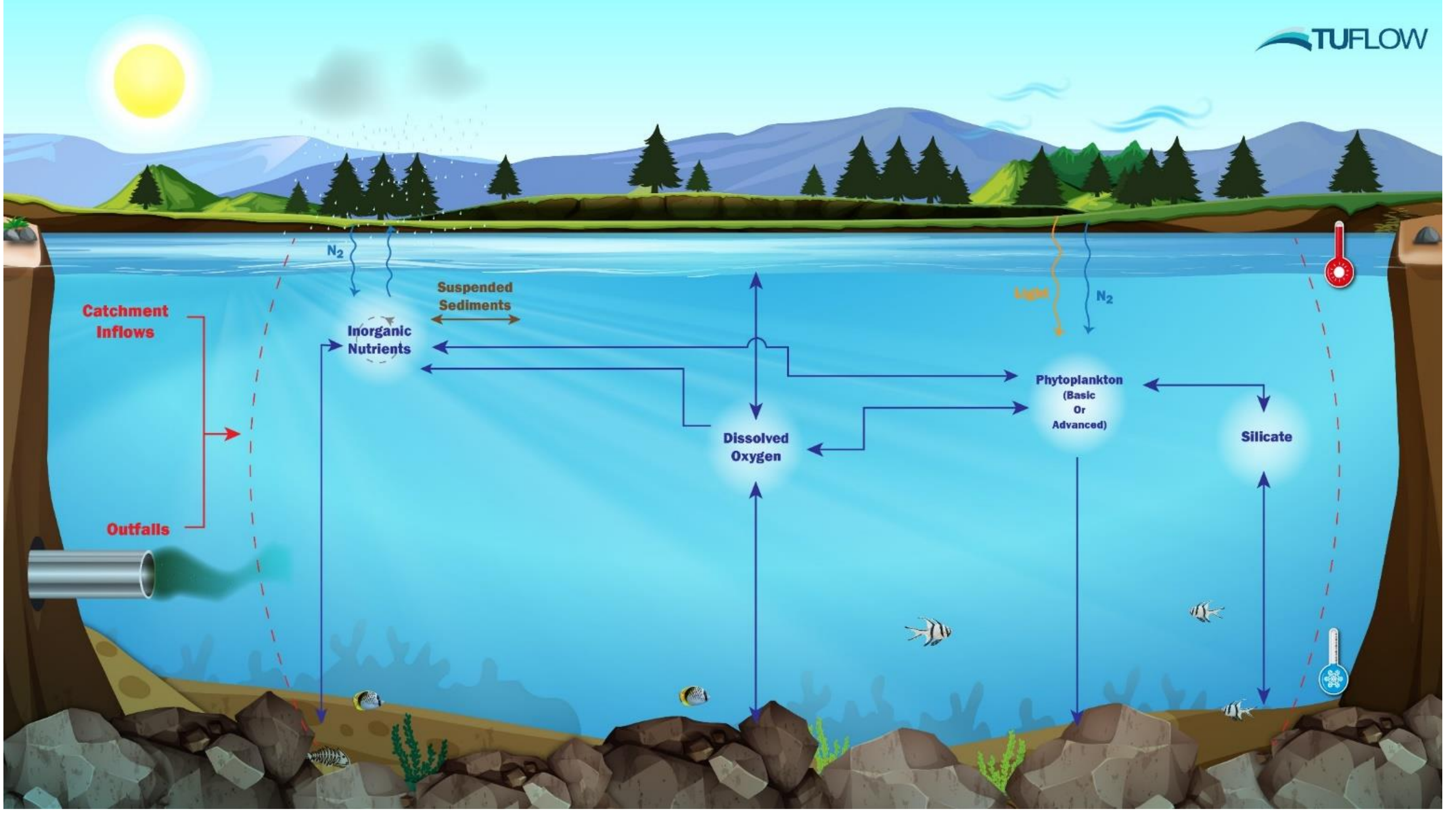


Figure 2-3 Conceptual diagram: Inorganics Simulation Class

2.5.3 Organics

The intent of this Simulation Class is to provide a more advanced water quality modelling tool. There are many applications for this class. For example, it might be used to examine detailed reservoir phytoplankton dynamics or the response of an estuary to both inorganic and organic pollutant loading.

The computed variables available in this Simulation Class are:

- Dissolved oxygen
- Silicate
- Ammonium and nitrate
- Free reactive phosphorus and its adsorbed equivalent
- Particulate, dissolved, labile and refractory organic carbon, nitrogen and phosphorus
- Any number of phytoplankton groups

The environmental processes included in this Simulation Class are:

- Sediment-water interface exchange
- Atmosphere-water interface exchange
- Inorganic nitrogen nitrification, denitrification, dissimilatory reduction of nitrate to ammonium and anaerobic oxidation of ammonium
- Inorganic phosphorus adsorption (and settling) and desorption
- Organic matter settling, hydrolysis, mineralisation, activation, photolysis and breakdown, and
- Phytoplankton productivity, respiration, mortality, excretion, exudation and settling

The computed variables and processes included in this Simulation Class are presented in Figure 2-4, in a conceptual diagram format.

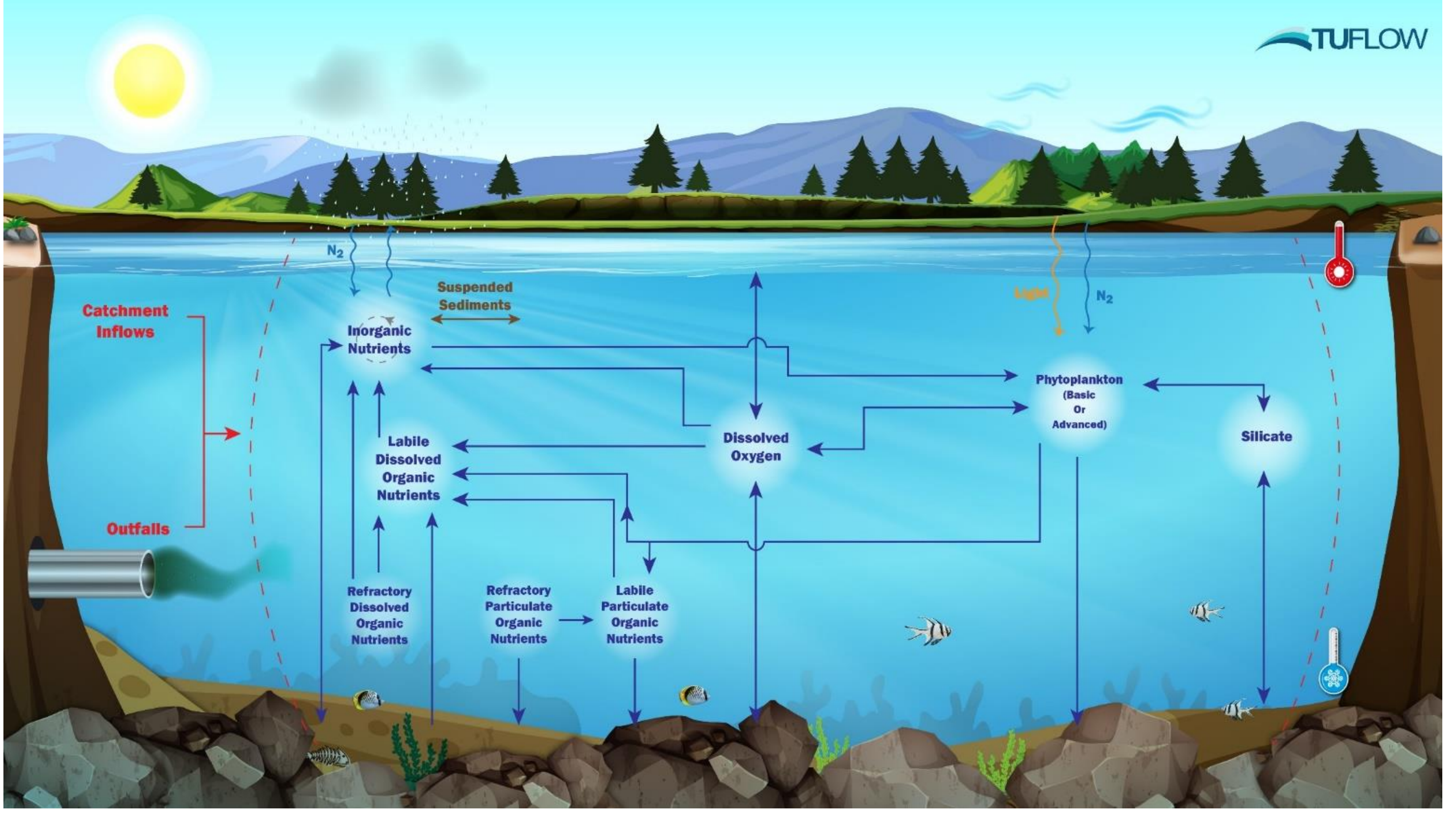
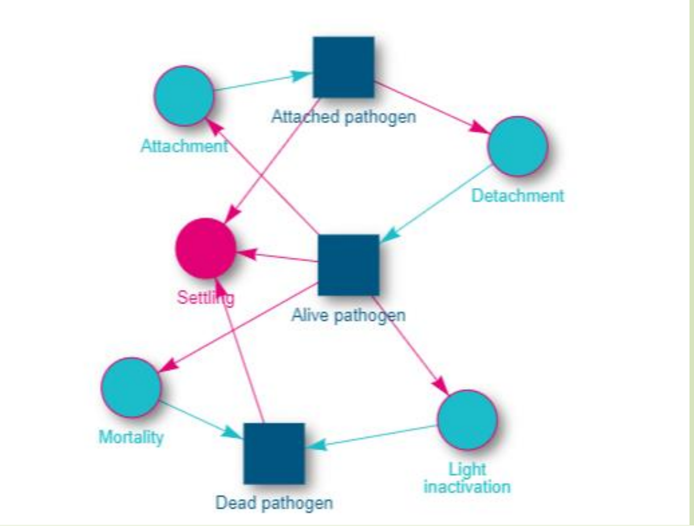


Figure 2-4 Conceptual diagram: Organics Simulation Class

2.6 Pathogens

The simulation of pathogens (such as E. coli, Cryptosporidium or Enterococci, for example) is now available within the TUFLOW Water Quality Module. The pathogens model class can be added to any simulation class (DO, inorganics and organics) as an optional inclusion, and as many pathogen groups can be included as required. The environmental processes that can be parameterised to simulate the fate of pathogens are presented graphically and listed below:



- Natural mortality (dark death), as modified by:
  - Temperature, and
  - Salinity
- Inactivation due to visible, UV-A and UV-B light, as modified by:
  - Salinity, and
  - Dissolved oxygen
- Attachment (and detachment) to suspended sediment
- Settling

Alive, dead and optionally attached pathogens can be simulated. The science describing these processes is presented here in the TUFLOW FV WQ Module manual.

This addition of pathogen simulation allows for detailed investigations of the interactions between wastewater or other contaminated waters with sensitive receptors (such as swimming areas) under a range of dynamically computed environmental conditions. A full suite of diagnostic variables relating to pathogenic fate and transport is also produced by the WQ Module, including:

- Natural mortality (dark death) flux
- Light inactivation flux
- Attachment/detachment flux
- Settling (sedimentation) flux

This allows for the complete mass balance of all pathogen groups to be executed, thus supporting understanding their fate, beyond examination of simple concentrations.

2.7 User Manual

The WQ Module User Manual is an interactive online manual.

2.8 Mass Conservation Model

In order to support users in their application of the WQ Module, a simple mass conservation model, supported by post-processing tools in multiple languages, has been prepared for general distribution and use. The supporting HD model:

- Is free to run without a licence
- Comprises four (4) 2D cells of approximately the same bed elevation and eighty (80) 3D cells. Its geometry is therefore four columns (Figure 2-5)
- Has no hydrodynamic inflows or outflows
- Is forced by atmospheric fluxes typical of mid latitudinal autumnal conditions
- Executes for a period of one month

The mass conservation model is intended to be used as described following.

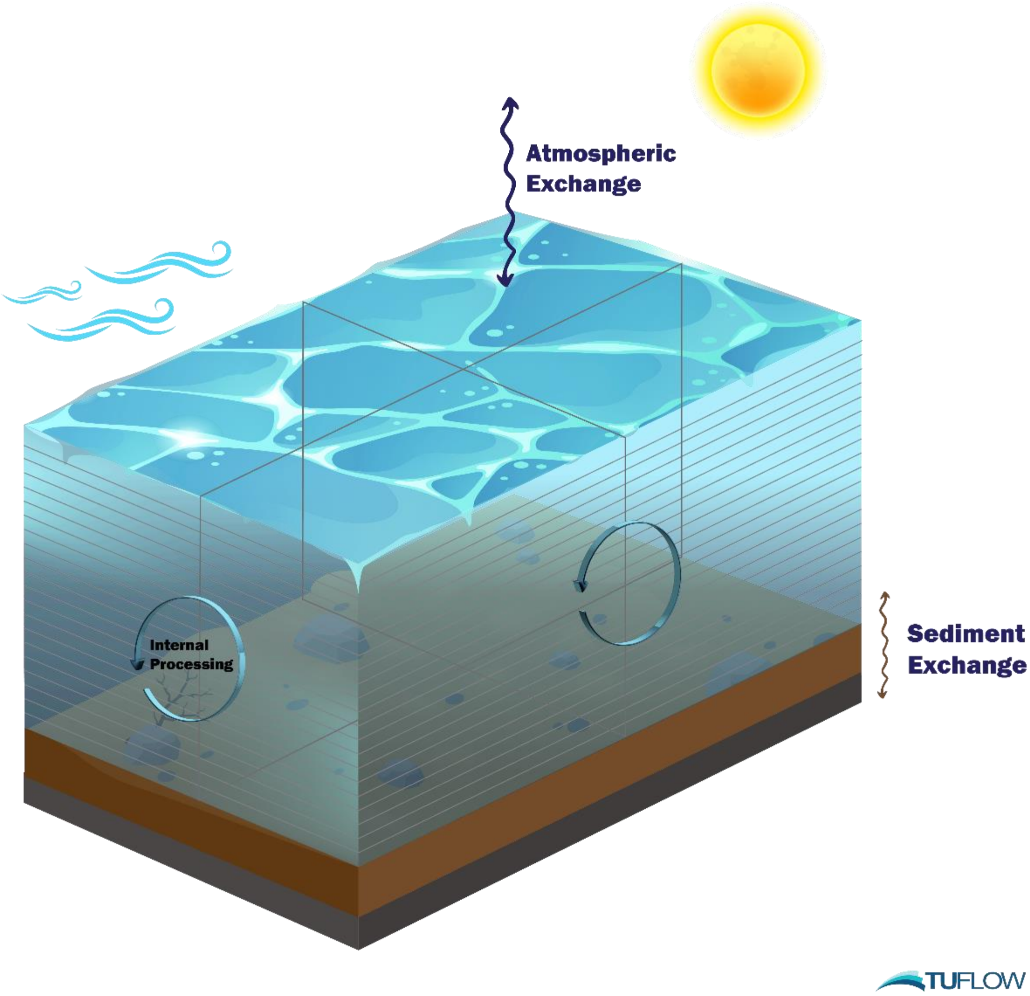


Figure 2-5 Mass conservation model schematic

2.8.1 Mass Conservation Assessments

The mass conservation model is primarily intended to support users in understanding the mass conservation capability of the WQ Module. Mass conservation is one of the most important features of any model and is particularly important in the case of water quality modelling. As such, the TUFLOW FV WQ Module mass conservation model has been constructed to allow a user to:

- Download the model as a zip file (see Section 2.8.3 for access instructions)
- Run the model without needing a licence (i.e. the model is available equally to TUFLOW WQ Module licensees and non-licensees)
- Use the bespoke post-processing tools provided in the download to execute (in MATLAB or Python) a full mass conservation analysis across all simulated computed variables, and
- Alter any WQ Module input parameters, rerun the model and reassess mass conservation

Once post processed, all mass conservation model outputs are saved to both MATLAB data files (if MATLAB is used) and Microsoft Excel spreadsheets for easy access and interrogation (if either

MATLAB or Python are used). Model predictions can also be visually interrogated in QGIS using the freely available TUFLOW Viewer plugin.

The WQ Module control files provided in the mass conservation model can also be deployed 'as is' or as templates by users wishing to build their own TUFLOW FV water quality models.

2.8.2 Algorithm Understanding

The mass conservation model has also been designed to allow users to alter (in conjunction with consulting the WQ Module manual) any WQ Module parameters, and then rerun and reinterrogate the model in a systematic fashion to come to a practical hands-on understanding of how the WQ Module algorithms work. The WQ Module mass conservation model is therefore a testing model, designed to support users in the construction and deployment of their own independent TUFLOW FV water quality models.

2.8.3 Access and Execution

The mass conservation model is downloadable here under Example/Demo models. Access and execution instructions are provided in this Appendix of the online TUFLOW FV WQ Module user manual.

3 Hydrodynamic Engine Updates and Enhancements

3.1 Hydraulic Structure Updates

3.1.1 Flux Limiting Improvements

Structure flux limiting has been added to provide consistency between nodestring, linked nodestrings and linked zones structure types. Flux limiting computes the non-linear shallow water equation (NLSWE) flux that would occur with no structure present (i.e. the highest cell/faces fluxes possible at a structure's location) and ensures that the structure flux does not exceed this NLSWE flux. Flux limiting is a relatively common technique used in many other numerical models (such as TUFLOW HPC) and is intended to reduce the likelihood of unstable model behaviour when connecting 1D structures with the 2D or 3D model domain.

To calculate the NLSWE fluxes at structures two new commands have been added to 2023, and are described as follows:

`Max Open Width ==`

`Zone Inlet/Outlet Orientation ==`

Max Open Width

For nodestring, linked nodestrings and linked zones, the maximum flow width for NLSWE limit calculations is specified using the `Max Open Width ==` command. Typically, you won't need to enter this command as it is calculated automatically based on the culvert or blockage/width file geometry. You can use the max open width command to optionally override the automatic calculation.

If running with metric units the max open width is provided in meters. If units are specified in US Customary, Imperial or English then width is provided in feet.

Zone Inlet/Outlet Orientation

For linked zones the inlet and outlet structure orientation must be specified via the mandatory command `Zone Inlet/Outlet Orientation ==` . This orientation is measured in degrees anti-clockwise from east as shown in Figure 3-1. For nodestring and linked nodestring structures, direction is defined normal to the nodestring and the zone inlet/outlet orientation command is ignored.

```
Structure == LinkedZones, 30 , 31
  Zone inlet/outlet orientation == 45,45     !Degrees anticlockwise from east
  Max open width == 4.2                      !Optional override, meters or feet
  Flux function == Culvert
  Culvert File == ..\model\csv\Culvert_dbase.csv, 1
End Structure
```

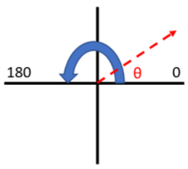


Figure 3-1 Linked zone example and inlet/outlet reference frame

3.1.2 3D Support for Nodestring and Linked Nodestrings

3D support is now available for nodestring and linked nodestring structures via the vertical distribution file and vertical coordinate type. Previously only linked zones supported 3D flux distribution.

The below example shows the use of the vertical distribution file culvert_intake.csv (shown on right) used with the 'height' vertical coordinate type. This will weight the inflow across cells so that all flow crosses the culvert within the range of 0-0.2m above the bed level.

```
Structure == Linked Nodestrings, 10 , 11
  Flux function == Culvert
  Culvert File == ..\model\csv\Culvert_dbase.csv, 1
  vertical coordinate type == height
  vertical distribution file == ..\model\csv\culvert_intake.csv
End Structure
```

HEIGHT	WEIGHT
0	1
0.2	1
0.21	0
99999	0

3.1.3 Weir Submergence

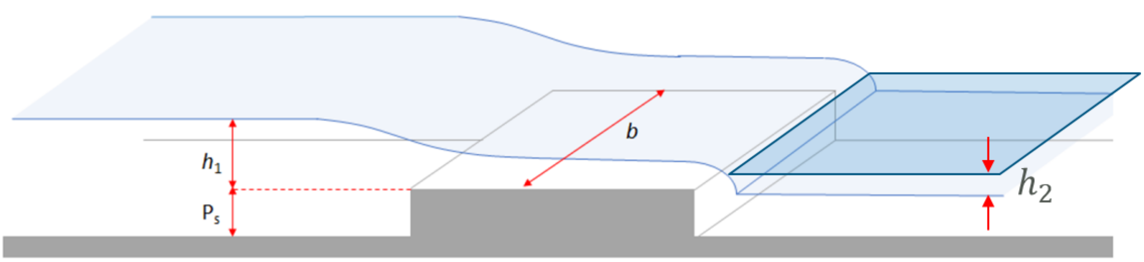
Weir functions and weir_dz flux functions have been enhanced to adjust for tailwater submergence by implementing the Villemonte Equation (Villemonte, 1947). Weir flow is adjusted by the inclusion of the multiplicative C_{sf} term in the weir equation as per:

$$Q = Cbh_1^{3/2} \times C_{sf}$$

$$C_{sf} = [1 - (h_1/h_2)^a]^b$$

Where:

- C: Weir coefficient (default = 1.705)
- H_1: Water depth above the weir crest upstream (m or ft)
- H_2: Water depth above the weir crest downstream (m or ft)
- B: Weir width (m or ft)
- Ex: Weir exponent (default = 1.5)
- a and b: Coefficients for the Villemonte Equation (defaults a=8.55 and b=0.556)
- C_{sf}: Weir submergence factor (default = 0.7)



Weir parameters are entered via the properties structure block command as follows:

```
Structure == Nodestring, 1
    Flux function == weir
    ! Properties     H       C     Ex      a      b      Csf_min
    Properties == 0.50, 1.705, 1.5, 8.55, 0.556,     0.7
End Structure
```

If only the H and C parameters are entered then default values for Ex , a, b and C_{sf_min} are used.

Weir submergence is now switched on by default. For backward compatibility the weir properties can be specified as follows:

```
!Properties ==  H,    C,   Ex,   a,  b, Csf_min
Properties == H, 1.6, 1.5, 1., 0., 0.
```

Villemonte, J.R., 1947. Submerged weir discharge studies. Engineering news record 866, 54–57.

3.1.4 Weir Coefficient Defaults

To provide consistency with TUFLOW Classic/HPC and the broad crested weir equation according to Miller (Miller, 1994) the default value of C has been modified from 1.6 to 1.705.

For backward compatibility the weir properties can be specified as follows:

```
!Properties == H, C, Ex, a, b, Csf_min
Properties == H, 1.6, 1.5, 1., 0., 0.
```

Miller, D.S. (1994). Discharge Characteristics. IAHR Hydraulic Structures Design Manual No. 8, Hydraulic Design Considerations, Balkema Publ., Rotterdam, The Netherlands, 249 pages.

3.1.5 Culvert Inlet Control Support

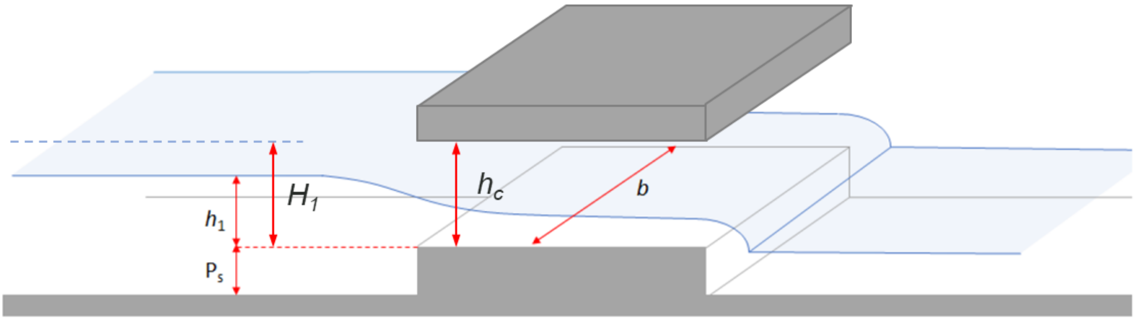
By default, upstream culvert controlling regimes are now supported. Two conditions must be met to trigger the upstream controlled flow regime:

(1) the flow rate calculated based on the upstream controlled flow regime must be smaller than that of the downstream controlled flow rate; and

(2) the upstream water depth above the culvert invert (h_1) must be lower than the following threshold:

h_1 < culvert_critical_hd * h_c

where: h_1 is the upstream water depth above the culvert invert, and h_c is the culvert height.



The default culvert_critical_hd is 99999, i.e. the upstream controlled flow regime is applied once the first condition is met. To adjust the depth threshold or to switch off inlet control for backward compatibility the culvert parameters can be set using the `Culvert Parameters ==` control file command (Note `Culvert Parameters ==` is not a structure block command), e.g. changing the first culvert parameter to 0. (metres or feet) switches off the inlet control regime:

```
! Culvert Parameters == culvert_critical_hd, reserved, reserved, entry/exit_loss_adjust, total_head

Culvert Parameters == 0., 0., 0., 1, 0
```

Note: the second and the third parameters are reserved for future use.

3.1.6 Automatic Entry and Exit Loss Adjustment

TUFLOW FV now supports automatic entry and exit loss adjustment for culverts. The entry and exit loss coefficients are specified in a culvert file. However, for 1D culverts embedded in a 2D domain, the expansion/contraction losses can be partially resolved by the 2D model and using fixed entry and exit loss coefficients can lead to duplication of loss calculations.

Culvert_database.csv

ID	Type	Ignore	Len_or_ANA	n_or_n_F	US_Invert	DS_Invert	Form_Loss	pBlockage	Width_or_Dia	Height_or_WF	Number_of	Height_Cont	Width_Con	Entry_Loss	Exit_Loss
1	2	0	15	0.015	2.1	2.1	0	0	0.9	1.4	1	0.6	1.0	0.5	1
2	1	0	20	0.015	5.0	4.3	0	0	1.8	0.0	1	0.6	1.0	0.5	1
3	1	0	15	0.015	3.0	2.9	0	0	0.6	0.0	1	0.6	1.0	0.5	1
4	2	0	16	0.015	3.0	2.8	0	0	2.4	1.4	6	0.6	1.0	0.5	1
5	2	0	12	0.015	1.1	1.0	0	0	2.4	1.4	3	0.6	1.0	0.5	1
6	2	0	12	0.015	1.1	1.1	0	0	2.4	1.4	3	0.6	1.0	0.5	1

Adjustment is calculated using the implementation below:

Total energy loss:

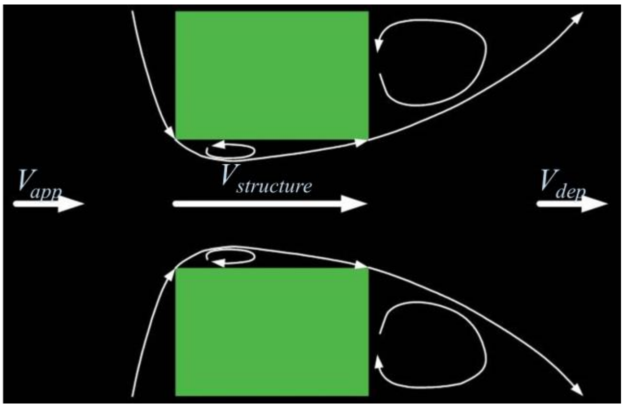
$$\Delta h = \left(K_{entry} + K_{exit}\right)\frac{V_{stucture}^2}{2g}$$

Adjusted energy loss coefficient:

$$K_{entry\_adjusted} = K_{entry}\left[1 - \frac{V_{app}}{V_{stucture}}\right]$$

Adjusted exit loss coefficient:

$$K_{exit\_adjusted} = K_{exit}\left[1 - \frac{V_{dep}}{V_{stucture}}\right]^2$$



Entry and exit loss adjustment is switched on by default as set by the fourth entry of the culvert parameters command:

```
! Culvert Parameters == culvert_critical_hd, reserved, reserved, entry/exit_loss_adjust, total_head
Culvert Parameters ==  99999., 99999., 99999., 1, 0
```

To turn off entry and exit loss adjustment simply set the fourth entry to 0 as follows:

```
Culvert Parameters ==  99999., 99999., 99999., 0, 0
```

Note: `Culvert Parameters ==` is a global control file command and not a structure block command.

In 2023.1 to provide better alignment with TUFLOW Classic/HPC defaults and results, the default automatic culvert upstream and downstream energy losses have been changed to off as per the below configuration:

```
Culvert Parameters == 99999., 99999., 99999., 0, 0
```

3.1.7 Culvert Total Head Option

TUFLOW FV now supports an option to use the total energy head (water level + velocity head) in culvert calculations. This enhancement can improve results for culverts under high velocity / high flow conditions (for example in a main channel). The option is switched off by default. To enable the culvert total head option, set the fifth culvert parameter to 1 as shown in the following example:

```
! Culvert Parameters == culvert_critical_hd, reserved, reserved, entry/exit_loss_adjust, total_head
Culvert Parameters == 99999., 99999., 99999., 1, 1
```

3.1.8 External Loop Update

Previously structure fluxes were updated via the internal timestep mode only. This could lead to model instability if there were significant changes between the upstream and downstream water levels during the internal mode flux update timestep (which is typically an order of magnitude longer than the external mode timestep). The new default for “NLSWE”, weir and culvert flux functions is to ensure that structure fluxes are updated on the external mode timestep.

```
Structure == Nodestring, 1
     …
     External Loop Update == 1
End Structure
```

For backward compatibility set `External Loop Update == 0` in each structure block.

3.1.9 Autoweir

In the specification of autoweirs now requires a change of syntax as follows:

(1) Remove or comment out:

```
!Structure == autoweir
```

```
!End Structure
```

And (2) replace with:

```
Autoweir == 1
```

Syntactic changes are aligned with a refactoring of the autoweir source code to make it more efficient when running on GPU hardware.

3.1.10 Culvert Flux Fix – Negative Water Levels

A fix has been added that addresses an issue introduced in 2023.0 that led to incorrect calculation of culvert fluxes when upstream or downstream water absolute levels were negative. This issue may affect models of intertidal environments and it is recommended to upgrade to 2023.1, especially if modelling culverts in areas where negative absolute water levels are likely.

3.1.11 Improved Structure Scalar Mass Flux Treatment

The calculation of scalar mass flux through structures has been improved to enhance model stability during periods of rapidly varying structure flow.

3.1 Imperial Units

Although supported in pre-2023 versions of TUFLOW FV, the 2023 build has refactored the handling of Imperial/US Customary/English units. Model pre-processing is now completed to convert all Imperial inputs to metric units and all computations are completed using the Metric System. At the time of writing results, outputs are rescaled back to Imperial/US Customary/English units. Both CPU and GPU compute options are available.

Imperial units are supported for 2D hydrodynamic and 3D hydrodynamic models that don't require density coupling. Any models that use the TUFLOW FV AD, Sediment Transport (ST), Water Quality (WQ) and Particle Tracking (PT) Modules need to be constructed in metric units.

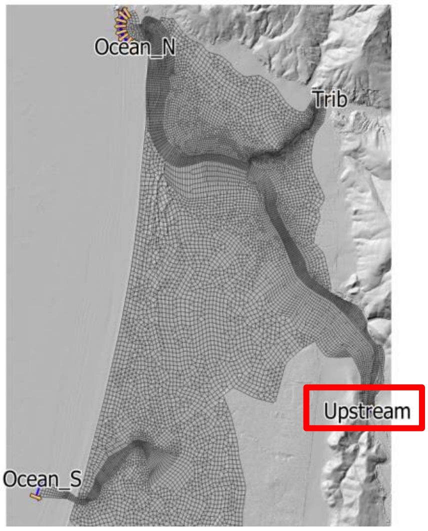
No command changes are necessary to invoke these updates. The only command necessary is to set either: `Units == Imperial` or `Units == US Customary` or `Units == English`.

3.2 GIS Integration Enhancements

3.2.1 Support for Nodestring String IDs

Nodestring IDs can now be specified as character strings with a maximum length of 100 characters. Nodestrings are input in the same manner as pre-2023 builds using the `Read GIS Nodestring ==` command, or alternatively via the 2dm input mesh file. Please see Section 3.2.2 for further guidance on nodestring ID naming.

The following example shows the attribute table of a 2d_ns layer (right) and a map of the nodestring locations (left):

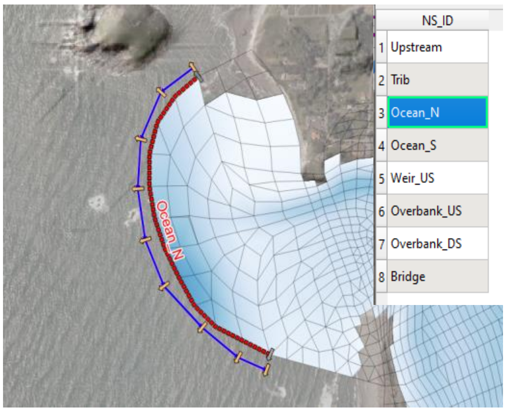


	ID	Flags
1	Ocean_N	BD
2	Ocean_S	BD
3	Trib	BD
4	Upstream	BD

BC blocks have been upgraded to support the new string IDs.

```
BC == Q, Upstream, ..\bc_dbase\steadyQ_01.csv
     BC Header == time, flow
     BC Default == 10
     Sub-Type == 4
End BC
```

Model outputs such as the 2d_ns check file and flux outputs have been upgraded to support string names.



	A	B	C
1	TIME	Upstream_FLOW [m^3 s^-1]	Trib_FLOW [m^3 s^-1]
2	1/01/2010 0:00	0.0	0.0
3	1/01/2010 0:15	-162.5	1.6
4	1/01/2010 0:30	-325.0	3.3
5	1/01/2010 0:45	-487.6	4.9
6	1/01/2010 1:00	-650.1	6.5
7	1/01/2010 1:15	-654.0	7.6
8	1/01/2010 1:30	-657.7	8.7
9	1/01/2010 1:45	-661.5	9.8
10	1/01/2010 2:00	-665.3	10.9

3.2.2 A Note on String Naming

String IDs or Names can contain spaces, are case insensitive, and can contain special characters except for tabs. Although supported, it is recommended to avoid using spaces and special characters. If multiple words are required to define IDs it is recommended to adopt one of the following common conventions:

- Use underscores '_' the so called 'snake_case'
- Or hyphens '-', the so called 'kebab-case'
- Use camelCase or PascalCase

There is no 'correct' method, other than aiming for consistency with the adopted naming approach.

Multiple 2d_ns, 2d_sa and 2d_zn layers can be specified using repeated `Read GIS Nodestring ==`, `Read GIS SA ==` and `Read GIS Zone ==` respectively, providing that IDs/Names are unique. Duplicate nodestring IDs or location names are not supported. If a duplicate ID or Name is encountered TUFLOW FV will error out with a message identifying the cause of the duplicate error.

To utilise the new string naming features a new set of empty template GIS files needs to be generated using the 2023 release. This can be output at model setup time using the `Write GIS Empty Files ==` command.

For backward compatibility, existing models with integer IDs are passed as a string. (i.e. 1 as '1') and the model should run with no additional changes required by the user.

3.2.3 2dm Nodestring ID String Support

TUFLOW FV 2023 supports nodestring names specified directly within the 2dm mesh input. These need to be set on the NS lines of the 2dm. If a string ID is found following the 2dm integer ID, TUFLOW FV will ignore the integer ID and read the string ID. For example, in the below image TUFLOW FV will read the first NS line (2dm line 45337) as 'Upstream' and ignore the integer value 1.

Please also refer to Section 3.2.2 for further guidance on nodestring ID naming.

```
45337  NS 1 2 3 4 5 6 7 8 -9 1 Upstream
45338  NS 101 102 103 104 105 106 107 108 -109 2 Downstream
45339  NS 21 22 23 24 25 26 27 28 -29 3 Lake Inflow
45340  NS 31 32 33 34 35 36 37 38 -39 4 Inflow A
45341  NS 41 42 43 44 45 46 47 48 -49 5 Inflow B
```

3.2.4 GIS Integration for QC Boundaries

QC boundary locations can be assigned via a GIS point layer using the `Read GIS SA ==` command. This provides substantial workflow efficiency improvements over the previous BC block x, y coordinate assignment method.

QC locations are assigned using 2d_sa point layers that include a single 'Name' attribute (Refer to Section 3.2.2 for further guidance on QC point ID naming).

The following example shows how several QC point locations can be included in the one GIS layer 2d_sa_inflows_001_P.shp:

```
Read GIS SA == ..\model\gis\2d_sa_inflows_001_P.shp
```

BC blocks and output logging have been updated to support QC naming.

```
BC == QC, Upstream A, ..\bc_dbase\Flow.csv
      BC Header == Time,Q
End BC
```

Cells selected by QC boundaries are output via the 2d_bc check file.

3.2.5 GIS Integration for QC_POLY Boundaries

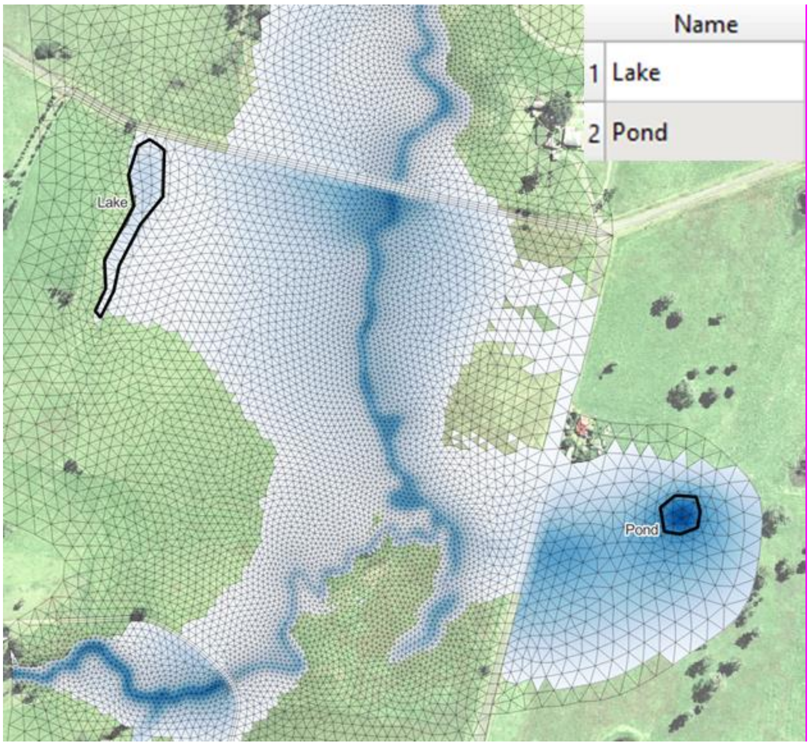
QC_POLY boundary polygons can be assigned using the `Read GIS SA ==` command with 2d_sa GIS layers of region geometry type. 2d_sa layers contain a single 'Name' attribute used to the identify the boundary location. Please refer to Section 3.2.2 for further guidance on QC_POLY ID naming.

The below example shows a model with two QC_POLY boundary conditions, 'Lake' and 'Pond'.

```
! BOUNDARY CONDITIONS
Read GIS SA == ..\model\gis\2d_sa_Flow_Inputs_001_R.shp

bc == QC_POLY, Lake, ..\bc_dbase\M01_002.csv   !Polygon Name, Input timeseries
  bc header == time_hr, FC02                   !Time column, data column
end bc

bc == QC_POLY, Pond, ..\bc_dbase\M01_002.csv   !Polygon Name, Input timeseries
  bc header == time_hr, FC04                   !Time column, data column
end bc
```



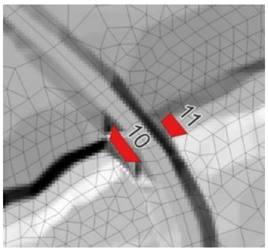
Cells selected by QC_POLY boundaries are output via the 2d_bc check file.

3.2.6 GIS Integration for Linked Zones Structure Type

A new 2d_zn GIS input layer is available in TUFLOW FV 2023 for defining linked zone structures. The 2d_zn layer is read into TUFLOW FV via the command: `Read GIS Zone ==` as shown in the example syntax below. The image on the right shows the location of the zone polygons. It should be noted that although the below example uses integer IDs, string naming is also supported (Please refer to Section 3.2.2 for further guidance on zone ID naming).

```
! HYDRAULIC STRUCURES
Read GIS Zone == ..\model\gis\2d_zn_Structure_Regions_001_R.shp

Structure == LinkedZones, 10 , 11
  Zone inlet/outlet orientation == 45.,45.
  Flux function == Culvert
  Culvert File == ..\model\csv\Culvert_dbase.csv, 3
End Structure
```



A new GIS check file layer _1d_to_2d_check_R.shp is output showing which mesh elements have been selected.

3.2.7 GIS Integration for Points and Profile Outputs

Points and profile outputs can now be specified via the output block `Read GIS PO ==` command. For example:

```
output == points
  Read GIS PO == ..\model\gis\3d_po_output_points_001_P.shp
  output parameters == h
  output interval == 900.
end output
```

The `Read GIS PO ==` command utilises the 3d_po point GIS layer which includes the following attributes:

- Type: Reserved (not used)
- Label: Used for name of point
- Comment: Optional user info
- Vert_min: 3D Vertical averaging override
- Vert_max: 3D Vertical averaging override

If running a 3D model, for a given point location the vert_min and vert_max attributes will override the default vertical averaging limits (0,99999.) or those specified via the `Vertical Averaging ==` output block command.

```
Output == points
  Read GIS PO == ..\model\gis\3d_po_reporting_002_P.shp
  vertical averaging == sigma-range, 0., 0.1
  Output parameters == H, V
  Output interval == 300.      ! (s)
End output
```

The assignment of 3d_po input layers has the following set of conditions:

- Duplicate location names in the same output block are not supported
- Multiple `Read GIS PO` commands are allowed in a single output block and are additive, provided the location names in each file are unique. For example, if file 1 contains ‘a’, ‘b’ and file 2 contains ‘c’ ‘d’ then the output block will output results at ‘a’, ‘b’, ‘c’ and ‘d’
- The same 3d_po layer can be used more than once in a model, providing that it is specified in a separate output block (noting the `suffix ==` command will also be required to get multiple outputs). If the same file is specified more than once in a single output block it will fail due to duplicate location names
- Mixing of legacy csv and GIS methods in the same BC block is not supported
- Legacy csv methods are still supported

3.2.8 TPC Outputs

The 2023 release by default uses a new additional plotting output folder structure providing the model utilises GIS integration. For the 2023 approach a “plot” folder is created as a subdirectory of the directory set by the `Output Dir ==` command. The plot folder contains the subfolders and files as listed in Table 3-1.

A Python library (TuPlot) and the QGIS TUFLOW Plugin are available to provide powerful scripts and a GIS viewing platform to view and post-process these data. This .csv plotting data can also be accessed by standard spreadsheet software such as Microsoft Excel.

Table 3-1 Plot Folder File Descriptions

Folder	Filename	Description
\plot\ Folder		
plot\	<simulation_id>.tpc	TUFLOW Plot Control file. This is a simple text file that contains information and links to the data available for the simulation. This file is used by the QGIS TUFLOW Plugin to load up complete plotting data sets and quickly access data.
\plot\csv\ Folder		
plot\csv\	<simulation_id>_2D_<output_variable>.csv	Contains flux or points output timeseries data for each nodestring or points dataset respectively.
\plot\gis\ Folder		
plot\gis\	<simulation_id>_PLOT.csv	Summary .csv file containing information on the GIS objects and plot types available.
plot\gis\	<simulation_id>_PLOT_L	GIS layer in shapefile or MapInfo file format containing all plot line objects (e.g. flux nodestring reporting locations).
plot\gis\	<simulation_id>_PLOT_P	GIS layer in shapefile or MapInfo file format containing all plot point objects (e.g. point output reporting locations).

3.3 Demo Licensing Mode

The TUFLOW FV 2023 release includes a new free demo licensing mode. This allows models to be run licence free provided the following demo model conditions are met:

- Models must have less than 5,000 2D cells, and
- A total runtime of less than 10 mins, and
- Not include restart files

Either `Tutorial Model == ON` or `Demo Model == ON` will run TUFLOW FV in demo mode. Provided a model meets the requirements for demo mode it can be run with any and all TUFLOW FV Modules. If a model is successfully started in demo mode the following message will be output in the TUFLOW FV log file.

```
Successfully initialised model output.

Checking tutorial model credentials:
Succesful. This simulation is running in demo mode.

Running simulation
Number of OpenMP threads=4
Thread stacksize=4194304

Entering timestep loop
t = 01/07/2010 00:00:00.  dt =   0.000 /  0.000 s.  elapse
    Writing H datfile output. t = 01/07/2010 00:00:00.
    Writing V datfile output. t = 01/07/2010 00:00:00.
    Writing D datfile output. t = 01/07/2010 00:00:00.
    Writing TEMP datfile output. t = 01/07/2010 00:00:00.
    Writing SAL datfile output. t = 01/07/2010 00:00:00.
```

3.4 New Command Line Arguments

3.4.1 Test Model Command Line Argument

The -t switch can be used in Windows or Linux batch/shell scripts to run a model in test mode. Using test mode, the model will start up, check all input layers, write GIS check files if requested and run the first timestep before cleanly exiting the simulation.

```
REM Example Test Switch
set exe="path_to_exe\TUFLOWFV.exe"
set OMP_NUM_THREADS=4

%exe% -t myrun_001.fvc
```

3.4.2 Restart File Command Line Arguments

Restart and/or bed restart files can be set in Windows or Linux batch/shell scripts via the command line arguments -rst and/or -sedrst. These commands will overwrite any restart or bed restart commands set within the TUFLOW FV control file or sediment control file. Examples of -rst and -sedrst switches are provided as follows:

```
REM Example HD and SED Restart Switches
set exe="path_to_exe\TUFLOWFV.exe"
set OMP_NUM_THREADS=4

%exe% -rst=.\log\myrun_001.rst -sedrst=.\log\myrun_001_bed.rst myrun_002.fvc
```

3.4.3 Control File Inputs Via Environment Variables

A powerful and flexible new feature has been added that allows environment variables specified in batch or shell script files to be read within any TUFLOW FV Control File using the $ syntax. This can be set for any values on the right hand side of the == in a TUFLOW FV command. In the below example the model's initial water level is specified via the command line variable **IWL**.

Batch file snippet where IWL has been set to the value of 2:

```
REM Example of environment variable 'IWL'
set exe_new=C:\TUFLOWFV\2023.1.ExampleOnly\TUFLOWFV.exe
set OMP_NUM_THREADS=8
set IWL=2

%exe% Env_Var_Example_001.fvc
```

TUFLOW FV Command File (.fvc) syntax where the $IWL will be dynamically replaced at runtime by the IWL variable value:

```
!________________________________________________________________
! Initial Conditions
Initial Water Level == $IWL
!________________________________________________________________
```

The resulting log file output showing the initial water level has been set to the value specified by IWL of 2.0:

```
BC == Q, 2, ..\bc\MyFlow.csv
  BC HEADER ==  t, Q
END BC
INITIAL WATER LEVEL ==    2.000000
OUTPUT DIR == ..\output\
OUTPUT == NETCDF
  OUTPUT PARAMETERS ==  H, V,
  OUTPUT INTERVAL ==    600.0000
END OUTPUT
```

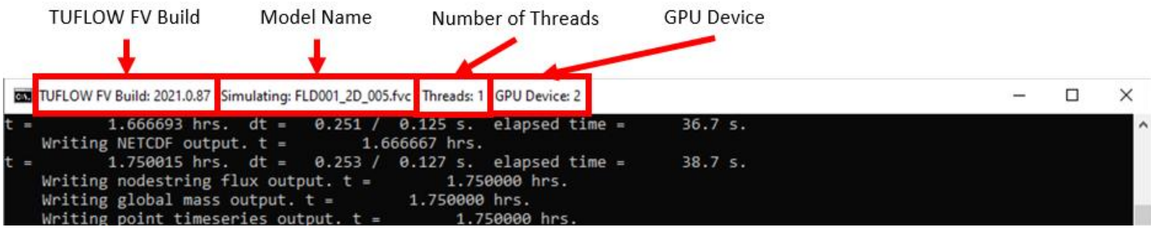
3.5 Bug Fixes and Minor Enhancements

3.5.1 Console Window Titles

When running on Windows Operating System the TUFLOW FV console will now show the TUFLOW FV build, the model fvc name and the number of threads being used to run the simulation. If running on GPU, the GPU device ID will be shown. This feature is not supported on Linux OS.

The following image shows a CPU only simulation:

And the following image shows a GPU enabled simulation run on GPU Device ID 2.



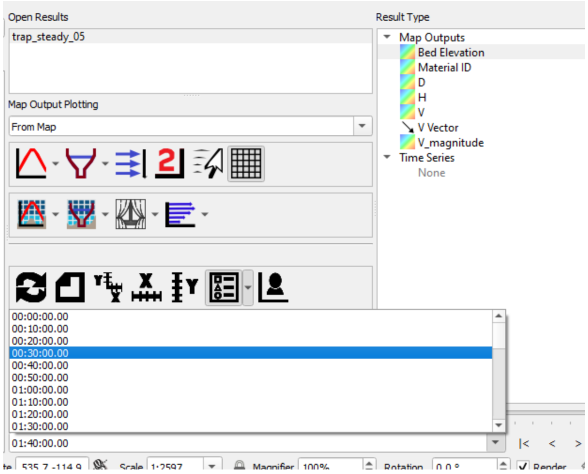
3.5.2 Model Output Time

A new output block command `Exact Timestep ==` (default 0) will report model output at the time requested by the user output block. The previous default was to output model results on the first model timestep following the requested output time.

```
!___________________________________________________________________
! OUTPUT COMMANDS
Output Dir == ..\results\

!New default for model output times
Output == netcdf
  Output Parameters == h, v, d
  Output Interval == 600.
  Exact Timestep == 0  ! New default not required but added for example.
End Output
```

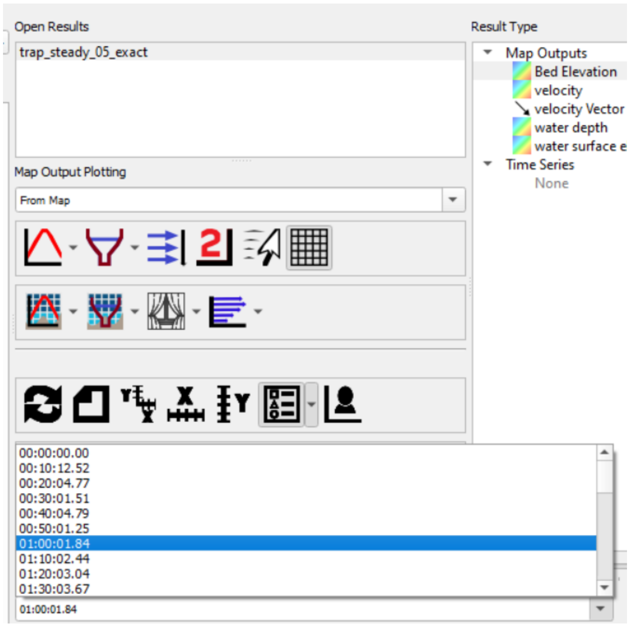
The resulting output times are demonstrated via the QGIS times below.



For backward compatibility simply set the output block command to `Exact Timestep == 1` as shown in the following example block:

```
!Legacy or backward compatibility
Output == netcdf
  Output Parameters == h, v, d
  Output Interval == 600.
  Exact Timestep == 1
  Suffix == exact
End Output
```

And the subsequent output times (the legacy default was to output the first model timestep following the requested output time):



3.5.3 GIS File Writes on Linux

A fix has been implemented to correctly output GIS Shape files to NTFS formatted drives if running a Linux build of TUFLOW FV. Previously if using a Linux Operating System and trying to write to NTFS the following error would occur.

```
Trying to open (BO) file log/trap_steady_05_messages_P.dbf...OK.  File Unit:1008
...Closed File Unit:1007

ERROR 0437 - Writing .shp file header.

Closing any unclosed GIS layers...

Closing GIS Layer 1 [log/trap_steady_05_messages.shp]...
...Closed File Unit:1008
...Closed File Unit:1003
...Closed File Unit:1004
...Closed File Unit:1005
...Closed File Unit:1006

...Closed File Unit:1001
...Closed File Unit:1002

Press Enter to Close
PAUSE prompt>
```

3.5.4 Read GIS Z Line Fix

A new command `Snap Tolerance ==` command has been added to fix point snapping issues in spherical models when using the `Read GIS Z Line ==` command. The issue would lead to breakline points being ignored and incorrect assignment of cell centre elevations.

Default values for snap tolerance are 0.001 m (or feet if `Units == Imperial`) and 1.0E-06 degrees. These values can be modified if required.

3.5.5 Matrix Flux Function with Negative Flows Fix

An issue affecting the 2020 release that set reverse or backflows to zero through matrix flux functions has been fixed.

3.5.6 Matrix Flux Function Scalar Fix

A fix has been added that resolved issues with matrix flux functions in models with scalar variables (salinity, temperature, suspended sediments, passive tracers or water quality variables). Although mass conserving, and not affecting results away from the structure location, some models with matrix flux functions were exhibiting highly localised oscillations in concentration fluctuations on the upstream side of the structure.

3.5.7 XMDF and NetCDF Compatibility Update

Both XMDF and NetCDF rely on the same underlying HDF5 libraries. Testing has shown that unexpected conflicts can occur when running a model outputting both XMDF and NetCDF in the same simulation. To avoid these conflicts, TUFLOW FV will now throw an error and require selection of either XMDF or NetCDF. The error message is as follows:

XMDF and NetCDF map output formats are incompatible. Please choose either XMDF or NetCDF

3.5.8 Mesh Coordinate Checks

Mesh coordinate checks are now completed during model initialisation. If a spherical model is run and any coordinates are found outside of the range $\pm 360^0$ longitude or $\pm 90^0$ latitude an error will be generated and TUFLOW FV will exit. For example:

ERROR: Out-of-range longitude [-360.,360.] encountered at node:401363

If accidentally running a spherical coordinate model with cartesian coordinates (the default) a new warning message has been added:

'Cartesian model specified with coordinates within range [-360.,360.], [-90.,90.]. Check if spherical coordinates should be specified.'

3.5.9 Aquaveo SMS Super File Fix

A fix has been included to SMS super files (.sup) extension to write the correct mesh path. Super files are written by default when datv or XMDF outputs are selected.

3.5.10 Mesh Check File Fix

Models with spherical coordinates have had a fix added that correctly outputs the mesh check file and csv check file coordinates to decimal degrees. Previously these were being output in radians.

3.5.11 Nodestring Overlap Checks

A new routine has been added to TUFLOW FV to check for nodestring overlaps. If internal nodestrings overlap a warning will be added to the log file. If nodestring overlap occurs on an open boundary an error will occur as follows:

ERROR: Found overlapping boundary condition nodestrings

ERRORSTACK:fvdomain_construct:init_dmn:fvmesh_construct:Check_nodestring_overlap:Found overlapping boundary condition nodestrings

3.5.12 GPU Command Line Argument Fix

A fix has been implemented so that `Hardware == GPU` will be invoked when using the -pu switch. In 2020.03.105 undesired behaviour had occurred where both the -gpu and -pu switch were required to run on GPU hardware. The below example with the new correct behaviour will run TUFLOW FV on GPU Device 0.

```
REM GPU Run and Device ID Switch
set exe="path_to_exe\TUFLOWFV.exe"

%exe% -pu0 myrun_001.fvc
```

3.5.13 Model Start Time

It is now mandatory to specify the start time in the TUFLOW FV Control Fil (.fvc). If the start time is commented out (for example, sometimes users will comment out the start time when using restart files) an error will now be thrown. This change was required to fix issues experienced with particle memory allocation when the Start Time command had not been specified.

3.5.14 DATV ISODATE Reference Time

When running with DATV outputs and ISODATE time format the reference time is now written to the dat file output. This improves dat file reading in Aquaveo SMS.

3.5.15 CPU Parallel Processing for GPU Runs

When running on GPU Hardware the number of CPU threads for non-GPU components can now be set using the OMP_NUM_THREADS environment variable. Previously the number of CPU threads was locked to 1 when using GPU. This is particularly relevant when using Modules such as Sediment Transport, which run on CPU.

In this example OMP_NUM_THREADS is set to 4 and the GPU device is being set to 0.

```
set exe="C:\TUFLOWFV\2023.1\TUFLOWFV.exe"
set OMP_NUM_THREADS=4

%exe% -pu0 FLD000_2D_001.fvc
```

3.5.16 Temporal Extrapolation Check

In 2023.1 by default TUFLOW FV will throw an error if any boundary condition temporal data does not cover the full model simulation period.

For example:

A model setup has `Start Time == 01/01/2023` and `End Time == 01/02/2023` (dd/mm/yyyy). If a boundary condition is specified that only covers the period 01/12/2022 to 15/01/2023 then the boundary does not include data for the full model period, and by default the following error will result:

ERROR: Model start or end time is outside of BC 6 temporal extent.

ERROR-STACK:fvdomain_construct:init_bc:fvbc_construct:temporal_extrap:Model start or end time is outside of BC 6 temporal extent.

where BC 6 is the 6th boundary read sequentially from the TUFLOW FV Control or read/include files.

This behaviour can be switched to WARNING globally via the command:

```
Global Temporal Extrapolation Check == {ERROR} | WARNING
```

If set to WARNING, a message will be written the log file and bc data will be extrapolated from the closest available time. Although possible, this is not recommended.

Global ERROR or WARNING behaviour can also be overwritten on an individual bc block basis via the bc block command `Temporal Extrapolation Check ==` as follows:

```
BC == WL_CURT, Ocean, FES14_MB_2012-12-01_2013-03-01_001_TND.nc
  BC Header == time_local, ns1_chainage, dummy, ns1_wl
  BC Update DT == 60.     ! (s)
```

```
  BC Time Units == days
  Includes MSLP == 0
  Temporal Extrapolation Check == {ERROR} | WARNING
End BC
```

3.5.17 Output Interval Check

Improved error reporting has been added where the model result output interval is less than the model timestep.

```
Timestep Limits == 0.01, 30.          ! (s)
Output == NetCDF
  Output Parameters == h, v
  Output Interval == 20.              ! (s)
End Output
```

ERROR: Output interval must be higher than the maximum allowed timestep.

ERROR-STACK:fvdomain_construct:fvinitout:fvout_construct:Output interval must be higher than the maximum allowed timestep.

3.5.18 3D NetCDF Statistics Fix

An output allocation issue during model startup (hard crash) that affected models with NetCDF statistics output specified has been fixed . This occurred in models with a mixture of 2D and 3D model output parameters, for example a 3D model with H (2D) and V (3D) specified.

3.5.19 Z-Layer Morphology Warning

A warning message has been added recommending the use of Sigma 3D vertical layering when running with dynamic morphology via the sediment transport module. If running with Z or Z-Sigma layers, erosion or deposition of the bed can lead to instabilities or model pause issues. This is because the height of Z layers are fixed at the start of the simulation. As the bed changes during dynamic morphology Z layers can become either very thin or extend below their initial z face elevation, which is not supported by fixed Z layers.

It is recommended to use Sigma 3D vertical layering (not Z or Z-Sigma) when sediment morphologic update is enabled.

3.6 XMDF and NetCDF File Locking (HDF5 Compatibility Issues)

This is an important note concerning the use of recent builds of QGIS, Python or other viewers/libraries that utilise HDF5 1.12 and later to view NetCDF or XMDF result output. This note applies to all TUFLOW FV versions, not only the 2023.1 Release.

HDF5 is a data model, library and file format for storing and managing large data files. Both the Aquaveo XMDF and Unidata NetCDF map output types offered by TUFLOW FV are built on HDF5 components for data storage, compression and management.

HDF5 Version 1.12 and later includes new default file locking behaviour that can result in TUFLOW FV model crashes if XMDF or NetCDF results are opened whilst TUFLOW FV is running, regardless if in read only or write modes. Console output will typically display the following prior to the model prematurely exiting:

NetCDF:

```
ERROR: Error invoking nf90_enddef. Netcdf status value is -101. NetCDF: HDF error.

ERROR-STACK:fvdomain_run:fvdomain_run_timestep:fvout_wrtout:fvout_wrtnetcdf:initialise_netcdf:CheckNCStatus:Error invoki
ng nf90_enddef. Netcdf status value is -101. NetCDF: HDF error.
```

XMDF:

```
Opening output files...
Opening XMDF Super File ...
Trying to open file: ..\results\FLD000_2D_001.xmdf.sup ... OK. File unit: 101
Trying to open file: ..\results\FLD000_2D_001.xmdf ...
```

Therefore, to avoid a model exiting prematurely it is not recommended to open XMDF or NetCDF if the viewer utilises HDF5 1.12 or later whilst a model is running. The current work-around is to make a copy of results and view the copy, or to wait until the simulation has completed.

This issue is temporary, and we are currently working towards a solution which includes upgrading the HDF5, NetCDF and XMDF libraries when suitable updates become available.

Please note that the ability to review results while a model is running is an important feature of TUFLOW FV and we will providing a fix as soon as possible.

4 Advection Dispersion Module

4.1 Advection Dispersion Fixes and Minor Enhancements

4.1.1 Shortwave Radiation Bed Absorption Enhancement

It has been identified that for some models the heat module was prone to over-predicting evaporative cooling of shallow areas. The problem was that the majority of shortwave radiation reaching the seabed was absorbed by the sediment and effectively lost from the water column heat budget.

We have now implemented a simple extension of the previous scheme that allows the seabed absorption to be split between a component that is retained in the sediment and a component that is radiated back into the water column.

TUFLOW FV now has two parameters defining the bed absorption, with the following defaults:

```
Shortwave Radiation Bed Absorption == 0.45, 0.45
```

That is, 45% of the shortwave radiation heat is absorbed into the sediment and 45% is radiated into the water column (this implies that 10% is reflected off the bed. This reflected light (shortwave radiation) will be absorbed in the water column as it reflects back towards the surface. Anything remaining will then leave the water column back into the atmosphere).

The previous default behaviour was 90% absorption of shortwave radiation at the seabed (and 10% reflection). For backward compatibility, specify:

```
Shortwave Radiation Bed Absorption == 0.90, 0.0
```

5 GPU Module

5.1 GPU Fixes and Minor Enhancements

5.1.1 Improved GPU-CPU Structure Data Passing

Code for hydraulic structure CPU-GPU data passing has been refactored to improve efficiency. These changes do not affect model results and for certain models this code refactoring has resulted in model speedups of 200%.

5.1.2 Sediment Density Coupling

Sediment density coupling is now available on both GPU and CPU hardware via:

```
Include Sediment == 1,1
```

Previously this functionality was available on CPU only.

5.1.3 Stokes Drift Fix

Stokes drift is now available on both GPU and CPU hardware via:

```
Include Stokes Drift == 1
```

Previously this functionality was available on CPU only. The default is off: `Include Stokes Drift == 0`

6 Sediment Transport Module

6.1 Sediment Transport Module Fixes and Minor Enhancements

6.1.1 Changes to Cell Diffusion Factor

TUFLOW FV applies a 'higher order' vertical concentration profile for suspended sediment. A bug where the 'higher order' profile was applied to dry cells (cell water depth is smaller than hwet) has been fixed. This issue was known to cause high deposition rates in dry cells and could lead to model instability.

6.1.2 Flocculation Settling Model

A bug that did not correctly allocate the flocculation parameter suite on GPU has been fixed and the flocculation model can be used in full. Please note this issue was not present when running on CPU hardware.

7 Particle Tracking Module

7.1 Particle Tracking Module Fixes and Minor Enhancements

7.1.1 Dry Boundary Particle Treatment Fix

A fix has been added that allows particles to dry out and then wet again in regions of the model with wetting and drying. Previously, if a particle dried out it would erroneously remain dry unless particle crawl motility was specified.

7.1.2 Velocity Profile Fix

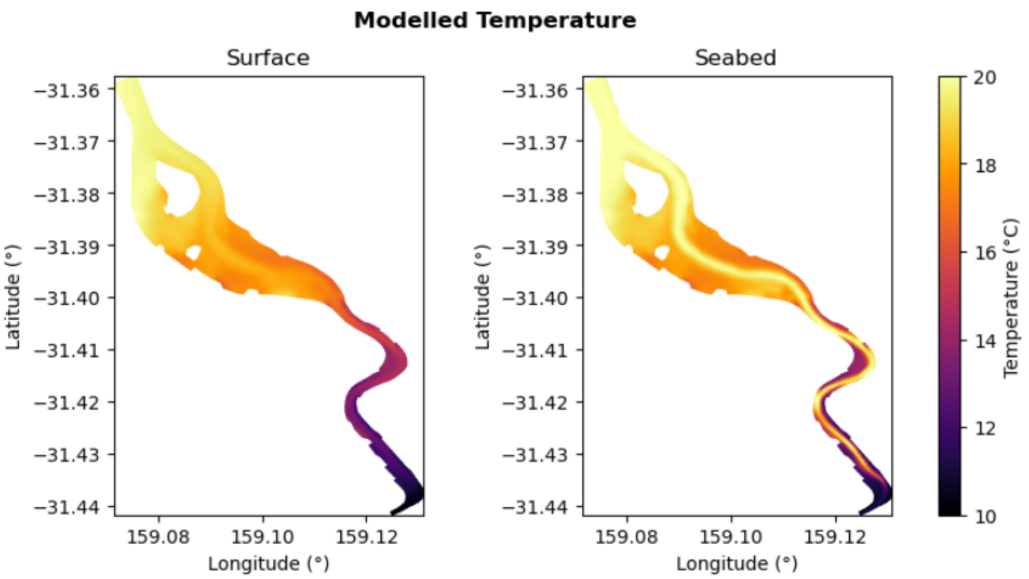
A fix has been included in the Particle Tracking (PT) Module for log velocity profile reconstruction. This only affected particle tracking simulations coupled with depth-averaged hydrodynamics and `Velocity Profile Model == logarithmic`

8 Visualisation and Model Development Tools

8.1 Python Toolbox

The TUFLOW FV Python Toolbox provides a free suite of tools for the post-processing and visualisation of TUFLOW FV NetCDF results.

A tutorial including installation instructions is provided on our TUFLOW FV Python Toolbox page with example scripts. The figure below demonstrates an output from the TUFLOW FV Python Toolbox *tfv* Python module.



Version 1.0.5 of the Python TUFLOW FV Toolbox is a significant upgrade that supports both pip and conda install environments. A fully documented API reference and suite of example Jupyter Notebooks have been added to our readthedocs page as follows:

https://tfv.readthedocs.io/en/latest/api_reference/index.html

https://tfv.readthedocs.io/en/latest/examples/index.html

The TUFLOW FV Python Toolbox page has been updated with new install instructions.

The following combined plot and many others can be viewed and used as a basis for further development

8.2 MATLAB Toolbox

A range of pre and post processing MATLAB tools available for free can be downloaded via the TUFLOW Downloads Page. The TUFLOW FV MATLAB Toolbox wiki page provides a tutorial to assist with the setup and use of the toolbox. Key functionality includes:

- Plug and play scripts to visualise and extract model results

* Sheet, curtain, profile and timeseries plotting
* NetCDF file handling
* GIS and mesh handling

In 2023, the TUFLOW FV MATLAB Toolbox has been enhanced to support the TUFLOW FV Water Quality Module.

8.3 TUFLOW Viewer QGIS Plugin

8.3.1 Overview

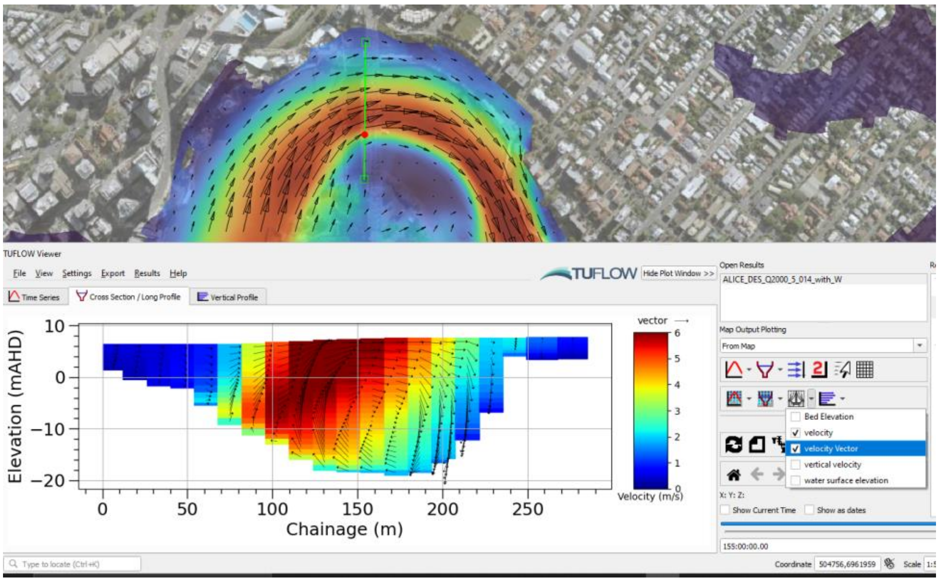
The TUFLOW Viewer QGIS Plugin provides tools to improve the efficiency of setting up, running and viewing the results of TUFLOW FV models. There is no cost associated with using the TUFLOW Viewer. Some of the available functions are described below:

* Initialise a TUFLOW FV model build, activating GIS integration
* Automated methods to create the TUFLOW FV folder directory and generate empty files
* Construct GIS layers to support assignment of boundary conditions, materials and more to a TUFLOW FV model
* Incrementing the active layer by creating a copy, assigning a new revision number, and closing the original layer
* Load and review simulation check files
* Interactively interrogate TUFLOW FV 2D and 3D results files
* Make and save images and animations of results
* Start a TUFLOW FV simulation from within QGIS

For further details on using the tools, please consult the GIS Integration section of TUFLOW FV Tutorial Model 1.

8.3.2 3D Curtain Secondary Current Vector Enhancement

Version 3.6 of the QGIS TUFLOW Viewer Plugin includes the automatic display of curtain secondary current vectors if TUFLOW FV has output the vertical velocity ‘W’ map output data type to NetCDF. The figure below shows an example velocity curtain with secondary current vectors overlaid.

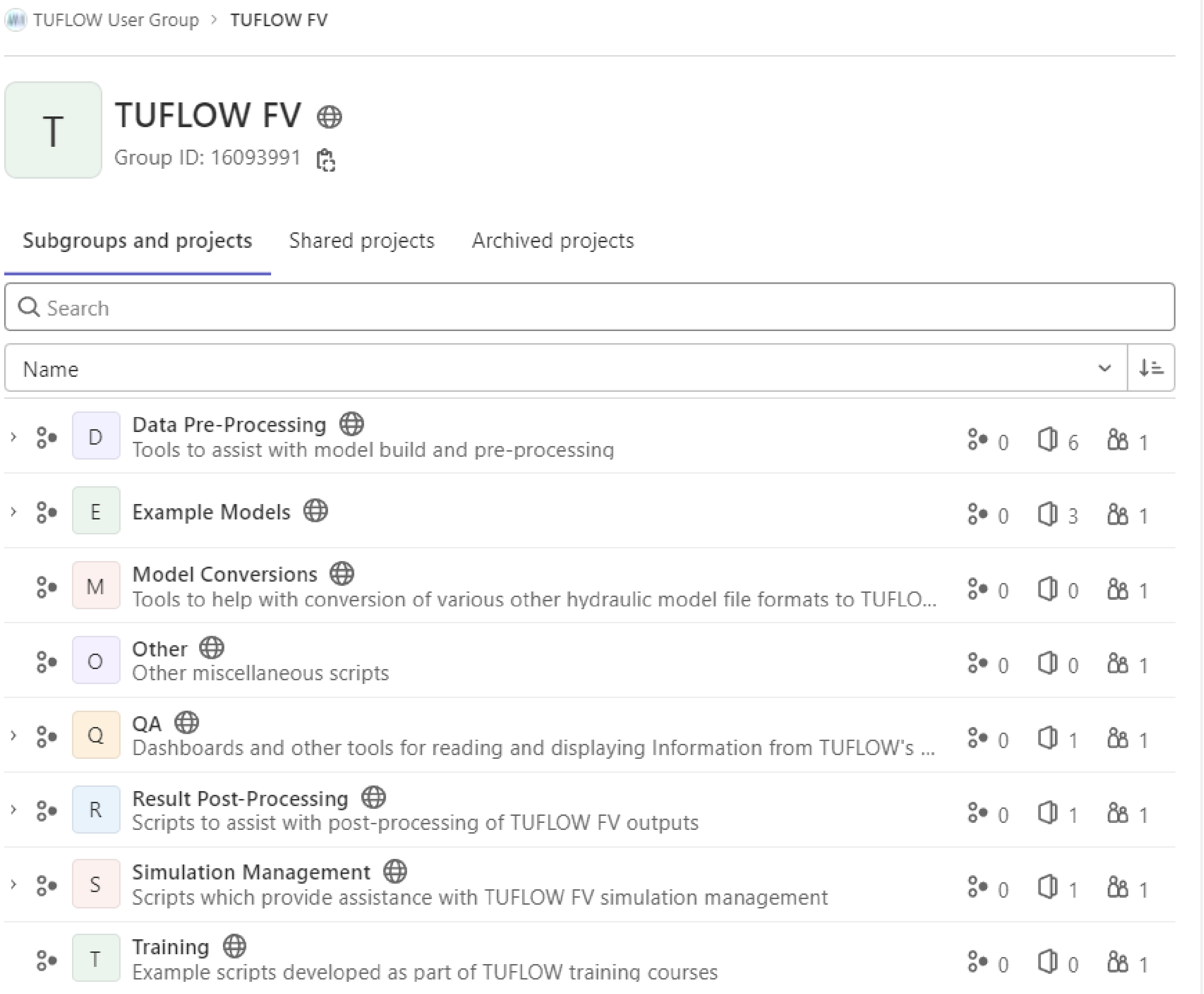


8.4 TUFLOW Open Source Gitlab User Community

8.4.1 Introduction

TUFLOW users are encouraged to explore the new scripts and tools available within the public facing TUFLOW User Group. The intent of the user group is to provide an opportunity for TUFLOW users to post their own scripts, or to find useful ones that the TUFLOW team have built, that can assist you with your project setup, boundary condition development, model simulation, model management and result processing/visualisation.

For TUFLOW FV users, the TUFLOW FV sub-group will be expanded on over time with new tools as we/you develop and share them, so watch this space!!!



8.4.2 Get Tide

A Python command line tool for developing astronomical tide curtain boundaries is available via the Get Tide project page. Please see get-tide README.md for further instructions on installation and usage.

A new tutorial wiki page is available via https://fvwiki.tuflow.com/TUFLOW_FV_Get_Tide that walks through the setup and usage of Get Tide.

8.4.3 Get Atmos

A Python command line tool for developing meteorological/climate gridded input boundary condition data such as NCEP CSFR/CFSv2 data of BoM BARRA data is available via the Get Atmos project page. Please see get-atmos README.md for further instructions on installation and usage. If you prefer ECMWF ERA5 data to source boundary conditions see also the Get ERA5 project.

A new tutorial wiki page is available via https://fvwiki.tuflow.com/TUFLOW_FV_Get_Atmos that walks through the setup and usage of Get Atmos.

8.4.4 Get Ocean

The Get Ocean Python command line tool makes it easy to download and setup models with HYCOM 3D ocean circulation model boundary conditions. These boundaries can be useful to apply mean sea level anomaly, 3D currents, 3D salinity and 3D temperature variation in space, time and depth to TUFLOW FV models.

A new tutorial is available via https://fvwiki.tuflow.com/TUFLOW_FV_Get_Ocean that walks through the setup and usage of the Get Ocean tools.

9 Mesh Generation Tools

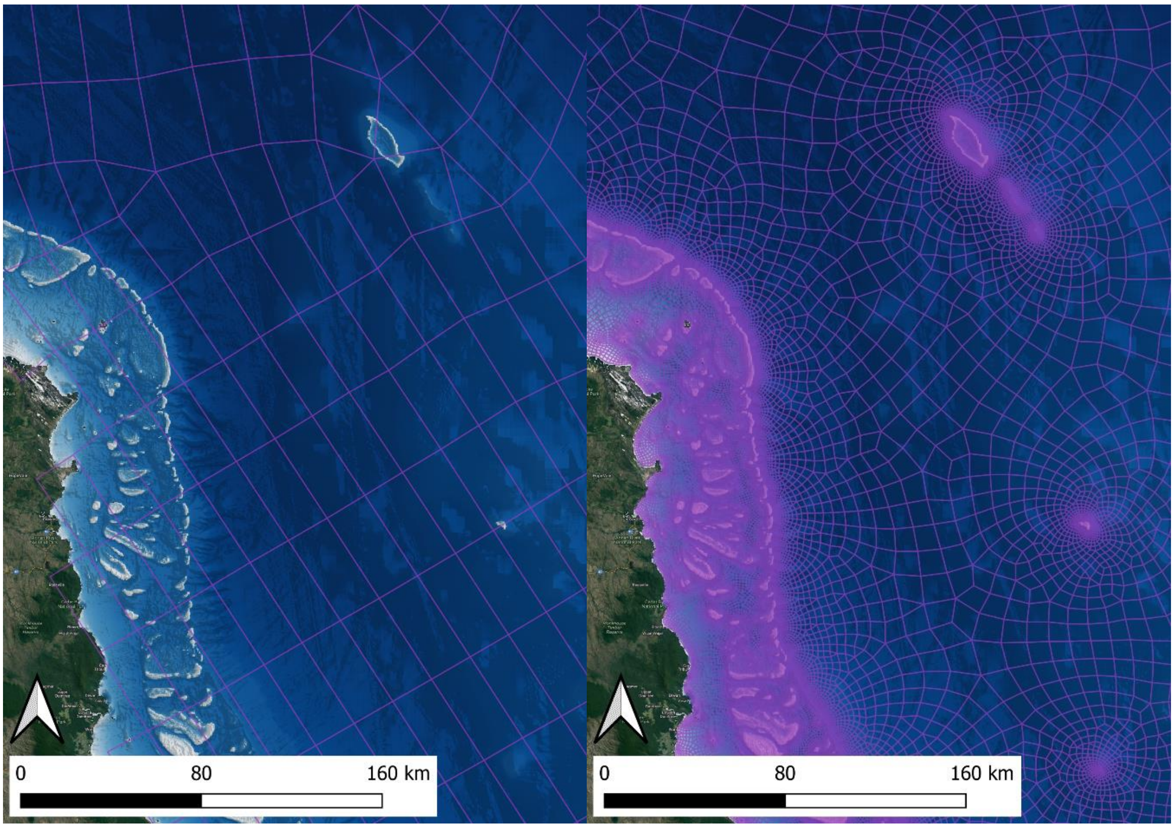
Although a separate component to TUFLOW FV, a suitable mesh generator is a fundamental component of flexible mesh modelling. The section provides a summary of recent mesher updates with links to assist in finding further information.

9.1 Rising Water Software GIS Mesher

The GIS Mesher from Rising Water Software allows users to develop and build meshes in their preferred GIS software and review it prior to running TUFLOW FV. Numerous GIS input file formats are supported including OGC GeoPackage, ESRI Shapefiles, and MapInfo GIS files (tab or mif/mid).

The GIS Mesher is designed to create quality meshes with less effort. The shape, alignment and size of mesh elements can all be controlled using GIS layers.

The Solution Guided Meshing (SGM) functionality iteratively refines meshes based upon a TUFLOW FV solution (Please see the insights article on Solution Guided Meshing). SGM varies the cell sizes based upon the change in relative depths to provide adequate resolution where needed. Cells are elongated in the direction of flow based upon user defined parameters. The figure below shows the initial mesh (left) and the resulting mesh after four iterations using SGM (right). Notice the increased resolution around large changes in relative depth (shelf or islands), with coarse cells in the deeper ocean. This is an automatic result of the SGM process.



GIS Mesher features of interest include:

- Sizes from GIS points or polylines (size parallel and perpendicular to elongate cells)
- Raster files (several formats are supported) to provide elevation data.
- Material assignment to mesh elements via GIS polygon layers
- Nodestring assignment to mesh faces via GIS polyline layers
- Automatic coordinate conversion from input files into the designated meshing coordinate system (must be a projected system in feet or meters)
- Specified output coordinate system that may be different than the meshing coordinate system
- Force cell edges (breaklines) along polylines to enforce topographic features
- Run TUFLOW FV simulations in a queue (runner capability does not require a license or registration)

When released, the 2023 version of the GIS Mesher will include a demo mode which allows meshes with fewer than 5,000 cells to be built without a license. All of the tutorial models can also be built without a GIS Mesher license including those using SGM.

For a demonstration of the GIS Mesher's functionality, please refer to the series of short videos on The GIS Mesher Youtube Channel, and check out the GIS Mesher Website and GIS Mesher Example Models.

9.2 Aquaveo SMS

9.2.1 SMS Version 13.2

Aquaveo have recently released Version 13.2 of SMS. For more information please refer to:

- https://www.aquaveo.com/software/sms-surface-water-modeling-system-introduction
- https://www.aquaveo.com/software/sms-whatsnew.

For a free trial of SMS you can download an evaluation version.

9.2.2 New TUFLOW FV Aquaveo SMS Dynamic Model Interface

A new TUFLOW FV SMS is available for Aquaveo SMS versions 13.2 and later. The interface currently supports the development of 2D hydrodynamic models, including support for TUFLOW FV NetCDF native result outputs.

It is planned to extend the interface to support basic advection dispersion tracers and sediment transport. For more information please contact support@tuflow.com and/or consult the TUFLOW FV SMS Interface Wiki Page and TUFLOW FV SMS Interface Tutorial.

9.2.3 SMS Community Edition

Aquaveo offers a community edition of SMS that provides a reduced set of functionalities to the full version, but is still useful for result viewing and mesh generation. Further information on the community edition download, features and limitations can be accessed via: https://www.aquaveo.com/software/sms-community.

10 User Manuals, Tutorials and Supporting Materials

10.1 User Manuals

10.1.1 Update of User Documentation for 2023

The new features documented within these Release Notes will be progressively added to our latest User and Science Manuals. In the interim we recommend using a combination of these Release Notes and latest manuals as described in Sections 10.1.2 to 10.1.5. Where any conflicts exist, these release notes take priority. If unsure or if you have any queries, please contact support@tuflow.com.

10.1.2 TUFLOW FV User Manual

The TUFLOW FV User Manual is the primary guide to help users understand the TUFLOW FV software ecosystem and to help with the development of TUFLOW FV models. The User Manual provides detailed instructions and examples on recommended folder structuring, control file setup and command descriptions.

10.1.3 TUFLOW FV Science Manual

The TUFLOW FV Science Manual provides detail on TUFLOW FV's underlying hydrodynamic and advection dispersion numerical scheme. There has been an upgrade to the TUFLOW FV Science Manual in that it now describes the available heat modules in detail. As in previous releases, scientific documentation concerning sediment transport and particle tracking is included in the Sediment Transport and Particle Tracking User Manual.

10.1.4 TUFLOW FV Sediment Transport and Particle Tracking Modules User Manual

A comprehensive Sediment Transport and Particle Tracking Modules User Manual is available for use with the Sediment Transport and Particle Tracking Modules of TUFLOW FV. This manual contains Scientific Documentation on the various sediment and particle model options available. The format of the Sediment Control File (.fvsed) and Particle Tracking Control File (.fvptm) is detailed with example code snippets and command appendices also provided.

10.1.5 TUFLOW FV Water Quality Module User Manual

The TUFLOW FV Water Quality User Manual provides a comprehensive online user guide for the TUFLOW FV Water Quality Module. It should be used to support all water quality model builds and simulations as it includes information beyond just commands: it is a full guide to the processes, constituents, diagnostics, demonstration models, science and set up of water quality simulations. It also includes interactive plots and network diagrams to support user understanding.

10.2 Tutorial Models

TUFLOW FV offers a range of tutorial models via our TUFLOW FV Wiki. These models can be run licence free by adding the following command to the top of the .fvc file:

```
Tutorial Model == ON
```

10.2.1 Module 1: Simple Trapezoidal Channel

The first tutorial module introduces the user to the TUFLOW FV software. The tutorial works through the steps required to develop a simple mesh model and run a simulation. The tutorial also offers an optional workflow to setup and use TUFLOW FV's GIS integration features. To access this module of the tutorial please see this page: Tutorial Module 1.

10.2.2 Module 2: Simple River Bend

The second module builds a simple river bend model using the TUFLOW FV SMS interface. To access this module of the tutorial please see this page: Tutorial Module 2.

10.2.3 Module 3: Floodplain Application

In the third module a real-world floodplain model example is used demonstrate the following:

- Mesh optimisation
- Geometry commands (e.g. break line commands, region commands)
- Optional workflow to setup and use TUFLOW FV's GIS integration features
- Hydraulic structure commands (weirs, bridges, culverts)
- Advection dispersion modelling options (salinity, heat, temperature, tracer)
- Various output processing options

To access this module of the tutorial please see this page: Tutorial Module 3.

10.2.4 Module 4: Coastal Application

The fourth module is a real-world coastal model example used to demonstrate the following:

- Application of a sloping water level boundary
- Cyclone/hurricane modelling using an internal Holland wind/pressure model
- Cyclone/hurricane modelling using an external wind/pressure and wave model
- Various output options

To access this module of the tutorial please see this page: Tutorial Module 4.

10.2.5 Module 5: 3D Estuary Application

In the fifth module a small coastal estuary is examined that demonstrates:

- Assignment of 3D layering
- Running in 3D barotropic and baroclinic mode
- Setup of the atmospheric heat module
- Coupling with the External Turbulence Model GOTM

- Introduction in the use of TUFLOW FV's MATLAB Toolbox and Python Toolbox for 3D result visualisation
- Basic particle tracking example files (provided in the tutorial download package)

To access this module of the tutorial please see this page: Tutorial Module 5.

10.2.6 Module 6: 3D Particle Tracking Tutorial

The TUFLOW Particle Tracking (PT) Module allows the 2D or 3D simulation of discrete Lagrangian particles as they are transported by the flow field (or other drivers such as wind or waves). Particle behaviour such as settling, buoyancy, decay, sedimentation and re-suspension can all be simulated. The tracking of discrete particles can be used to output particle fate and age, which is useful for purposes of animal migration, search and rescue, environmental contaminants and oil spill modelling.

The PT Module is invoked through the TUFLOW FV hydrodynamic engine which controls the overall simulation and supplies the hydrodynamic forcing to the particle transport module. Tutorial Module 06 covers a number of aspects of particle tracking modelling, including:

- Add a single set of particles via a point source and simulate their fate
- Review the outputs using the QGIS TUFLOW Viewer plugin
- Add additional groups of particles via a polygon source
- Add deposition and erosion to the particles to allow them to interact with the bed

To access this tutorial please see this page: Tutorial Module 6.

10.2.7 Module 7: Riverine Sediment Transport Tutorial

The TUFLOW Sediment Transport Module is a flexible and powerful bed load and suspended load sediment transport model that enables the 2D/3D simulation of sediment transport in rivers, estuaries and coastal environments. One or more sediment fractions can be simulated as they are distributed within the bed and transported as bed or suspended load. The tutorial investigates a riverine system and floodplain including the setup and review of:

- Modelling suspended sediment
- Erosion and deposition processes
- Bed load and bed armouring
- Coupled bed morphology
- Sediment transport visualisation tools

To access this tutorial please see this page: Tutorial Module 7.

10.2.8 Module 8: SWAN GIS Tools

The SWAN GIS Tools provide a simple graphical user interface for the development of SWAN model control and input files and are designed to streamline the development of SWAN models via easy visualisation and automated procedures in a familiar GIS setting. The tools are provided for free as a sub-menu of the TUFLOW Viewer Plugin for QGIS.

Core functionality includes the ability to:

- Generate and visualise SWAN computational grids
- Inspect many Digital Terrain Models (DTM) onto computational grids
- Pre-process wind input grids and spectral wave boundary conditions
- Detect and connect nested simulations
- Generate SWAN model control files
- Post-process SWAN results for input to TUFLOW FV

To access this tutorial and for instructions on how to setup and install the SWAN GIS Tools please see this page: Tutorial Module 8.

In addition to the SWAN GIS Tools Tutorial we have also updated the TUFLOW FV User Manual to include wave commands (refer Chapter 14: 'Waves').

10.2.9 Module 9: Water Quality Module Tutorial

The ninth tutorial model uses the basic configuration and set up of Tutorial 5 (estuary model) but extends this to focus on the set up, execution and interrogation of a suite of three water quality simulations. These simulations mirror the three Simulation Classes available within the TUFLOW FV WQ Module.

10.3 Example Models

10.3.1 Water Quality Mass Conservation Model

Mass conservation is critical to reliably executing water quality simulations. As such, the TUFLOW FV WQ Module is supported by a free downloadable model suite (under Example Demo Models) that is intended to provide users with simple three dimensional models that can be used to:

- Assess WQ Module mass conservation performance, and/or
- Support exploration of the features and behaviour of the WQ Module, and/or
- Provide templates for building water quality control files for other simulations

The suite does not need a licence to execute with either existing or altered water quality parameters. Access instructions are provided here, and descriptions and usage instructions for the associated post processing tools are here. It is strongly recommended that users who are new to the TUFLOW FV WQ Module download this mass conservation model and execute it before starting their own water quality model builds.

10.3.2 Example Flood and Coastal Models

Over 40 flood and coastal models have been added to our wiki. They can be run license-free and are available via https://fvwiki.tuflow.com/TUFLOW_FV_Example_Models.

The models include common features that are used for model development and provide a valuable reference for testing model features or experimenting before applying . We will be progressively adding more to the dataset over time. Please contact support@tuflow.com with suggestions.

10.4 Supporting Materials

10.4.1 TUFLOW FV Wiki

The TUFLOW FV Wiki is the home of our tutorial modules and includes useful guidance on:

- TUFLOW FV installation and licensing
- Model Troubleshooting
- Model development and review tips and tricks
- Useful links to user documentation and third party software products

10.4.2 TUFLOW LinkedIn User Group

If you're interested, the TUFLOW Users Group on LinkedIn is a great place to keep up to date with all things TUFLOW.

10.4.3 TUFLOW Gitlab User Community

A new user community has been created on Gitlab that contains a rich range of resources for TUFLOW FV users. These resources aim to provide (at least):

- Workflows and examples to support easier modelling workflows
- Opportunities to collaborate
- Pre-processing, post-processing and other useful scripts

10.4.4 Australian Water School Webinars

Recent recordings of the TUFLOW team presenting modelling content specific to TUFLOW FV is available as follows:

- Hydrodynamics
  - 2D Coastal Modelling
  - 3D Coastal Modelling
  - Operational Structures, Wetland Modelling
- Water Quality
  - The Future of Water Quality Modelling
  - Water Quality Modelling of Lakes
  - Coastal Water Quality Modelling
  - Pathogens
  - Integrated Catchment and Receiving Water Quality Modelling
- Sediment Transport
  - 2D and 3D Sediment Transport Modelling
  - Sediment Transport Modelling Applications

- Project Case Studies
    - Applied Hydrodynamic Modelling – Part 1
    - Applied Hydrodynamic Modelling – Part 2

11 Licensing and Installing

11.1 Windows and Linux Installation Process

To obtain the latest TUFLOW FV release and instructions on licensing and running models please refer to the New User Installation Guide.

11.2 Security Certificate

Windows builds of the TUFLOW FV 2023 release are digitally signed. This can be checked by right clicking on the .exe file and selecting properties. Under the “Digital Signatures” tab the following should be present.