



# TUFLOW Classic and HPC 2018-03 Release Notes

## Document Updates and Important Notices

(in chronological order)

**Apr 25, 2018:** The 2018-03-AA Build is a mid-year release that can significantly reduce run-times for 1D/2D linked models and includes a range of new features, enhancements and minor bug fixes. TUFLOW Classic results should be unchanged from 2017-09-AC, and HPC results should be unchanged or have very slight changes. Note there are new features or default settings that may cause changes to results (e.g. treatment of disconnected pits as virtual pipe pits, and a new option to extrapolate HPC virtual pipe pit y-Q curves), for which a backward compatibility switch or workaround is provided as described in these release notes. Also see Item 10.

**May 29, 2018:** Updated for the 2018-03-AB Build, which addresses several bug fixes. A description of bug fixes and other changes are highlighted in light green.

**October 15, 2018:** Updated for the 2018-03-AC Build, which addresses bug fixes and provides new enhancements and general functionality. A description of the changes for this revision are highlighted in light blue.

**March 15, 2019:** Updated for the 2018-03-AD Build, which addresses bug fixes and provides new enhancements and general functionality. A description of the changes for this revision are highlighted in light orange.

The majority of the changes for the 2018-03-AD version are updates to the advection-dispersion module and these changes are detailed below in Section 11.

**June 27 2019:** Updated for the 2018-03-AE Build, which addresses bug fixes and provides new enhancements. A description of the changes for this revision are highlighted in light grey.

**All 2018-03-AA to AD users should upgrade to the 2018-03-AE version, and all users of the 2017-09 release are strongly recommended to upgrade to the latest 2018-03 release.**

**Note:** The [TUFLOW Manual \(in .docx format\)](#) has been updated to align with the last 2017-09 build, therefore, these release notes cover changes since Build 2017-09-AC.

Note, this document will be updated from time-to-time with any new builds or new content.



## Licensing and Executable Versions

To run simulations using the 2018-03-AA and AB builds requires payment of the 2017/2018 annual software maintenance fee and for the TUFLOW licence to have been updated (ie. via RaC/RaU files). For the 2018-03-AC, 2018-03-AD and 2018-03-AE versions, payment, and an updated licence, for the 2018/2019 year is required. For tutorial and demo models, or if running in free demo mode, no licence is required. For any licensing enquiries please contact [sales@tuflow.com](mailto:sales@tuflow.com), or for general support [support@tuflow.com](mailto:support@tuflow.com). Use of the TUFLOW software is bound by the [TUFLOW Products Licence Agreement](#).

For the TUFLOW HPC solver, this solver is available with a standard TUFLOW licence when running on CPU hardware. To run on NVidia GPU devices a TUFLOW GPU Hardware Licence (previously referred to as the TUFLOW GPU Solver Licence) is required for each GPU device. Refer to the [TUFLOW Price List](#) for more details.

For the 2018-03 release, two executables are provided; 64-bit single precision (TUFLOW\_iSP\_w64.exe) and 64-bit double precision (TUFLOW\_iDP\_w64.exe). From the 2017-09 release onwards, no 32-bit versions of TUFLOW are provided. Note, if using the HPC solver, it is rare that the double precision version is required due to the nature of the solution scheme.

# Table of Contents

<b>Document Updates and Important Notices</b> .....	<b>1</b>
<b>Licensing and Executable Versions</b> .....	<b>2</b>
<b>1 2018-03 Release Overview</b> .....	<b>5</b>
<b>2 1D Domains</b> .....	<b>6</b>
2.1 Pits – Enhancements and Changes for 2018-03-AA.....	6
2.2 Open Channels .....	12
<b>3 2D Domains</b> .....	<b>12</b>
3.1 Topography.....	12
3.2 Computational Domain .....	14
<b>4 Boundaries and Links</b> .....	<b>16</b>
4.1 Faster run times for 1D/2D linked models .....	16
4.2 Boundary XF Files .....	16
4.3 Fixed issues in HPC with IWL at 1D nodes connected to 2D via SX connection.....	16
4.4 Delft FEWS boundary timeseries are now supported .....	16
4.5 Allow null value in rainfall timeseries .....	18
4.6 New external stress control file to allow for time varying global or spatially varying external forcing such as wind or wave radiation stresses .....	18
4.7 Extra Check for SX Connection linked to Flood Modeller Pro.....	21
4.8 Changes to Soil Infiltration and Material Imperviousness .....	21
4.9 Bug Fixes.....	22
<b>5 Outputs and Check Files</b> .....	<b>25</b>
5.1 Map Output .....	25
5.2 Time-Series Output.....	28
5.3 Grid and Cell Centred Outputs .....	29
5.4 GIS Layer Outputs .....	30
5.5 Mass Balance Outputs.....	31
5.6 Check Files .....	31
5.7 Summary and Log Files.....	32
<b>6 Simulation Control</b> .....	<b>33</b>
6.1 New 1D/2D computational effort outputs.....	33
6.2 Fixed issue with model copy and -ncf (no check file) switch .....	33
6.3 HPC version displayed if TUFLOW .exe is started with no input file.....	33
6.4 TUFLOW reports ERROR LEVEL to Windows .....	33
6.5 Enhancements to no console option .....	34
6.6 New Copy File List.....	34

<b>7</b>	<b>Minor Enhancements and Bug Fixes from the 2017-09 Release</b>	<b>36</b>
7.1	Minor Enhancements from the 2017-09 Release	36
7.2	Minor Enhancements for 2018-03-AC	36
<b>8</b>	<b>TUFLOW HPC</b>	<b>38</b>
8.1	1D Timestep	38
8.2	Repeated timesteps	38
8.3	Bug fix for HQ boundary	38
8.4	Change for SA Streams	38
8.5	New HPC Timestepping Efficiency Output	38
8.6	Minor Enhancements	39
8.7	Bug Fixes	40
8.8	Bug fix for HPC Initial Volume	40
8.9	Bug fix for should not be here at the end of simulation	40
8.10	Bug fix for first map output if using restart file	40
8.11	Bug fix for PO points inside variable z shape	40
<b>9</b>	<b>Licensing</b>	<b>42</b>
9.1	Licence free test and model copy functionality	42
9.2	Licence free package model functionality	42
9.3	More licence free models for hardware benchmarking	45
9.4	Enhancements and Bug Fixes	45
<b>10</b>	<b>Backward Compatibility</b>	<b>47</b>
10.1	With 2017-09 Release	47
10.2	Build 2018-03-AC versus Builds 2018-03-AA and AB	47

## 1 2018-03 Release Overview

The 2018-03 release is a mid-year release that includes a range of new enhancements and bug fixes. The most significant of these are discussed below.

- Significant runtime improvements for 1D/2D models, particularly for models with large numbers of SX / HX links.
- Range of enhancements for pipe network modelling including: new pit input and check layers; %blockage attribute for pits now supported; virtual pipe and 1D pipe networks can now occur in the same model; virtual pipe inlet pits now available via Classic.
- New licence free package model option which creates a model copy for all event / scenario combinations referred to in the control files.
- New, licence free, model test and copy options.
- Double precision timestep tracking for HPC 1D/2D.
- Improved HPC timestepping after repeat steps.
- Temporally and spatially varying wind stresses can now be applied to HPC and Classic
- HPC now supports all hazard output types.
- A range of new output types and improvements.

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 (subtracting the two maximum h data sets and reviewing any differences is an easy way to do this). If you have any queries on the comparison outcomes, or require clarification or more detail on any of the points below, please email [support@tuflow.com](mailto:support@tuflow.com).

## 2 1D Domains

### 2.1 Pits – Enhancements and Changes for 2018-03-AA

#### 2.1.1 Blockage now supported for Pits

As of Build 2018-03-AA, a % blockage at a pit is now supported using the 1d\_nwk pBlockage or 1d\_pit pBlockage attribute. This is a new feature for all types of pits except for TUFLOW HPC virtual pipe pits, which already supported this feature via the 1d\_pit layer.

#### 2.1.2 Pits can now be connected to Virtual Pipes and/or 1D networks in the same model

As of Build 2018-03-AA, pits can be modelled using the virtual pipe concept in Classic and HPC, and importantly, virtual pipe pits and 1D networks can now both exist in the same model. Previously, Classic did not support virtual pipe pits and HPC did not allow virtual pipe pits and a 1D network in the same model. Note, that Classic does not support the VPO virtual pipe pit as discussed further below.

#### 2.1.3 HPC now supports “Pit Default Road Crossfall ==” for Q and VPI pits

TUFLOW HPC as of Build 2018-03-AA now supports the 1D “Pit Default Road Crossfall ==” feature for Q and VPI pits.

#### 2.1.4 Changes to extrapolation of pit depth-discharge curves

Q and VPI pits are now consistent between all solvers in that the curve is automatically extrapolated using an orifice flow equation (this has been the approach in Classic since Q pits were first released). Previously, the original TUFLOW GPU/HPC virtual pipe feature would cap the flow at Q pits to the last value in the depth-discharge curve. The HPC solver now extrapolates by default in line with the approach by TUFLOW Classic’s 1D solver (ESTRY). Note, the original TUFLOW GPU solver is unchanged and will cap the flow at an Inlet (I pit).

To not extrapolate the depth-discharge curve the 1D command “Pit Default Extrapolate Q Curve == OFF” can be used. For existing HPC virtual pipe models, this command can be used to provide backward compatibility if needed. Note, this command applies to all Q pits, not just to VPI pits.

The flow regime flag output to the .eof file and \_TSF layer is as follows:

- R – Reverse flow (surcharging)
- N – Normal flow (interpolated from the depth-discharge curve)
- O – Orifice or extrapolated flow
- T – Reached the top of the curve and flow is capped (ie. not extrapolated)

Note, the above flags are only reported for Q and VPI pits computed by ESTRY. For HPC computed VPI flows, no flags are (yet) available.

### 2.1.5 New 1d\_pit Layer for TUFLOW Classic and Linked HPC Pits

Prior to Build 2018-03-AA, the 1d\_pit layer was only used for the TUFLOW GPU/HPC virtual pipes feature. The 1d\_pit layer can now be used for all types of pits in Classic and HPC, whether connected to a 1d\_nwk or a virtual pipe.

An additional (optional) attribute, Number\_of to be able to specify more than 1 pit at that connection, has been added as the 11<sup>th</sup> attribute. The 1d\_pit\_empty layer includes all 11 attributes.

The 1d\_pit layer attribute descriptions and functionality changes are provided in the table below.

Other points to note are:

- All pits processed via 1d\_pit layers are assigned invert elevations of the 2D cells they are connected to (which is via SX links). To manually specify the invert the pit must be in a 1d\_nwk layer.
- For pits connected to a channel end (usually a pipe), either by snapping or within the “Pit Search Distance ==”, the pit calculations are carried out by ESTRY irrespective of whether Classic or HPC is used.
- For unconnected pits the following logic applies:
  - VPI and VPO pits are always treated as disconnected, even if snapped or within a “Pit Search Distance ==”.
  - For Classic, all unconnected pits and VPI pits are treated as being connected to a virtual pipe. The pit’s discharge is extracted from the model, for example, if flowing into the ocean or a pipe system that does not cause any backwater or surcharging effects at that pit. Note: VPO pits in Classic are not yet supported and will show zero flow.
  - For HPC, all VPI and VPO discharges are calculated by the HPC solver as per the virtual pipe feature. For all other unconnected pits, their discharges are calculated by the TUFLOW 1D solver (ESTRY) in the same manner as for Classic above.
- Unconnected pits are now, by default, simulated and will extract water from a 2D domain (i.e. they are treated as a virtual pipe pit). A new CHECK 1625 message is issued for unconnected pits (excluding VPI and VPO pits), alerting the modeller to the possibility of a pit possibly being inadvertently not snapped or within the “Pit Search Distance ==”. Prior to Build 2018-03-AA an ERROR would occur for unconnected pits such as ERROR 1353 - No NA data for Node Pit1. If all the pits should be connected (excluding VPI and VPO pits), the user can specify the .ecf command “Pit No 1D Connection == ERROR” to force an ERROR 1626 in the case of a pit not being connected by snapping or within the “Pit Search Distance ==”.
- For the mass balance reporting, flow through unconnected pits will appear:
  - Within the \_MB2D.csv or \_HPC.csv “Es SX V In” and “Es SX V Out” SX in and out flow columns.
  - For unconnected pits calculated by ESTRY the discharge exiting the model is included in the \_MB1d.csv or \_MB.csv “H Vol Out” column.
  - For HPC VPI and VPO pits the volumes exiting or entering the model are included in the SX Volumes.

- In outputs such as the .eof file and check layers VPI (or I) pits are labelled as Q(VPI) and similarly for VPO (or O) pits. The downstream node of an unconnected pit channel will have a boundary type “V” in the .eof file as shown below.

```

NODE DATA
#      Node      BC Flag  Init Head      Connected Channel(s)
              (m)
5      Out.1      V          0.000          Out
6      Out.2      V          0.000          Out
3      Pl.1              0.000          Pl
4      Pl.2              0.000          Pl
    
```

- The velocity time-series from a HPC virtual pipe pit is now output (previously just showed as zero).
- For an unconnected pit, which must have an ID, the upstream and downstream nodes are assigned .1 and .2 extensions respectively.
- The downstream node of an unconnected pit is now assigned a nominal elevation 10 m (or 10 ft) below the pit invert (i.e. 10 m below the 2D cell elevation) – previously for HPC virtual pipe nodes this was -99999.
- The old TUFLOW GPU solver does not support these new features, but does support the first virtual pipes implementation.

**Updated Table 5-25 from 2016-03 or 2017-09 TUFLOW Manual for the 1d\_pit Attributes**

No.	Default GIS	Description	Type
	Attribute Name		
<b>Read GIS Pits</b>			
1	ID	<p>Unique identifier up to 12 characters in length. For pits that connect to a 1d_nwk channel, either by being snapped or connected using <a href="#">Pit Search Distance</a>, if ID is blank the pit channel’s ID is automatically assigned based on one of the connecting channel IDs. Note VPI and VPO pits are never connected to a 1d_nwk channel, therefore, they always require unique IDs. For any pit not connected to a channel, ID cannot be blank and must be a unique (channel) ID.</p> <p>ID may contain any character except for quotes and commas. As a general rule, spaces and special characters (eg. “\”) should be avoided, although they are accepted. The same ID can be used for a pit (which is converted to channel) and a node, but no two nodes and no two channels can have the same ID.</p>	Char (12)
2	Type	<p>“VPI” or “I” (Virtual Pipe Inlet): VPI pits are Q pits treated as being disconnected from all 1d_nwk channels (even if they are snapped to a channel). For the HPC solver VPIs can optionally be</p>	Char (4)



No.	Default GIS	Description	Type
	Attribute Name		
		<p>associated with a VPO (Virtual Pipe Outlet) to discharge flow back into the 2D domain. For the Classic 2D solver as of Build 2018-03-AA, all VPI pit discharges are irretrievably extracted from the 2D domain. Prior to Build 2018-03-AA a VPI pit could only be denoted as “I” and would not extract or surcharge virtual pipe flow unless there was an outlet (“O”) pit with the same VP_Network_ID. As of Build 2018-03-AA if there is no VPO pit with the same VP_Network_ID the VPI pit discharge is extracted from the 2D domain. “VPI” is the preferred notation.</p> <p>“VPO” or “O” (Virtual Pipe Outlet): HPC 2D solver only (not used by the Classic 2D solver). VPOs are treated as being disconnected from all 1d_nwk channels (even if they are snapped to a channel). They act as a discharge of the total or proportion of VPI inflows on the same VP_Network_ID back into the 2D domain. Prior to Build 2018-03-AA a VPO pit could only be denoted as “O”, however, “VPO” is now the preferred notation. More than one VPO pit can exist for the same VP_Network_ID.</p> <p>“C”, “Q”, “R” and “W”: As of Build 2018-03-AA, the standard C, Q, R and W pits can be specified via a 1d_pit layer, and can be connected, or disconnected, to a 1d_nwk. If a C, Q, R or W pit is not connected to a 1d_nwk channel, either by snapping or using <a href="#">Pit Search Distance</a>, the discharge through the pit is permanently extracted from the 2D domain. Alternatively, unconnected C, Q, R or W pits can be treated as an ERROR using “Pit no 1D Connection == ERROR”. For C, R and W pits, the default width, contraction and weir coefficients are used – to vary from these use a 1d_nwk layer for the pits.</p>	
3	VP_Network_ID	VP_Network ID (as an integer) used to determine which outlet(s) the flow into the inlet pits discharges from. Separate or independent networks should have a different network ID. VP_Network_ID should be a positive integer as some negative integer numbers are reserved for special purposes.	(Positive) Integer
4	Inlet_Type	For a VPI or Q pit, the name of a pit inlet type in the <a href="#">Pit Inlet Database</a> (see Section <a href="#">5.12.3</a> ). Not used for other pits.	Char (32)
5	VP_Sur_Index	Only used for VPI pits if using the HPC solver (not used for the Classic solver). In the event that the total flow into the VPIs on the same VP_Network_ID exceeds the maximum outflow capacity of the VPOs of that VP_Network_ID (see Qmax below), one or	Float

No.	Default GIS Attribute Name	Description	Type
		<p>more of the VPI flow rates must be modified. The surcharge index is used to determine how VPI flows are adjusted. The VPIs with the lowest surcharge index are given priority.</p> <p>When the total inflow of all VPIs exceeds the total capacity of all VPOs for a VP_Network_ID, the VPI inflow with the lowest surcharge index is reduced. If this VPI's inflow is reduced to zero, this VPI is permitted to become an outlet (ie. surcharge) with the flow rate up to but not exceeding its maximum outflow (see QMax below). If the total inflow of all remaining VPIs exceeds the maximum outflow plus the maximum outflow of this VPI, then the VPI with the next highest surcharge index is adjusted using the same methodology. To prevent a VPI from surcharging, set QMax to zero.</p> <p>For example, the ground level at the VPI could be used for the surcharge index, in which case the lowest VPIs surcharge first. However, if you have reliable data for the order in which they tend to surcharge in reality then this can be used. The virtual pipe network is flow conserving, but not necessarily energy conserving. The highest VPI could be incorrectly specified with the lowest surcharge index in which case water would be flowing uphill!</p> <p>VPI for the Classic solver: Not used. VPO, C, Q and W: Not used.</p>	
6	VP_QMax	<p>VPI and VPO if using the HPC solver: For VPOs the maximum flow that can discharge back into the 2D domain. For VPIs the maximum surcharge capacity or flow back on to the 2D domain. To prevent a VPI from surcharging enter a QMax of 0.0 (zero).</p> <p>If the total flow capacity (sum of QMax values for VPOs on the same VP_Network_ID) of the VPOs is reached, the inflow to VPIs is restricted and/or surcharged at the VPIs using VP_Sur_Index value above.</p> <p>R pits: Sets the height of the pit section in the vertical plane.</p> <p>VPI and VPO for the Classic solver: Not used. C, Q and W: Not used.</p>	Float
7	Width	<p>For C pits the diameter of the pit cross-section in the vertical plane. For R and W pits, the width of the pit section in the vertical plane.</p>	Float

No.	Default GIS	Description	Type
	Attribute Name		
		For all pits the width is used to determine the number of 2D cells to connect.	
8	Conn_2D	Used to control the connection type. For pits this field must be blank or set to “SX”. If set to blank, is treated as a SX connection. Other options maybe provided in the future.	Char (8)
9	Conn_No	Overwrite the automatically determined number of connected 2D cells (see Width attribute above). <ul style="list-style-type: none"> <li>• A positive value increases the number of automatically selected 2D cells by the value of Conn_No.</li> <li>• If negative, this ignores the automatic approach and fixes the number of 2D cells connected to the absolute value of Conn_No. For example, a value of -1 would only connect the pit or node to one (1) 2D cell irrespective of the width.</li> </ul>	Integer
10	pBlockage	The percentage blockage (%) of the pits. Reduces the flow capacity through the pit by the specified amount. For VPI and VPO pits this feature was introduced for Build 2016-03-AB, and for C, Q, R and W pits it was introduced for Build 2018-03-AA. Prior to these builds the pBlockage attribute was reserved and not used.	Float
11	Number_of	Number of pits. This optional attribute was introduced for Build 2018-03-AA. If this attribute is missing or is set to zero (0), a value of one (1) is used.	Integer

### 2.1.6 New 1d\_pit\_A\_check Layer

Build 2018-03-AA includes a new pits check layer containing **all** pits from all 1d\_pit and 1d\_nwk layers as point objects. The layer is called 1d\_pit\_A\_check. The first 11 attributes are the same as for a 1d\_pit layer described above, plus another three attributes as follows:

- Invert – the elevation from the 2D cell the pit connects to or that entered via a 1d\_nwk layer.
- Q\_factor – this is the multiplication factor applied to the discharge to allow for any blockage and/or if Number\_of pits is greater than 1.
- i\_1d\_layer – the integer index of the GIS layer that the pit came from.

### 2.1.7 Bug fix with Virtual Pipes and Pit Search Distance

Addresses a bug that occurs in the 2018-03-AA release if a model has virtual pipes and a non zero pit search distance (Pit Search Distance ==). For the 2018-03-AA release this would cause the simulation to fail during start up with an ERROR 1370.

## 2.2 Open Channels

### 2.2.1 Effective flow widths output to .eof file

Build 2018-09-AC now correctly shows the effective flow width values for cross-sections. Previously, the effective flow width was being output as the total (storage) flow width. The effective flow width can differ from the storage flow width where the Manning's n value varies across the cross-section. The values output to the 1d\_ta\_tables\_check.csv file are correct, so there are no changes to this output.

### 2.2.2 Bug fix for water level lines and x connectors

Build 2018-03-AD fixes a bug that would cause a "Should not be here [no\_in]" message. This can occur when an X-connector is mistakenly snapped to a channel vertex. The fix also corrects the occurrence of overlapping WLL triangles where an X-connector is used to separate channels and WLLs at a confluence. To keep WLL triangles separate and/or not generate overlapping triangles, the last WLL along the side tributary needs to have a middle vertex snapped to the side tributary's last channel end.

### 2.2.3 Bug fix for external 1D water level lines

Build 2018-03-AE fixes an issue which could occur when running TUFLOW linked to an external 1D schemes in combination with water level line output. This issue could cause TUFLOW to stop without an error message at the first map output interval. This issue was intermittent and did not impact all simulations / models.

## 2.3 Culverts

### 2.3.1 Matrix blockage fix for energy loss method

Build 2018-03-AD fixes an issue with the Matrix Blockage functionality for culverts when using the "Blockage Method == ELM". As per Section 5.12.6 of the TUFLOW manual, a modified loss coefficient is used when the culvert is flowing in outlet control regimes. When flowing in inlet control regimes a reduced area should be applied, for builds prior to 2018-03-AD, the reduced area was not applied when flowing in inlet control regime.

The "Blockage Method == RAM" is not affected by this issue.

1D flow area outputs report the flow areas used in TUFLOW.

**NOTE: All models with Matrix Blockages and "Blockage Method == ELM" must upgrade to the 2018-03-AD (or later).**

## 3 2D Domains

### 3.1 Topography

#### 3.1.1 Bug fix for variable Z Shapes

Build 2018-03-AB fixes a bug that was introduced into the 2018-03-AA version of TUFLOW for models with variable Z shapes. For models with large numbers of Zpts that are assigned as variable Z

Shape zpts, this could cause the cell code at the perimeter of the model to be incorrectly set (this shows up as spurious cells in the `_grd` check or `DEM_Z` check files. For all models with this issue observed in testing, this did not affect the model results, however, it has the potential cause spurious results.

**NOTE: All TUFLOW 2018-03-AA models with Variable Z Shapes must upgrade to the 2018-03-AB (or later).**

### 3.1.2 Bug fix for Gully Z Shapes

Build 2018-03-AC fixes bugs for the `2d_zsh` GULLY lines where the line is precisely parallel to the 2D grid alignment. The changes occur where:

- A `2d_zsh` GULLY line perfectly aligned to the 2D grid's X axis did not always change the bathymetry for double precision models.
- A `2d_zsh` GULLY line perfectly aligned to the 2D grid's X or Y axes sometimes modifies the bathymetry at cells beyond the end points of the line.

If the above bug fixes cause a change in the Zpt values, changes in results may occur from prior builds and releases. The changes in results are usually very slight due to the rare occurrence of the above situations.

The check files (e.g. [\\_zsh\\_zpt\\_check](#) and [\\_zpt\\_check](#)) correctly show the unmodified or modified Zpt values due to the above situations, therefore, any changes in Zpt values due to these bug fixes can be established by comparing the check Zpt values from prior builds with those for Build 2018-03-AC.

### 3.1.3 Bug fixes/enhancements for FC Shapes and Layered FC Shapes

Build 2018-03-AC has made the following changes for `2d_fcsh` and `2d_lfcsh` shapes:

- Bug Fix: A `2d_fcsh` and `2d_lfcsh` THIN line did not change the bathymetry correctly.
- Enhancement: The MERGE ALL option, which was previously not implemented for a `2d_fcsh` polygon, has now been implemented for Build 2018-03-AC.

With these changes in Build 2018-03-AC, `2d_fcsh` and `2d_lfcsh` shapes can now perform all the Zpt modification functions supported for `2d_zsh` layers.

The [\\_fcsh\\_uvpt\\_check](#) and [\\_lfcsh\\_uvpt\\_check](#) files respectively show the cell sides that have flow constriction shapes and layered flow constrictions applied, and any changes due to the above situations can be established by comparing these check layers from prior builds with those for Build 2018-03-AC.

### 3.1.4 LandXML TIN surfaces now support “invisible” faces

Build 2018-03-AC introduces support for invisible faces as documented on the land XML Schema (<http://landxml.org/schema/LandXML-2.0/LandXML-2.0.xsd>). Any faces in the TIN surfaces that are marked with “i=1” to indicate that they are not visible are not used for setting elevations. Prior to 2018-03-AC these faces were used for setting elevations for any Z points that fell within the faces.

Any invisible faces will cause a CHECK 2519 to be issued that refers to the GIS location of the ignored face.

### 3.1.5 Bug fix for layered flow constriction shape

Build 2018-03-AD fixes a bug introduced in Build 2018-03-AC relating to the changes outlined above in 3.1.3. This could cause the L2\_Obvert and L3\_Obvert to be incorrect for several 2d\_lfcsh configurations, these are detailed below:

- “Thick” line with Shape Option “Min”
- “Thick” line with Shape Option “Add”
- Polygon with Shape Option “Min”

The incorrect values used were reported in the \_lfcsh\_uvpt check file.

**NOTE: All TUFLOW 2018-03-AC models with Layered Flow Constrictions Shapes must upgrade to the 2018-03-AD (or later).**

### 3.1.6 Bug fix for LandXML TIN surfaces without line breaks

Fixes an issue that could cause TUFLOW to stop with no message if a LandXML TIN file had no line breaks. Build 2018-03-AD ensures that ERROR 2515 is now correctly reported and the simulation closed cleanly.

## 3.2 Computational Domain

### 3.2.1 Bug fix for memory issue with large models

Build 2018-03-AC fixes an issue with large models (typically more than 10 million 2D cells), or models with a very large number of source-area (SA) inflow boundaries, that cause a negative memory allocation to be reported in the .tlf file (see example below), and could cause TUFLOW to fail to initialise.

```

Domain_001 : Allocating Memory Pointers:
Domain_001 : Grid data memory ..... 15157 Mb
Domain_001 : Infiltration data memory ..... 3566 Mb
Domain_001 : Variable Z and Layered FC memory 0 Mb
Domain_001 : Sub-domain linking memory ..... 0 Mb
Domain_001 : Flow constrictions memory ..... 0 Mb
Domain_001 : Weirs and viscosity factor memory 1783 Mb
Domain_001 : Pressure, wind and waves memory 0 Mb
Domain_001 : General Memory ..... 0 Mb
Domain_001 : Boundary conditions memory ..... -4034 Mb
Domain_001 : Wind & waves boundary memory ... 0 Mb
Domain_001 : Hydraulic results memory ..... 19615 Mb
Domain_001 : Maximums results memory ..... 9807 Mb
Domain_001 : Mass balance tracking memory ... 0 Mb
Domain_001 : Minimums results memory ..... 0 Mb
Domain_001 : 2D grid indexing memory ..... 6241 Mb
Domain_001 : SMS High Res format memory ..... 0 Mb
Domain_001 : Plot output (PO) memory ..... 0 Mb
Domain_001 : Longitudinal output (LP) memory 0 Mb

Memory requested for 2D domain Domain_001 = 52137 Mb
Total Memory requested thus far = 52137 Mb
  
```



## 4 Boundaries and Links

### 4.1 Faster run times for 1D/2D linked models

A sophisticated indexing system has been implemented to improve the run times of models with many HX and SX links. The improvement in run time is highly dependent on the size of the 2D domain(s) versus the number of HX and SX connections. It will also be more noticeable for HPC simulations on high end GPU devices where the clock time of the 2D computational effort is substantially lower than if running on a CPU. Improvements in run time have varied from around 10% for the tutorial models to over 4,000% (i.e. over 40 times faster!) for a HPC model with 30,000 pit SX connections. This new feature benefits both Classic and HPC simulations, and will also benefit links to external 1D schemes.

For 2018-03-AA the indexing is on by default, but can be switched off using the .tcf file "Index 1D2D Links == OFF". Note that a very slight change in results can occur with indexing on, especially for single precision simulations, however, the majority of our benchmark testing showed no difference, with any changes due to numerical precision issues. For example, when summing the water levels along an SX line to take the average, the order in which SX cells are summed can be different between indexing on and off, occasionally resulting in a very slightly different average water level (i.e. fraction of a mm), especially if using single precision and/or ground elevations are very high (e.g. greater than 1,000 m). If you notice a significant change in results due to the new indexing feature, please contact us at [support@tufLOW.com](mailto:support@tufLOW.com).

### 4.2 Boundary XF Files

Removed limit on the number of rows of data in a boundary .csv file if converting to a .xf file. Prior to build 2018-03-AA there was a limit of 50,000 rows of data when converting a boundary .csv (but not .ts1) causing an ERROR 0160. When converting the .csv file to a .xf file, the file is now scanned to determine the number of rows.

### 4.3 Fixed issues in HPC with IWL at 1D nodes connected to 2D via SX connection

For a 1D node connected to a 2D cell via an SX connection, the 2D cell receives a flow boundary from the 1D model and the 2D water level is passed back to the 1D node. If the 1D node has an Initial Water Level (IWL) set and is connected to the 2D via an SX connection before the 1<sup>st</sup> timestep, a check is performed that the IWL in the 1D nodes matches the water level in the 2D cells. For TUFLOW Classic simulation this was working correctly, however, for HPC simulations the IWL of the 1D node was being ignored. After the 1<sup>st</sup> 2D timestep the 2D water level is correctly being passed back to the 1D node. This could affect the 1D results for the 1<sup>st</sup> computational timestep, but does not affect the remainder of the timesteps. For Build 2018-03-AA for HPC simulations the 1D node water levels at the 1<sup>st</sup> timestep are set based on the 2D as per TUFLOW Classic. No backward compatibility is provided for this fix.

### 4.4 Delft FEWS boundary timeseries are now supported

Boundary timeseries in the Delft FEWS boundary format are now supported. The boundary database supports boundaries in the FEWS timeseries .xml (refer 4.4.1) and .csv (refer 4.4.2) file formats. These are described below.



It is also possible to control the duration of the simulation of the NetCDF start time based on a FEWS input file – refer to Section **Error! Reference source not found.** below.

#### 4.4.1 FEWS Boundary .csv file

For the FEWS boundary in the boundary database the source column should contain the keyword “FEWS” to indicate the .csv file format is a FEWS file and not a standard .csv file, followed by a vertical bar “|” and then the filename (FEWS | <sourcefile.csv>). For example:

FEWS | source.csv

Column 1 data is not required. This is calculated based on the dates in the FEWS file, with the 1<sup>st</sup> date in the FEWS file to be TUFLOW model time 0.

In Column 2, the Location ID and Parameter ID need to be defined, separated by a vertical bar. The FEWS .csv file also supports event ensembles, an optional 3<sup>rd</sup> argument can be specified in Column 2 of the boundary database and this is the ensemble ID. In the 1<sup>st</sup> boundary database entries below a FEWS .csv file is specified with the location ID “Location1” and the parameter ID “Q.sim.hist”. The 2<sup>nd</sup> boundary entry includes a boundary ensemble number 1.

Name	Source	Column 1	Column 2
FC01	FEWS   Input.csv		Location1   Q.sim.hist
FC02	FEWS   Ensemble_Input.csv		Location2   Q.sim.hist   1

#### 4.4.2 FEWS Boundary .xml file

The .xml scheme for the FEWS timeseries is available on the DELFT website via the following link: [http://fews.wldelft.nl/schemas/version1.0/pi-schemas/pi\\_timeseries.xsd](http://fews.wldelft.nl/schemas/version1.0/pi-schemas/pi_timeseries.xsd)

To refer to a FEWS .xml file in the boundary database, the source column should contain the keyword “FEWS” followed by a vertical bar “|” and then the .xml filename.

The Column 1 (time) in the BC database does not need to be filled out, the time is calculated based on the Start Date defined in the header.

In Column 2 of the BC database the Location ID and Parameter ID need to be specified and separated by a vertical bar. An example BC database is shown below:

Name	Source	Column 1	Column 2
FC01	FEWS   Input.xml		Location1   Q.sim.hist

The header section for an input .xml is shown below. Any data in the timeseries with the value equal to that defined by the <missVal> in the header are ignored. Additional header tags are ignored by TUFLOW.

```

<series>
  <header>
    <locationId>Location1</locationId>
    <parameterId>Q.sim.hist</parameterId>
    <timeStep unit="second" multiplier="1800"/>
    <startDate date="2017-07-22" time="15:30:00"/>
    <endDate date="2017-08-30" time="15:30:00"/>
    <forecastDate date="2017-07-22" time="15:30:00"/>
    <missVal>-99.0</missVal>
    <stationName>201</stationName>
    <lat>0.0</lat>
    <lon>0.0</lon>
    <x>0.0</x>
    <y>0.0</y>
    <z>0.0</z>
    <units>mm</units>
  </header>
  <event date="2017-07-22" time="15:30:00" value="0.18" flag="0"/>
  <event date="2017-07-22" time="16:00:00" value="0.18" flag="0"/>

```

**Note:** It is not possible to define an ensemble ID using the FEWS .xml file only the FEWS .csv file format.

#### 4.4.3 FEWS input for simulation control

A new .tcf file command “FEWS Input File == <fews boundary file>” can be used to set the duration of the simulation and the NetCDF Output Start Date. The “FEWS Input File ==” command can refer to either a FEWS .csv or FEWS .xml file.

The duration of the simulation is determined from the input file and this is used to set the “End Time ==” value. If using this approach, no “End Time ==” command should be included in the .tcf file.

The start date defined in the FEWS input file is used to set the NetCDF output start date e.g. “NetCDF Output Start Date ==”.

#### 4.5 Allow null value in rainfall timeseries

Rainfall timeseries read in via the Rainfall Control File (.trfcf) can have a null value, which is user defined with the command “Rainfall Null Value == <null value>”. The default null value is -99999, and only applies to the IDW rainfall interpolation approach. If a null value is detected at a point location, the IDW interpolation is revised to ignore the location. If all locations have a null value at a boundary time, an ERROR 2642 is generated.

#### 4.6 New external stress control file to allow for time varying global or spatially varying external forcing such as wind or wave radiation stresses

This allows the user to specify one of the following types of wind / stress boundary:

- Global wind (i.e. temporally but not spatially varying wind field is applied).
- Grid Interpolation. Point winds are applied at locations within / near the model, these point winds are interpolated to gridded stresses across the model.
- User specified time varying gridded datasets

The wind time-series curve is entered as three columns of data (using Column 1, 2 and 3 labels in the bc\_dbase). The three columns are:

- Time (h)
- Wind (m/s)
- Direction (degrees relative to East, ie. East = 0°, North = 90°, etc.).

Prior to the simulation the wind boundaries are converted into a shear stress boundary and this shear stress is applied to the model. This functionality is compatible with both TUFLOW Classic (including multiple domains) and TUFLOW HPC. The shear stress is calculated based on the equation below:

$$\tau = C_{10} \times \rho_{air} \times U_{10}$$

Where:

$\tau$  is the shear stress in N/m<sup>2</sup>

$\rho_{air}$  is the density of air in kg/m<sup>3</sup>.

$U_{10}$  is the wind velocity at 10 m above the mean water surface in m/s.

$C_{10}$  is the Wind Stress coefficient and is calculated using the equation below based on Wu (1980) and Wu (1982).

$$C_{10} = (0.8 + 0.065 U_{10}) \times 10^{-3}$$

For English units, wind velocity is specified in ft/s, density is specified in lb/ft<sup>3</sup> and shear stress is calculated / output in lbf/ft<sup>2</sup>.

For the interpolated stress grids, these are output to disk at model startup and then loaded in during the model simulation as required. The grids can be used as a “check” file as they are written prior to the simulation starting. For each time in the specified boundary series two stress grids are produced, shear stress in the x direction ( $\tau_x$ ) and shear stress in the Y direction ( $\tau_y$ ). When interpolating grids from point data the wind boundary at each point is converted to  $\tau_x$  and  $\tau_y$  and these x and y component stresses are interpolated using the selected method (as described below).

Alternatively, a series of  $\tau_x$  and  $\tau_y$  grids can be directly read into the simulation. The input grids must be in the ESRI ascii (.asc), ESRI float (.flt) or NetCDF (.nc) file format. For the .asc and .flt input formats an index .csv file is input containing the time data in Column 1, and the filenames for the x and y component stresses in Columns 2 and 3 respectively. For the NetCDF format, time,  $\tau_x$  and  $\tau_y$  must all be variables in the same NetCDF file. The format of the user specified external grids is the same as produced by TUFLOW when interpolating from points to grids.

Time (hr)	Tau X component (N/m <sup>2</sup> )	Tau Y component (N/m <sup>2</sup> )
0.0	rect_TPD_Wind10_HPC_taux_t0.0.asc	rect_TPD_Wind10_HPC_tauy_t0.0.asc
3.0	rect_TPD_Wind10_HPC_taux_t3.0.asc	rect_TPD_Wind10_HPC_tauy_t3.0.asc
4.0	rect_TPD_Wind10_HPC_taux_t4.0.asc	rect_TPD_Wind10_HPC_tauy_t4.0.asc

As the user specified gridded stresses are applied as a stress term (rather than a wind boundary) these could potentially be used for external stress forcing other than winds, for example wave radiation stresses.

All stress related commands are specified in new command file, this External Stress File is referred to in the .tcf using the command.

**External Stress File** == <filename.tesf>

If this command is included TUFLOW automatically invokes the external stress functionality.

**Note:** that it is only possible to model wind stress either using an external stress file, or with a “WT” point in the 2d\_bc layer format. If both a wind point boundary (WT) and an external stress file are specified an error ([ERROR 2649](#)) will be returned.

Other .tcf commands relevant to the wind stress functionality are:

**Density of Air** == [{1.25} <value in kg/m<sup>3</sup>>]

This sets the density of air, for English units this is specified in in lb/ft<sup>3</sup>.

**Density of Water** == [{1025} <value in kg/m<sup>3</sup>>]

This sets the density of water, for English units this is specified in in lb/ft<sup>3</sup>.

**Wind/Wave Shallow Depths** == [ {0.2, 1.0} | <y1, y2> in metres or feet]

Sets the depths of water when the stress is reduced to zero. This command is necessary to avoid a divide by zero, and model instabilities when high stresses are applied to zero or very shallow depths. Below y1, the stress is set to zero, above y2 the full stress is applied, and between y1 and y2 the stress is interpolated. If not specified the default values in metres are y1 = 0.2m and y2 = 1.0m. If using English units, the defaults are converted to the equivalent in feet.

A new map output data type “tau” can be included, refer to Item 5.1.3 for more details on the new output type.

Note, that the legacy map output data type “W10” is not available for stresses applied using the external stress file and “tau” should be used instead.

In the external stress file (.tesf) one of the following commands is required to apply global, interpolated from point or gridded stresses. Note that the “Read GIS Wind Point ==” layer contains a single attribute being the Name of the wind boundary in the BC database.

**Global Wind BC** == <boundary name> !invokes a global wind boundary

**Read GIS Wind Point** == <gis file> !invokes grid interpolation

**Grid Interpolation Method** == IDW | Poly | {no default}

**Read GIS Wind Poly** == <gis file> !invokes grid interpolation

**Read Gridded Tau** == <path to .nc or grid index .csv> !invokes user specified grids

If applying a Global wind boundary, no additional .tesf file commands are required.

For the grid interpolation the following commands are applicable.

**Output Grid Format** == ASC | FLT | {NC}

Sets the output grid format to be used if interpolating point data to a stress grid. The default is to use NetCDF as this packages all output grid data for the simulation into a single .nc file.

**Output Grid Cell Size** == <value> | {10 x smallest 2D cell size}

Sets the grid size for the generated stress grids. If omitted, a value of 10 times the smallest 2D cell size is used. Typically, wind stresses can be satisfactorily represented on a much coarser resolution than that required for the hydraulic computations, therefore, using high resolution stress grids is typically not required and may unnecessarily increase memory usage, disk space and may slow down the simulation.

`Output Grid Origin == x, y !` if not specified, model origin is used

Sets the origin for the output stress grids. If this command is omitted, the model origin is used.

For the IDW interpolation method, the following options are available:

`IDW Exponent == {2.0} | <value>`

Sets the exponent term to be used in the IDW interpolation.

`IDW Maximum Distance == {not used} | <max_dist>`

This can be used to set the maximum distance for a point to be considered in the IDW interpolation, any points further than this are not used in the IDW interpolation. If not specified no maximum distance is considered.

`IDW Maximum Point == {all} | <maximum points>`

Controls the maximum number of points considered in the IDW interpolation. By default, all point locations will be used. If a very large number of point locations are provided this command can be used to reduce the memory usage. Only the closest points are used. For example, if 100 wind locations are provided, and the IDW Maximum Point is set to 20 at each output grid interpolation point only the closest 20 points are used.

#### 4.7 Extra Check for SX Connection linked to Flood Modeller Pro

For 2D SX connections linked to Flood Modeller Pro (FMP) an additional check is performed that the FMP node has been assigned as a HTBDY node. If this is not the case, a new spatial message (ERROR 2043) is issued and the simulation is stopped.

This error message can be set to a warning with the .tcf command "SX FMP Unit Type Error = OFF". If set to off, the above will cause a WARNING 2043 to be issued, but the simulation will be allowed to continue. The default for the 2018-03-AB release is an error.

Note: Build 2018-03-AC fixes an error that could cause an FMP manhole connected via a SX link to spuriously generate this warning / error message.

#### 4.8 Changes to Soil Infiltration and Material Imperviousness

Build 2018-03-AC provides additional options for handling of material imperviousness when applying soil initial losses, either using ILCL or Horton soil infiltration methods.

For 2018-03-AC two options are provided and can be set with the .tcf command:

`Soil Initial Loss == {Ignore Material Impervious} | Use Material Impervious`

The initial losses can be used to model interception losses (i.e. water that does not reach the soil), in which case the use of a material's imperviousness is not warranted and the default "Soil Initial Loss == Ignore Material Impervious" is preferred.

For the "Soil Initial Loss == Use Material Impervious" option, the initial loss is reduced by the material's fraction imperviousness.

The default approach for 2018-03-AC and prior builds is to ignore material imperviousness for the soil's initial loss component if using the ILCL or Horton approach. The exception is for materials with 100% imperviousness values (which never allow any soil infiltration to occur).

The above options only apply to the initial loss applied via the Horton and ILCL soil infiltration approaches. The continuing loss rate is not affected, and as per previous builds, the rate of CL infiltration is reduced pro-rata by the fraction imperviousness. There are similarly no changes for the Green Ampt infiltration approach (which also accounts for material imperviousness by reducing the rate of infiltration pro-rata).

The CI (Cumulative Infiltration) and IR (Infiltration Rate) map outputs are useful to show the modelled infiltration over time to check and review the final infiltration.

Also refer to the critical bug fix for soil infiltration detailed in Item 4.9.2.

## 4.9 Bug Fixes

### 4.9.1 Bug fix for rainfall initial loss in TUFLOW Classic

Build 2018-03-AC fixes a bug that affected material based rainfall initial losses (losses applied in the material .csv or .tmf file) for TUFLOW Classic simulations. The initial rainfall loss which applies at the cell centre was based on the material ID inspected at the cell side and not the cell centre material ID causing the inspection of GIS layers to be offset by ½ the 2D cell size.

This issue affects builds 2017-09-AA through to 2018-03-AB, and may cause a change in results, which testing has shown to be very minor. No backward compatibility option is provided at this stage.

The RFML (Rainfall Material Loss) output correctly shows the modelled amount of rainfall loss.

This change does not affect TUFLOW HPC simulations.

### 4.9.2 Bug fix for soil infiltration in TUFLOW Classic

Build 2018-03-AC fixes a bug with Horton method soil infiltration in TUFLOW Classic that incorrectly used the simulation time rather than the duration a cell was wet when applying the infiltration equations. For the Horton infiltration approach the time  $t$  in the equation below is now set to the period that the cell has been wet rather than the simulation time elapsed.

$$f = f_c + (f_0 - f_c)e^{-kt}$$

Where:

$t$  is time in hours (period that the cell is wet)

$f_0$  is the initial infiltration rate in mm/hr or inches/hr  
 $f_c$  is the final (indefinite) infiltration rate  
 $k$  is the Horton decay rate

For cells not wet for the duration of the simulation the bug would cause the infiltration rate to decay to the final rate faster than it should and therefore a lower amount of infiltration to occur. The amount of water being infiltrated could be checked and reviewed using the CI (cumulative infiltration) and IR (infiltration rate) map outputs.

For the ILCL and Green Ampt methods very slight changes in results may occur due to a reworking of the infiltration code to accommodate this fix.

This change does not affect TUFLOW HPC simulations.

#### 4.9.3 Bug fix for SX Region with “Z” flag

Build 2018-03-AC fixes a bug for SX regions with the Z flag specified. Previously only the cell centre (ZC) was lowered, but now the cell sides (ZU and ZV) and the cell corner (ZH) are also lowered, which is the behaviour described in Section 4.5.4 of the 2017 release notes.

The message and check files correctly identify the Zpts modified before and after the bug fix.

#### 4.9.4 Bug fix for 1D BC QT regions using the C, O and P flags

1d\_bc region objects using the C, O and/or P flags may not allocate the inflow according to the flags. The correct amount of flow is allocated, but the allocation may not be directed to the nodes intended. Build 2018-03-AD should correctly apply the 1d\_bc flow to the nodes C (closed), O (open) and/or P (bottom of a pit) nodes as intended.

Closed nodes are those nodes with only culverts/pipe (C, I or R) channels connected (with or without pits). Open nodes are those with one or more channels that are not a culvert/pipe. Pit nodes are those nodes with a pit connected, noting the flow is applied to the bottom of the pit. Any node with an SX connection is excluded from the selection process as these nodes are a 1D water level (1D HX) link/boundary, and therefore, cannot have a flow boundary applied. Any combination of C, O and P can be specified, and the 1d\_bc\_check layer and 1d\_bc\_tables\_check .csv file contain information on the nodes selected.

If C, O or P is not specified the 1d\_bc flow is equally distributed to all nodes with the region.

#### 4.9.5 Bug fix for HPC models with Gridded Rainfall Boundary in NetCDF format

Build 2018-03-AD fixes an issue when running TUFLOW HPC with gridded rainfall in the NetCDF file format. Due to precision issue this could cause the rainfall grid to be not be loaded to the HPC solver at the correct time, meaning the timing of the rainfall hyetograph applied is out by the rainfall hyetograph increment.

Testing shows that not all HPC models with NetCDF rainfall are affected. The RFC (cumulative rainfall) and RFR (rainfall rate) map output data types can be used to view the rainfall applied to the model.

**NOTE: All HPC models with gridded NetCDF Rainfall inputs must upgrade to the 2018-03-AD (or later).**

#### 4.9.6 Bug fix for models with small SX regions

Build 2018-03-AE fixes an issue with SX connections created with a GIS region object if the region is small and no 2D cells are selected (no cell centres are within the region). Prior to the 2018-03-AE update, if an SX region didn't select a 2D cell then no SX connection was created and no error / warning message was created.

For the 2018-03-AE update, if an SX region selects no 2D cells, then the cell that the region centroid falls within is used (and CHECK 2541 is issued). If the region centroid does not select a cell due to the centroid falling outside of the model domain then an ERROR 2541 is issued, and the simulation is stopped.

#### 4.9.7 Bug fix for models with overlapping 1d\_bc regions

Build 2018-03-AE fixes an issue which could apply incorrect flows from a 1d\_bc region with type QT. The issue occurs where the first (primary) node of two or more nodes within a 1d\_bc QT region also receives an inflow from another object (point or region), for which it is the only node receiving that inflow.

**NOTE: Any models with overlapping 1d\_bc regions and/or points should update to the 2018-03-AE version (or later).**

#### 4.9.8 Extra error message for unsupported boundary regions in 2d\_bc

Build 2018-03-AE fixes an issue if the user specifies either a "QT" boundary or "HX" boundary region. Both boundaries should be digitised as lines / polyline objects. Prior to 2018-03-AE the simulation would stop with ERROR 0120 - Could not find <name> in database <database file>. For build 2018-03-AE the error message is more descriptive - ERROR 2242 - Region objects not supported for 2D BC object with type <bc type>.



## 5 Outputs and Check Files

### 5.1 Map Output

#### 5.1.1 HPC now supports tracking of maximums for all TUFLOW Classic hazards

TUFLOW HPC has been extended to allow tracking of maximums for all hazard outputs as per TUFLOW Classic.

Previously the tracked maximums in HPC was limited to a subset of the hazard outputs (e.g. Z0, ZUK0, ZUK1).

#### 5.1.2 New map output data type “ZMW3”.

The existing Melbourne Water hazard outputs ZMW1 (Property Safety Risk) and ZMW2 (Safety Risk in Roads) are based on the 2008 Technical Specification. For the November 2016 Guidelines and Technical Specification, the Safety Risk In Roads classification was revised. This revised Safety Risk in Roads is available from Build 2018-03-AA as output data type “ZMW3”. The logic used is as follows:

$D*V < 0.4$  and  $D < 0.4$ : Category 1 (Low Risk)

$D*V < 0.8$  and  $D < 0.8$ : Category 2 (Medium Risk)

Otherwise Category 3 (High Risk)

Dry points are assigned Category 0.

Note that output ZMW2 is still available and this output is the Safety Risk in Roads based on the 2008 Technical Specification with the categories as documented in the TUFLOW manual.

#### 5.1.3 New map output data type “tau”. Vector output of the shear stress applied

A new shear stress output is available, this vector output contains the shear stress values applied via the external stress file (as described in Item 4.6).

The output values are in Newtons per square metre (N/m<sup>2</sup>) for SI units and pound-force per square foot (lbf/ft<sup>2</sup>) for US customary (English) units.

#### 5.1.4 HPC now support Manning’s n Map output

Manning’s n map output (for use with depth varying Manning’s n values) is now available from the HPC solver from Build 2018-03-AA.

#### 5.1.5 HPC now tracks maximum flood extent

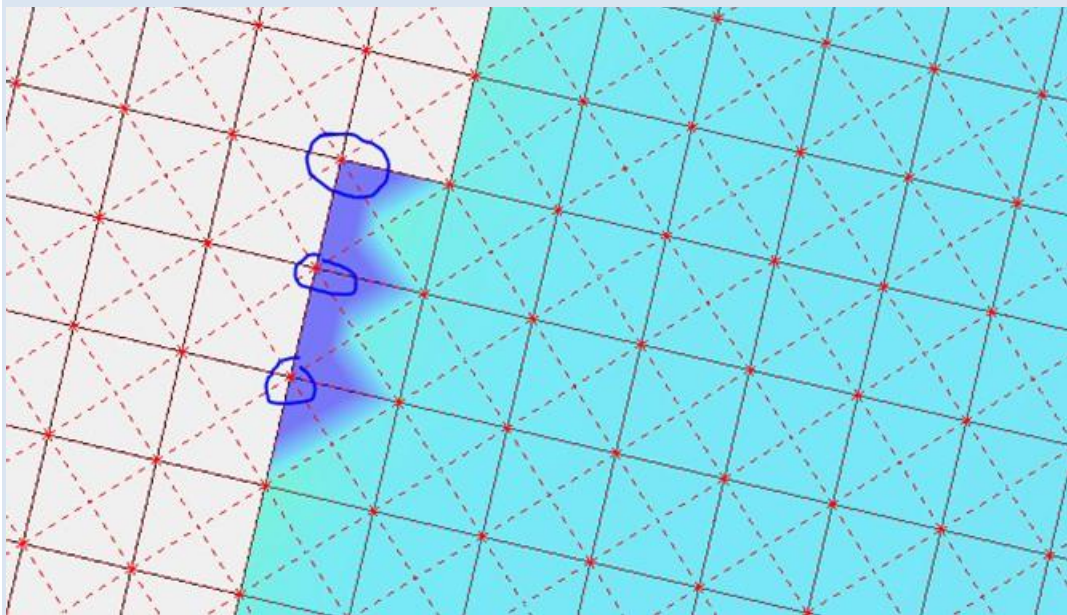
TUFLOW HPC now tracks the maximum flood extent within the HPC kernel. This may change the flooded extent where cells have very shallow flooding at or just above the wet/dry depth (default of 2 mm). Previously, the maximum flood extent could differ from the extent shown in the Time of Duration and Exceedance map outputs, which were already tracked in the HPC kernel, due to precision issues if running in single precision. The peak water levels are not affected by this change. This issue does not occur if running in double precision.

**5.1.6 TUFLOW HPC now supports tracking of maximum Energy and Stream Power**

For Build 2018-03-AB TUFLOW HPC has been extended to allow tracking of maximums for Energy (E) and Stream Power (SP) outputs. For the 2018-03-AA the maximum BSS and all hazards (Z\_) outputs were supported, but not maximum E or SP.

**5.1.7 Changes to Time Output Cutoff tracking**

For the 2018 release the tracking of the time of exceedance and duration (e.g. Time Output Cutoff Depths == 0.35, 0.50), is tracked at each timestep at both the cell centres and the cell corners. At the flood fringe, and in locations adjoining thin breaklines, where the cell is wet, but the interpolated cell corner value does not exceed the cutoff threshold strange map display issues can arise. For example, in the three corner locations circled below the threshold is not exceeded causing the dark blue shading.



For Build 2018-03-AC options have been provided as outlined in the table below to aid map display.

TCF Command	Description
Time Output Corner Interpolation == WET	The cell corner values are based on an interpolation of the surrounding wet cells. Note, this can mask the presence of thin breaklines in the output.
Time Output Corner Interpolation == VALUE   <user specified value> Example: Time Output Corner Interpolation == VALUE   -999.	The cell corners are given a fixed value which is user specified. In the example any cell corners that have not been exceeded will be given the output value of -999 and this value can be used to improve the contouring.

<pre>Time Output Corner Interpolation == START   &lt;Offset&gt;</pre> <p>Example:</p> <pre>Time Output Corner Interpolation == START   -1</pre>	<p>The cell corners are given a fixed value based on the simulation start time plus an offset. In the example if the model start time is 0 (Start Time == 0.0) any cell corners not exceeded will be given the output value of -1.0.</p>
---	--

The default for the 2018-03-AC release is to output a fixed value of -99.0 (Time Output Corner Interpolation == VALUE | -99). For Builds 2018-03-AA and 2018-03-AB the default was to output a very large negative number (-9.0 x 10<sup>29</sup>), which could cause issues with some visualisation software and file formats.

Please also see Item 5.3.3 below.

#### 5.1.8 Bug fix for time of maximum for HPC if output zones are present

Build 2018-03-AC fixes a bug for the time of maximum outputs if the HPC solver was used and an output zone is used in the model. If both output zones and HPC solver were used the time of maximum values outside of the output zone were set to 0.0. Within the output zone the values were correct.

#### 5.1.9 Bug fix for Time Output Cutoff Depth / Hazards

If time output cutoff depths or hazards were output from a model with no other cell centre outputs (which includes any raster datasets), the time of first exceedance and duration of exceedance outputs were both blank.

#### 5.1.10 Bug fix caused by different Map Output Interval for grids

If the “Map Output Interval ==” for grids is a different value to that for other formats, and “Maximums and Minimums Only For Grids == ON” then for HPC 2D only simulations the map output may not be output at the correct times. This bug has been corrected in Build 2018-03-AD and the workaround for prior builds is to remove the “Map Output Interval ==” command relevant to the grid output, e.g. “FLT Map Output Interval == 1200” as it is not used (but causes this bug!).

#### 5.1.11 Enhancement NetCDF syncing

At each map output interval the NetCDF outputs are synced to disk. This means that other processes are able to access the data as TUFLOW writes it (for example Python / Matlab scripts). Previously, data changes would be inaccessible until the TUFLOW simulation finished. This enhancement is available for both raster output (Map Output Format == NC) and cell centred (Map Output Format == CC).

### 5.1.12 New Hazard Output

Build 2018-03-AE introduces a new hazard output that is based on the City of Gold Coast City Plan Flood Overlay. This output is specified with the identifier “ZGC1” and is available for both TUFLOW Classic and HPC.

This is based on Table 8.2.8-5 “Table to acceptable outcome AO11” of the City Plan document available on the City of Gold Coast website via the link below:

[https://cityplan.goldcoast.qld.gov.au/Pages/Plan/Book.aspx?exhibit=GCCC\\_CityPlan](https://cityplan.goldcoast.qld.gov.au/Pages/Plan/Book.aspx?exhibit=GCCC_CityPlan)

The output categories and thresholds are summarised in the table below:

**ZGC1 Hazard Output Categories and Thresholds**

Hazard Category	Depth - Velocity Product (m <sup>2</sup> /s)	Depth (m)	Velocity (m/s)
Low Category 1	< 0.25	< 0.3	< 0.4
Medium Category 2	< 0.4	< 0.6	< 0.8
High Category 3	< 0.6	< 1.2	< 1.5
Extreme Category 4	>= 0.6	>= 1.2	>= 1.5

## 5.2 Time-Series Output

### 5.2.1 HPC Solver now supports Write PO Online ==

The HPC solver now supports “Write PO Online == ON” in order to write time-series and plot output results as the simulation progresses. As per TUFLOW classic these results are written at each map output interval.

### 5.2.2 HPC now outputs last time in 1D output for HPC 1D linked simulations – Build 2018-03-AA

In some situations, the last time for 1D time-series outputs did not contain any data due to precision issues associated with using a variable timestep. HPC simulations now finish slightly longer than the end time to ensure all outputs are captured.

### 5.2.3 HPC computed virtual pipe pit flow output changes – Build 2018-03-AA

- Flows and velocities in HPC computed virtual pipe pits are now signed positive if the flow is into the pit (ie. out of the 2D domain) to be consistent with linked 1D pits.
- HPC virtual pipe pit velocity estimates are now output in the same manner as for 1D linked pits (previously they would show as zero).

#### **5.2.4 Bug fix for PO region and Shapefile outputs**

Build 2018-03-AB fixes an issue that would cause TUFLOW to stop when writing time-series output for a model with plot output regions and shapefile outputs (GIS Format == SHP). This issue only occurred if there were no 1D elements in the model.

#### **5.2.5 HPC no longer writes Reporting Locations**

As documented in the 2017-09 Manual, the Reporting Location feature is not currently supported for HPC simulations. For the 2018-03-AB build if running a model with HPC and also reporting locations, the associated output .csv and GIS files are now suppressed. Prior to the 2018-03-AB release, these files were written, but did not contain correct data.

#### **5.2.6 Bug fix for PO water level points for HPC Solver**

Build 2018-03-AC fixes a bug for PO water level points located within variable Z Shapes if using the HPC solver. The previous build uses the bed elevation at the end of the simulation to obtain the PO water levels, and doesn't take into account changes in the bed elevation due to the variable Z shape. Build 2018-03-AC now uses the water depth and the bed elevation at each timestep to record the PO water levels, so now reflects the change in bed elevation due to the variable Z shape. The hydraulic calculations carried out by the HPC solver are not affected by this bug fix.

#### **5.2.7 Bug fix for PO region with Sink/Source output**

Build 2018-03-AE fixes an issue which caused the plot output regions with type "SS" (sink/source) to be written as zero (0). This issue affected both Classic and HPC simulations.

### **5.3 Grid and Cell Centred Outputs**

#### **5.3.1 Bug fix with NetCDF output raster being offset by ½ cell size**

Build 2018-03-AA fixes a bug that outputs the coordinates in the NetCDF raster at the cell corner instead of the cell centre, causing the data to be offset by ½ the cell size.

#### **5.3.2 Bug fix in BSS and SP outputs for cell centred output formats**

Build 2018-03-AA fixes an issue with the BSS and SP outputs for cell centred outputs. Water level was used instead of depth in the calculation causing the BSS and SP values to be spurious. This did not affect cell cornered outputs such as .xmdf.

#### **5.3.3 Extended the time of maximum tracking to cell centres to be consistent with other output types**

Prior to the 2018-03-AA release, the time of maximum and time output cutoffs were tracked only at the cell corners, unlike the rest of the tracked outputs, which are tracked at the cell centres and cell corners. For the 2018-03-AA release, these outputs are now tracked at the cell centres and cell corners to be consistent with other map outputs. This may mean slight changes in the extents of these outputs compared to previous builds.

### 5.3.4 Cell Centred NetCDF Output Bug fixes

Build 2018-03-AB has a number of bug fixes for the Cell Centred NetCDF output format (Map Output Format == CC). These are:

- Origin X and Y coordinates (geo\_ox and geo\_oy) now come out based on the reduced computational model grid, as per the output data. Previously the origin was based on the model origin, whilst the output grid data was based on the reduced computational grid.
- Maximums are correctly written, previously the maximum extent was incorrectly coming through based on the last timestep.
- Fixed a bug that could cause the wetted cells to occur outside the active model area for the temporal outputs (water level was based on the initial water level specified in the model control files).

**NOTE: All users of the “CC” map output data should upgrade to the 2018-03-AB (or newer) release.**

## 5.4 GIS Layer Outputs

### 5.4.1 Evacuation routes \_RCP file also supports shapefile format (previously .mif only)

Build 2018-03-AA supports output of Route Cutoff Points (\_RCP) output in the Shapefile .shp GIS format. Prior to this build if running a model with “GIS Format == SHP” the \_RCP file was output in MapInfo MIF/MID format.

### 5.4.2 Bug fix for Grid Format and GIS Format

Fixed a minor bug for that occurs if no “GIS Format == “ command is specified in the .tcf. If no gis format is specified TUFLOW checks if either a “MI Projection ==” or “SHP Projection ==” command is specified and uses this to set the GIS format for the model. Build 2018-03-AB fixes a bug that caused the “Grid Format == “ command to also set the GIS Format to MIF if no “GIS format ==” was included.

Note that the GIS Format command only controls the output format from TUFLOW, the inputs can be a mixture of MIF and SHP files with the file extension controlling the GIS format for input layers.

### 5.4.3 Added warning if no Shapefile projection exists

Build 2018-03-AC onwards now issues a WARNING 0307 if the output format is set to shapefile (GIS Format = SHP), but no SHP Projection has been specified.

Prior to Build 2018-03-AC for each output shapefile a .prj file was created, but this was blank.

### 5.4.4 Bug fix for QGIS project file

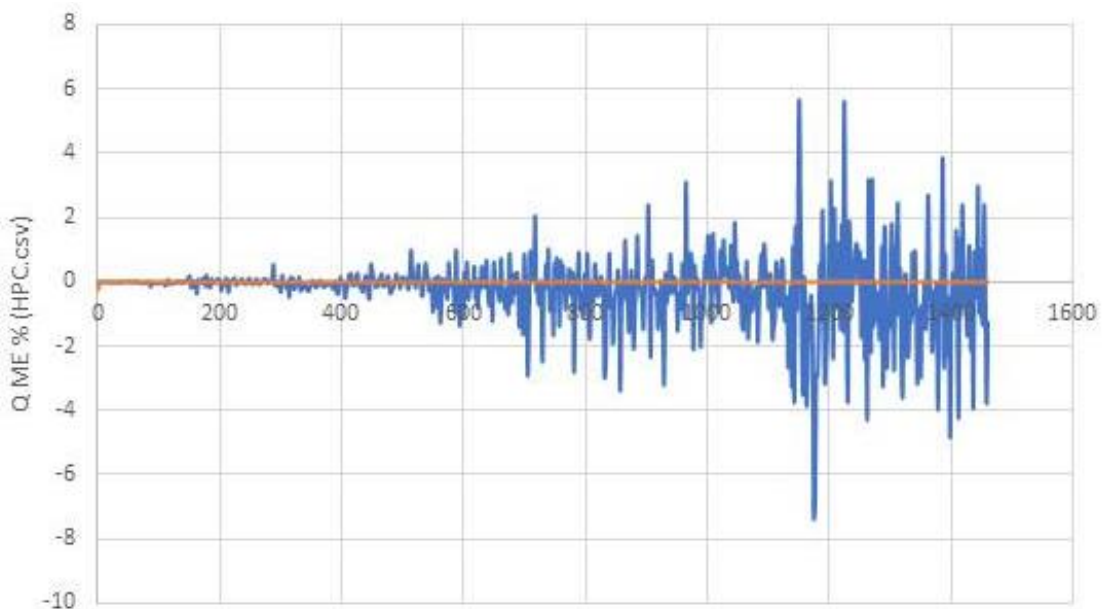
Build 2018-03-AE tidies up an issue with the legend entries in the QGIS project file (.qgs) for simulations with a large number of GIS input and output layers (more than 100). This could cause the legend entry in QGIS to show a question mark (?) instead of the layer name.

This issue was introduced for the 2018-03-AD version of TUFLOW.

## 5.5 Mass Balance Outputs

### 5.5.1 HPC tracks mass balance in double precision for single precision runs – Build 2018-03-AA

TUFLOW HPC now uses double precision for mass tracking in single precision simulations as of Build 2018-03-AA. Previously, single precision was used, which for long simulations could cause an apparent mass balance error that was due to using single precision variables, not due to the hydraulic computations. An example of the Q ME % output from the HPC.csv file for 2017-09-AC-iSP (blue) and 2018-03-AA-iSP (orange) is shown for a long simulation with substantial in and out flows. Note: the recommendation for HPC simulations is to use single precision builds unless in a rare case double precision is required.



### 5.5.2 Enhancement to HPC mass balance outputs – Build 2018-03-AA

A number of minor enhancements and additional information on HPC mass balance tracking in the HPC.csv, .tsf and .tlf files have been made for Build 2018-03-AA.

## 5.6 Check Files

### 5.6.1 Change precision of x1d check files

For Build 2018-03-AA if “Write X1D Check Files == On”, the output .csv files are written with extra precision with four digits following the decimal place instead of two (e.g. 1.2345 instead of 1.23).

### 5.6.2 Check File changes for Output Zones

Build 2018-03-AA fixes a bug that could cause Write Check File Include == or Write Check File Exclude == to produce the incorrect files if used within output zones.

### 5.6.3 Bug fix for Write Check Files Include

Build 2018-03-AE fixes a bug that caused the 1D .csv check files to be written, even if they were not included in the check file string. These are now only written if the identifiers outlined in the table below are included:

Check File Identifier	.csv Check File
1D_BC	_1d_bc_tables_check.csv
1D_PIT	_1d_pit_inlet_tables_check.csv
1D_TA	_1d_ta_tables_check.csv

## 5.7 Summary and Log Files

### 5.7.1 Fixed issue with mass balance outputs if initial and final volume were both zero

Build 2018-03-AC fixes an issue with writing mass balance outputs to the log file (.tif) and TUFLOW Summary File (.tsf) if both the initial and final volume of water in the model is zero (0). Previously this situation would cause the simulation window to disappear (crash) when writing these outputs.

### 5.7.2 Computer Name now output to Summary File

As of Build 2018-03-AC the computer name is now output to the TUFLOW Summary File (.tsf), in addition to the .tif and the simulation log files.

For example:

```
Computer Name == SuperGrunt
```

### 5.7.3 Bug Fix with QGIS project file when using Output Drive

Build 2018-03-AC fixes a bug in the QGIS project (.qgs) file if using an output drive. Prior to 2018-03-AC the path to the check files in the .qgs file was incorrect, causing QGIS to ask for the file location when opening the project.

### 5.7.4 1d\_bc check now include QT region locations

The 1d\_bc\_check layer output for 1d\_bc QT regions is now working correctly as of Build 2018-03-AD. Also see Item 4.9.4.

### 5.7.5 Bug Fix with QGIS project file

Build 2018-03-AD fixes an issue which would cause TUFLOW to write a legend entry for layers that were not opened, this could cause QGIS to crash if 'Zoom to Group' was requested. This could occur for example if the original GIS layer was not available and an .xf file was used.

### 5.7.6 Summary File bug fixes

Build 2018-03-AE update fixes a number of issues with the TUFLOW Summary File (.tsf) which occurred when running with the HPC solver, these were:



- Current Simulation time in .tsf was not being update during the run
- HPC timestep was being written as 0.0 if running with linked 1D
- HPC timestep was being written with too many decimals

### 5.7.7 Summary File default output interval

For build 2018-03-AE the TSF Update Interval now defaults to every 180 seconds (3 minutes). Prior to this update it was only written to at the start and end of the simulation. This causes the file to be updated every 3 minutes of clock time (as opposed to 3 mins simulation time).

This can be suppressed by setting TSF Update Interval == 0

The .tsf file has a TUFLOW control file style syntax and contains information on the solution scheme, hardware, primary simulation inputs, the simulation status, mass balance outputs, etc. Additional information will be added based on user feedback (please email [support@tuflow.com](mailto:support@tuflow.com) if you have a suggestion).

## 6 Simulation Control

### 6.1 New 1D/2D computational effort outputs

For 2018-03-AA, a new output file is created named “<simulation\_name>\_run\_stats.txt”. This file contains the amount of time that TUFLOW spends in the 1D and 2D computations. At each mass balance output interval, the percentage of the total computational effort that TUFLOW has spent in 1D calculations, 2D calculations and other is output to the run\_stats file. The “other” column includes a variety of tasks that are neither 1D or 2D computations, such as writing of outputs, and transfer of data to GPU (if running on GPU devices). Other also includes time spent within an external 1D solver.

This information is also output to the console window as the simulation progresses. For the 2017-09 releases this file was output only if running with “Solution Scheme == HPC”. For 2018-03-AA this output is available for both Classic and HPC solution schemes.

### 6.2 Fixed issue with model copy and -ncf (no check file) switch

Build 2018-03-AA fixes a bug that could cause check files to be written when running a model copy with the -ncf (no check file option). If “Write Check Files Include ==” or “Write Check Files Exclude ==” was specified in the .tcf file, these check files would be written even if the -ncf option was specified.

### 6.3 HPC version displayed if TUFLOW .exe is started with no input file

For Build 2018-03-AA if the executable is started with no input file (e.g. double click on the .exe) then the HPC version is displayed in the console output.

### 6.4 TUFLOW reports ERROR LEVEL to Windows

If TUFLOW exits unsuccessfully, e.g. an error during initialisation or due to the model going unstable an ERRORLEVEL is reported to the Windows operating system. This value is 1 if a premature exit has been encountered and 0 for a normal exit. This can be useful for tracking simulation issues if

using batch files, scripts or a simulation run manager. In a batch file the error level can be obtained with the %errorlevel% variable. For example:

```
Start "TUFLOW" /wait TUFLOW_iSP_w64.exe runfile.tcf  
echo error level is %errorlevel%
```

Prior to the 2018-03-AA version of TUFLOW no error level was reported.

## 6.5 Enhancements to no console option

Build 2018-03-AB has some enhancements when running with the -nc (no console option). These are designed to remove any user input to TUFLOW.

- If no .tcf is specified, no licence check is performed and the simulation is halted. Previously a licence check was performed and the user was prompted to hit a key to release licences and close the simulation.
- If an invalid .tcf file is specified, the simulation stops and returns an error level of 1 to the operating system. Previously the user was prompted to enter a valid .tcf
- If the set log path (-slp) is specified in conjunction with the no console option (-nc) the simulation stops and returns an error level of 1 to the operating system. Previously the user was prompted to confirm the set log path.
- Removed a number of locations where a simulation could pause and wait for a user input before closing.

### 6.5.1 Bug fix for No Console

Build 2018-03-AE fixes an issue with the -nc option that occurred if the "-nc" command line argument was specified after any scenario or event arguments. Prior to the 2018-03-AE update the following would not recognise the no console option:

```
TUFLOW_iSP_w64.exe -e1 QPMF -nc runfile ~e1~_001.tcf
```

However, the command line below would work as expected:

```
TUFLOW_iSP_w64.exe -nc -e1 QPMF runfile ~e1~_001.tcf
```

For the 2018-03-AE update either option will work.

## 6.6 New Copy File List

When running TUFLOW with the copy model input switch (-c) a new list option is available, this will output the files used by TUFLOW into a .tcl (TUFLOW Copy List) file but not copy the files to a destination folder. This can be useful if scripting the copying of models. To run the copy list the character "L" needs to be specified after the -c input argument. This works for all copy options, for example the following are all valid; -cL, -caL -capL

The .tcl file produced is output in the same directory as the .tcf and takes the simulation name.

This function was introduced in Build 2018-03-AB.

#### **6.6.1 Include projection files**

Build 2018-03-AC fixes a bug that would cause projection .prj and .mif files to be skipped in the copy list.

#### **6.6.2 Bug fix for .tcl filename**

Build 2018-03-AC fixes a bug that would cause the output .tcl to have an incorrect filename if the simulation file name had multiple "." characters.

For example "FMA\_T2\_CLA\_CPU\_NT1\_20.00m\_001.tcf" would incorrectly create a file named FMA\_T2\_CLA\_CPU\_NT1\_20.tcl and not FMA\_T2\_CLA\_CPU\_NT1\_20.00m\_001.tcf

#### **6.6.3 Mapinfo .tab files are now listed**

Build 2018-03-AC now correctly lists MapInfo .tab files in the .tcl if using "Check MI Save Date == ERROR" (the default) or "Check MI Save Date == WARNING". Prior to this unless using the copy all flag the .tab file was omitted.

#### **6.6.4 Correct File Case**

When writing to the copy list (.tcl) file, Build 2018-03-AC checks the case of the filenames and outputs the correct case. This may be important if using a case sensitive operating system (e.g. Linux) for copying / moving files.

## **7 Minor Enhancements and Bug Fixes from the 2017-09 Release**

### **7.1 Minor Enhancements from the 2017-09 Release**

#### **7.1.1 Fixed bug that incorrectly didn't allocate PO memory for VU PO type**

If the VU PO type was specified, the VU output in the PO.csv files could be incorrect – this was corrected for Build 2018-03-AA.

#### **7.1.2 Bug fix where no pump or pit data output to .eof file if no other structures**

Build 2018-03-AA fixes a minor bug that did not output pump or pit data if there were no B, W, R, C or I structures.

#### **7.1.3 Mass balance reporting bug fix at 1D nodes with a HT or HQ boundary and a 2D HX link**

Build 2018-03-AA fixes a bug that did not correctly perform mass balance calculations on a 1D node with a HT or HQ boundary **and** a 2D HX link. The flow via the 2D HX link (or a Flood Modeller 1D H link) was not accounted for and would, if the flow was substantial, cause erroneous mass balance reporting.

#### **7.1.4 Bug fix for IDW rainfall interpolation**

Build 2018-03-AA fixes a bug that occurred when creating a gridded rainfall from point hyetographs if the rainfall gauge location is exactly on the raster output grid cell centre location. Prior to Build 2018-03-AA this could cause a Not a Number (NaN) to be output to the rainfall grids due to a divide by zero.

#### **7.1.5 Bug fix for TUFLOW AD simulations**

Build 2018-03-AA fixes an issue when using TUFLOW AD (Advection-Dispersion) module. This could cause an unexpected end of file to be encountered when reading the AD Global BC Database. This error is due to a known Fortran v17 compiler issue that does not correctly detect the end of a file, and unfortunately occurs at random. This issue is related to Item 3.2 of the [2017 release notes](#) and could be present in all 2017-09 versions of TUFLOW.

### **7.2 Minor Enhancements for 2018-03-AC**

#### **7.2.1 Changes for multiple concurrent simulations**

Running multiple simulations that start at the same (clock) time, could cause issues during start-up. These issues had been reported by users running on “cloud” or datacentre infrastructure, due to a race condition between concurrent simulations. For example, if two simulations start at the same time, the existence of output directories is tested, and if the output folder does not exist, both simulations can try to create the output folder. However, only one will be successful and the other simulation will fail. Build 2018-03-AC removes the possibility of race conditions occurring.

#### **7.2.2 Successful copy or test model reported to the simulation .log**

Build 2018-03-AC reports a successful model copy or model test to the simulation .log file. Previously this was only reported in the log file (.tlf) for each simulation. If testing / copying a number

of simulations the user can now check the status of all these in the simulation .log file, rather than having to open each .tlf file individually.

### 7.2.3 Bug fix for stderr output stream

Build 2018-03-AC removes spurious output of the text “Special Channel Data” to the standard error (stderr) output stream, which could occur for a model with hidden 1D nodes. This does not affect computations or model outputs.

### 7.2.4 Additional Empty files

Build 2018-03-AC outputs an empty (template) GIS file for the following file types:

- 2d\_ztin, as used by “Create TIN Zpts == “
- 2d\_at, as used by “Read GIS Auto Terminate == “

### 7.2.5 Wiki message links now link to secure https:

For Build 2018-03-AC all TUFLOW Wiki message links have been changed from http:// to the more secure https:// protocol. For example:

[http://wiki.tuflow.com/index.php?title=TUFLOW\\_Message\\_2051](http://wiki.tuflow.com/index.php?title=TUFLOW_Message_2051)

has been replaced with:

[https://wiki.tuflow.com/index.php?title=TUFLOW\\_Message\\_2051](https://wiki.tuflow.com/index.php?title=TUFLOW_Message_2051)

### 7.2.6 Bug fix for “XF Files Include ==” in Filename

Build 2018-03-AC fixes a bug with the “XF Files Include in Filename ==” option. The command was correctly read in and output to the .tlf, but was previously not used for the naming of .xf files.

### 7.2.7 End of File bug fixes

Fixed several instances where an incorrect end of file during a file read occurs, due to a known issue with the Intel Fortran v 17 compiler. This could occur for the Read RowCol Mat == command or when processing a gauge level output (2d\_glo) .csv file. Any model experiencing this issue would fail to start, and will report an “Unexpected End Of File Encountered”.

Note: See also Item 7.1.5 above.

### 7.2.8 Bug fix for CHECK 2108 and MapInfo output

Build 2018-03-AC fixes an issue that prevented “CHECK 2108 – 2D HX link applied more than once at HX cell.” from being correctly written to the GIS messages layer when using the MapInfo format. The message was still output in the messages.csv file and also the .tlf.

### 7.2.9 Bug fix for Z Shape if attribute data is not correct

If a 2d\_zsh layer does not have the correct number and type of attribute fields (as defined in the manual), TUFLOW now correctly outputs ERROR 2172. Prior to 2018-03-AC if the attribute fields were incorrect TUFLOW would stop with a “Should not be here” error.

## 8 TUFLOW HPC

### 8.1 1D Timestep

For variable timestep simulations Build 2018-03-AA now tracks the 1D solutions simulation time using double precision for single precision runs (previously, single precision variables were used). This can slightly change HPC 1D linked model results from the 2017-09 builds, especially for simulations with large numbers of timesteps. **There is no backward compatibility option for this change.** Fixed timestep simulations are not affected and can adequately track time in single precision.

### 8.2 Repeated timesteps

Build 2018-03-AA includes a new HPC command "Timestep Repeats == [ <n\_allowed\_repeats> | {10} ]" should the number of repeated timesteps in one step need to be changed. One situation where this might be useful is if a 2D only model remains totally dry for some time the timestep can become very large, and then needs to be rapidly reduced once inflows commence. Build 2018-03-AA also includes an improved timestep reduction approach that handles this situation much better (which may cause very slight, fractions of a mm, differences in results), however, in very rare situations you may need to increase the limit of 10 repeats using this command.

### 8.3 Bug fix for HQ boundary

Build 2018-03-AB fixes a bug in the HQ boundary for the HPC solution scheme that in rare instances could cause the timestep to drop to zero and stop the simulation. This bug fix may cause slight changes in the timesteps used for all HPC models with HQ boundaries and therefore may cause typically minor differences when compared to 2018-03-AA.

**NOTE: All HPC simulations with HQ boundaries are recommended to update to the 2018-03-AB (or newer) version of TUFLOW.**

### 8.4 Change for SA Streams

For Build 2018-03-AB if inputting SA boundaries with the Streams Only option (Read GIS SA Streams Only == ), if no streams are present inside the SA polygon, HPC now reverts to a standard SA inflow approach. TUFLOW will issue a WARNING 2208 and the simulation will continue. This is consistent with the behaviour in TUFLOW Classic, prior to this if no stream cells were found inside the polygon an ERROR 2442 message was issued and the simulation was halted.

### 8.5 New HPC Timestepping Efficiency Output

From Build 2018-03-AC onwards a timestep efficiency is output for HPC simulations, this is reported to the console window, the HPC log file (.hpc.tlf) and the timestep history file (.dt.csv). This contains a cumulative assessment of the timestepping efficiency. A value of 100% indicates that the HPC timesteps are perfectly aligned with the minimum stability timestep for complying with the three control numbers representing the Courant, Wave Celerity and Diffusion criteria. Factors that reduce the timestepping efficiency are:

- Lower initial timestep than required. This will be evident by low values at the start, then steadily increasing towards 100% as the timestep approaches the optimum value – see example further below.

- Synchronisation with a 1D scheme. The 1D/2D linking with TUFLOW's 1D solver (ESTRY) is designed to synchronise exactly using the larger of the 1D or 2D timesteps. For external 1D schemes, the timestepping is similarly synchronised, with some external 1D schemes also offering an unsynchronised option. In all cases, the HPC 2D timestepping will be below optimum by varying degrees, with the inefficiency shown in the timestepping efficiency output.
- Frequent map or time-series output. The timestepping is set up to align exactly with map and time-series output intervals, therefore, timesteps nearly always need to be reduced as an output interval is approached.
- Using a fixed timestep.

For example, in the output window below, the efficiency is initially poor (~40%) due to the initial timestep in the model being too low, however the efficiency rapidly increases as the HPC timestep increases.

**Note:** Should your model exhibit low overall timestepping efficiency (i.e. values less than around 70 to 90%), please let us know via [support@tufLOW.com](mailto:support@tufLOW.com), and attach the .tlf, .hpc.tlf and dt.csv files. This feedback is of special interest for 1D linked models, as there may be opportunities for further optimisation in the 1D/2D timestepping.

```

TUFLOW Build: 2018-03-AC-iSP-w64 Control File: C13_q1000.tcf Simulation: C13_q1000_HPC+2018-03-AC GPU Devices: 0
Writing HPC Output at: 0:00:00 Clock: 0:00:12 CPU: 0:00:09
HPC:  iStep  time  maxNu  maxNc  maxNd  nWet  vol  dt  Eff
HPC:  60    0:00:12  0.004  0.588  0.006  85    213.338  0.329  39.3%
HPC:  120   0:00:42  0.008  0.999  0.011  85    213.338  0.549  78.0%
HPC:  180   0:01:14  0.008  0.999  0.011  85    213.338  0.549  87.7%
HPC:  240   0:01:47  0.008  0.999  0.011  85    213.338  0.549  91.5%
HPC:  300   0:02:20  0.008  0.999  0.011  85    213.338  0.549  93.4%
HPC:  360   0:02:53  0.008  0.999  0.011  85    213.338  0.549  94.7%
HPC:  420   0:03:26  0.008  0.999  0.011  85    213.338  0.549  95.5%
HPC:  480   0:03:59  0.008  0.999  0.011  85    213.338  0.549  96.1%
HPC:  540   0:04:32  0.008  0.999  0.011  85    213.338  0.549  96.6%
SIM:  0:05:00  -d 0 0 0D 7% 1D 3% 2D 90% CE -0.0% 0.0% -0.0% Vi 0 Vo 0 dV 0
Writing HPC Output at: 0:05:00 Clock: 0:00:24 CPU: 0:00:22
HPC:  600   0:05:05  0.008  0.999  0.011  85    233.583  0.548  97.0%
HPC:  660   0:05:38  0.142  0.999  0.107  85045  366.499  0.548  97.2%
HPC:  720   0:06:11  0.186  0.999  0.138  104489  496.475  0.548  97.5%
HPC:  780   0:06:44  0.179  0.999  0.127  116070  623.659  0.548  97.7%
HPC:  840   0:07:17  0.210  0.999  0.150  123519  748.362  0.548  97.8%
HPC:  900   0:07:49  0.219  0.999  0.134  129444  870.806  0.548  98.0%
HPC:  960   0:08:22  0.223  0.999  0.136  133860  991.014  0.548  98.1%
HPC:  1020  0:08:55  0.237  0.999  0.138  137145  1109    0.548  98.2%
HPC:  1080  0:09:28  0.250  1.000  0.138  139697  1224    0.548  98.3%
SIM:  0:10:00  -d 0 0 0D 5% 1D 4% 2D 91% CE -0.0% -0.0% 0.0% Vi 1230 Vo 4 dV 1226
Writing HPC Output at: 0:10:00 Clock: 0:00:49 CPU: 0:00:46
HPC:  1140  0:10:01  0.233  1.000  0.140  152383  1342    0.548  98.4%
HPC:  1200  0:10:34  0.269  1.001  0.142  154381  1607    0.549  98.5%
HPC:  1260  0:11:07  0.279  1.000  0.144  156806  1864    0.547  98.6%
HPC:  1320  0:11:40  0.296  1.003  0.147  159055  2113    0.547  98.6%
  
```

## 8.6 Minor Enhancements

### 8.6.1 Precision enhancements

Build 2018-03-AC includes some minor precision enhancements to minimise the very slight differences that can occur between running on CPU or GPU hardware due to use of different compilers. Very slight changes in results may occur in HPC runs due to these changes.

## 8.6.2 Extra Warning Messages

Build 2018-03-AE introduces new warning messages if the HPC solver has been used with “Defaults ==” to set the default to PRE 2017. The TUFLOW HPC solver was only introduced in 2017, so the use of Defaults prior to this is unusual.

In particular, if “Defaults == PRE 2016” (or earlier) is used in conjunction with SA boundaries, these are applied as SA ALL, which is consistent with the (old) TUFLOW GPU solver (the precursor to TUFLOW HPC). In this instance a WARNING 2787 message is generated.

## 8.7 Bug Fixes

### 8.8 Bug fix for HPC Initial Volume

Build 2018-03-AC fixes an issue with the initial mass balance for HPC models. The volume at the start reported in the log file and console output was at the 1<sup>st</sup> mass balance output interval, and not at the start of the simulation. This was also the 1<sup>st</sup> output in the MB.csv file.

### 8.9 Bug fix for should not be here at the end of simulation

Build 2018-03-AC fixes an issue that could cause a “Should not be here” message to occur at the end of a HPC simulation. This was to do with a precision issue when checking if a simulation had reached the end. This situation only occurs for the 2018-03-AA or 2018-03-AB versions of TUFLOW.

The error reported was:

“Should not be here [tE=<time>, tN<time>]. Please contact [support@tuflow.com](mailto:support@tuflow.com)”.

### 8.10 Bug fix for first map output if using restart file

Build 2018-03-AC fixes an issue if the map output is written immediately after reading a restart file using the HPC solver. In this case the wetted extent and output values were incorrect, however, any subsequent outputs were not affected.

### 8.11 Bug fix for PO points inside variable z shape

Build 2018-03-AC fixes an issue with water level plot output if the PO point was located on a cell that had also had a variable z shape modifying the elevation during the simulation. Prior to 2018-03-AC the water level reported was the tracked depth over time plus the final model elevation. From 2018-03-AC the water level is tracked within the HPC solver.

### 8.12 Bug fix for negative rainfall

2018-03-AD fixes an issue with negative rainfall being applied to dried cells. This could result in large numbers of Warning 3991 – negative Depth at <cell reference>, to be output from the simulation resulting in long runtimes and large log files. For the 2018-03-AD version negative rainfall is not applied to cells that have become dry.



### 8.13 Bug fix for Boundary viscosity factor

Build 2018-03-AD fixes an issue where a user specified “Boundary Viscosity Factor == “ was not correctly applied in the HPC solver.

### 8.14 Bug fix for constant viscosity

Build 2018-03-AE fixes an issue with specifying “Viscosity Formulation == CONSTANT” for a TUFLOW HPC simulation.

TUFLOW HPC solver does not recognise a separate “constant” viscosity formulation, but constant viscosity can be modelled with the Smagorinsky formulation by setting the variable Smagorinsky component to zero (0) and the constant component to the desired value. The values used in the HPC solver are displayed in the .hpc.tif file as per the below:

```
Smagorinsky turbulence model selected
Setting nut_m = 0.000 ! this is the value of the variable component
Setting nut_c = 1.000 ! this is the value of the constant component
```

Prior to the 2018-03-AE update if setting the following:

```
Viscosity Formulation == CONSTANT
Viscosity Coefficient == 1.0
```

The HPC solver would use a Smagorinsky formulation with the coefficient specified applied as the variable component, not the constant component (which used 0). This is the same as applying:

```
Viscosity Formulation == SMAGORINSKY
Viscosity Coefficients == 1.0, 0.0 ! Variable, Constant components
```

The modelled value used was output if using the viscosity coefficient map output type “t”, e.g. Map Output Data Types == t.

For the 2018-03-AE update, if “Viscosity Formulation == CONSTANT” has been specified in the .tcf file, the coefficient specified is passed through to the HPC solver as the constant component, and the variable component is set to 0.

This issue does not affect Classic simulations.

### 8.15 Bug fix for POMM (min / max)

Build 2018-03-AE fixes an issue with the minimum and maximum values in the POMM.csv file to be written as 0.0 for some plot output types. Note that For TUFLOW HPC the plot output maximums and minimums are not tracked at each timestep, rather these are extracted from the time-series values at the end of the simulation, this is identified in the maximum output files.

### 8.16 Bug fix for repeated timesteps

Build 2018-03-AE fixes an issue with the number of repeated timesteps in the summary produced at the end of the simulation being set to zero (0). Prior to 2018-03-AE the summary output in the .tif, .tsf and the console window reported 0 repeated timesteps (either due to a high control number or

NaN) even if repeated timesteps occurred during the simulation. The messages outputs and the .hpc.tlf file correctly reported any repeat timesteps.

### 8.17 Bug fix for layered flow constriction message

Build 2018-03-AE fixes a minor bug that incorrectly reported "Layered Flow Constrictions not yet supported..." to the console window and .tlf file for HPC models with no layered flow constrictions instead of "No Layered Flow Constriction data to transfer...".

## 9 Licensing

### 9.1 Licence free test and model copy functionality

For Build 2018-03-AA it is possible to use the model copy (-c option) or test model (-t option) without using a licence. To utilise this licence free copy / test, the -nlc (No Licence Check) input argument must be specified, e.g. the command line below would run the model in test model (-t) without needing a licence (-nlc):

```
TUFLOW_iSP_w64.exe -t -nlc runfile.tcf
```

The -nlc option also works for all copy options, e.g; -c, -ca, -cap.

If running without a TUFLOW licence no diagnostic output is generated (e.g. messages layer). If these are required, the -nlc option must be removed.

#### 9.1.1 Bug fixes for licence free copy / test mode

Build 2018-03-AC fixes an issue with GPU licencing and licence free test / copy. Prior to 2018-03-AC, if multiple GPU devices are specified in the control file or command line TUFLOW would stop to say that insufficient GPU licences were available. For 2018-03-AC no GPU licences are required if running with the -nlc for testing / copying of TUFLOW models.

### 9.2 Licence free package model functionality

The 2018-03-AA introduces a new option for packaging model inputs, this is specified with the -pm input switch. An example command line is below:

```
TUFLOW_iSP_w64.exe -pm PR_~s1~_~e1~_~e2~_001.tcf
```

This can be used instead of the copy (-c) switch. The copy option:

- Works for a single event / scenario combination
- Checks that all inputs are valid for the simulation, and so can take time to initialise.

In contrast the package model function, does not do any checks on the validity of the data. It attempts to copy input files for all events and scenarios defined in the model.

The package model function does not require access to a TUFLOW licence.

Because the data is not read and processed the -pm option is much faster than using the model copy (-c) option. Should a file not be copied using the -pm option, please advise [support@tuflow.com](mailto:support@tuflow.com) including details of the file(s).

Optional switches are:

- “All” (e.g. -pmAll) copies all file extensions (e.g. 1d\_nwk\_culv\_L.mif, will become 1d\_nwk\_culv\_L.\*).
- “L” (e.g. -pmL) list the files to be copied into an output file, but don’t copy.
- “ini” (-pmini <file.ini>) provides a .ini file with user defined options as described below.

Combinations of the above are also valid, with the order of the optional switches not being important (-pmAllL would be treated the same as -pmLAll).

Three options are available for handling of the binary processed files created by TUFLOW to speed up the simulation start. These options are:

- -xf0 Do not copy .xf files, only the original inputs are copied.
- -xf1 Copy both raw input files and .xf files.
- -xf2 Copy only .xf files, if xf files exist for an input only the xf will be copied.

For example, the command line below will package the model but not include any .xf files.

```
TUFLOW_iSP_w64.exe -pm -xf0 PR_~s1~_~e1~_~e2~_001.tcf
```

When packaging a model, all input files are copied relative to the base model folder. This is done to ensure that the relative file paths are maintained. By default the base folder is two folder levels above the .tcf file. For example, if the .tcf file is in “C:\Projects\Modelling\TUFLOW\runs\Run\_001.tcf”, the base folder for the model packed would be “C:\Projects\Modelling”.

When using package model the default destination folder is created in the same directory as the .tcf file, with the prefix “pm\_”. For example, C:\Projects\Modelling\TUFLOW\runs\Run\_001.tcf will create a package in the folder “C:\Projects\Modelling\TUFLOW\runs\pm\_Run\_001”.

A .ini file can be used to overwrite the default base and destination folders. A .ini file is specified by including the optional ini argument after -pm followed by the ini file name. For example:

```
TUFLOW_iSP_w64.exe -pmini package.ini PR_~s1~_~e1~_~e2~_001.tcf
```

Valid commands in the .ini file are:

Base Folder == <folder>

Copy Destination == <folder>

E.g.

Base Folder == C:\Projects\Modelling

Copy Destination == C:\tmp\pm\

A package model log file is created in the copy destination folder.

## 9.2.1 Package Model Bug Fixes 2018-03-AB

Build 2018-03-AB fixes a number of minor issues with the package model functionality these are:

- If a GIS file has no extension, it is assumed to be a .MIF as per the TUFLOW engine
- Operational Control files (.toc) are now correctly copied
- A more robust approach is used when variables are referenced within an input filename (e.g. 2d\_loc\_<<SIM\_ID>>\_v01\_L.shp). Previously these files could be missed when packaging the model.

## 9.2.2 Package Model Enhancements for Events and Scenarios

For Build 2018-03-AC the package model functionality has been updated to allow for automatic scenario variables (~s1~, ~s2~ ... ~s9~) and automatic event variables (~e1~, ~e2~ ... ~e9~) to appear in filenames and event sources. An example of an automatic variable in a filename is below:

```
Read Grid Zpts == grid\Lidar_<<~s1~>>.flt
```

If using the package model function on a model with automatic scenario and/or event variables, the valid options for these need to be defined in the package model .ini file using the commands:

```
Model Scenario ~s<scenario number>~ == <valid scenarios, separated by "|">
```

```
Model Event ~e<scenario number>~ == <valid events, separated by "|">
```

For the Read Grid Zpts == example above the scenario flag is used to define the year of collection of the base topography data set. The package model .ini file line below could be used to define the options.

```
Model Scenario ~s1~ == 2011 | 2015 | 2018
```

When running package model, this packages three input DEM files; Lidar\_2011.flt, Lidar\_2015.flt and Lidar\_2019.flt.

Automatic variables can also appear in a BC Event Source command, for example, in the below the scenario is being used to define an ensemble ID.

```
BC Event Source == __ensemble__ | <<~s1~>>
```

In this case the boundary file in the boundary database (bc\_dbase) contains "MC\\_\_ensemble\_\_.csv", which relies on the event source "\_\_ensemble\_\_". This is defined as an input switch e.g. "-s1 001" (ensemble number 1) or -s1 120 (ensemble 120). A valid list of all available ensemble IDs is not defined in the TUFLOW control files (it isn't needed), however, when packaging the model inputs this is required and the package model .ini would be used to specify this as follows:

```
Model Scenario ~s1~ == 001 | 002 | 003 | ... | 120
```

Event examples include:

```
Model Event ~e1~ == Q001 | Q010 | Q100
```

```
Model Event ~e2~ == 01hr | 02hr | 03hr | 06hr | 12hr | 24hr
```

### 9.2.3 Bug fix for automatic ESTRY Control File

Build 2018-03-AC now correctly processes the .tcf command “ESTRY Control File Auto” when packaging a model.

### 9.2.4 Package model Enhancements

As of Build 2018-03-AD when running package model functionality this copies files associated with:

- Rainfall Control Files
- AD Control files

Prior to 2018-03-AD these files were missed when running package model.

### 9.2.5 Package model bug fix

As of Build 2018-03-AE any GIS files referred to from within an event file (.tef) are now copied correctly.

## 9.3 More licence free models for hardware benchmarking

As part of the hardware benchmarking on our [TUFLOW Wiki](#), we have increased the number and size of the models that can be simulated without a licence for hardware benchmarking. The hardware benchmarks allow easy testing (i.e. no licence drivers need to be installed or configured), to understand the relative performance of TUFLOW across a range of hardware configurations and model sizes.

More details on the hardware benchmarking will be made available on the TUFLOW wiki after the release.

## 9.4 Enhancements and Bug Fixes

### 9.4.1 Bug fix for licensing that would lose access to licence for simulations with infrequent output

Build 2018-03-AA fixes a licencing issue with the HPC and GPU solvers, which could occur if the map output was very infrequent. This could cause TUFLOW to drop the licence and attempt to re-take the licence after a long period of 100% GPU calculations.

### 9.4.2 Fixed issues if using WIBU licences with multiple software providers

A WIBU licence server can contain licences from several different software providers. Build 2018-03-AC fixes an issue that occurs if a WIBU software or hardware lock contains more than 100 different licence types or product types. TUFLOW typically uses between 4 and 10 product codes regardless of the quantity of licences, e.g. a Network 1 TUFLOW licence uses the same number as a Network 100 licence. Therefore, this situation only arises if a WIBU Lock hosts licences from numerous software providers and software types totalling more than 100 separate entries.

If this error occurred TUFLOW would report “Licence Failed. Return Code T2/112” and fail to start a simulation.

### 9.4.3 Excess GPU Hardware licences now released

For Build 2018-03-AC, if the number of GPU devices specified for running the model exceeds the number of available GPU devices then the extra licences are released.

When running the HPC solver using multiple GPU devices (either with “GPU Device IDs ==” command or with the processing unit command line switch “-pu<device id>”) during model initialisation the number of GPU device licences used is based on the specified GPU devices.

When the HPC solver is initialised, the number of available CUDA compatible GPU devices is determined. If this is less than the user specified number of devices a WARNING 2784 message is generated, and a reduced number of GPU devices is used. Prior to 2018-03-AC the number of GPU licences used was not reduced, with the excess licences taken out for the duration of the simulation.

## 10 Backward Compatibility

### 10.1 With 2017-09 Release

TUFLOW Classic results should be unchanged from 2017-09-AC, and HPC results should be unchanged or have very slight changes. As this is a mid-year release for the TUFLOW 2017-2018 support year there is no “Defaults == Pre 2018” command provided for the 2018-03 release. **All users of the 2017-09 release are strongly recommended to upgrade to the 2018-03 release.**

Note that for HPC an enhancement to the extrapolation of Pit curves to provide consistency with Classic has been implemented, as described above in Item 2.1.4. This extrapolation can be turned off for backward compatibility using the .tcf command “Pit Default Extrapolate Q Curve == OFF”.

Also note the treatment of unconnected pits, which previously would trigger an ERROR, has changed to apply these pits as a virtual pipe pit that discharges water directly out of the model. To revert to issuing an ERROR use “Pit No 1D Connection == ERROR” – see Item 2.1.5.

The advanced index feature (detailed in Item 4.1) which can significantly improve 1D/2D model simulation speeds may cause very slight differences for single precision simulations. The majority of our benchmark testing showed no difference, however, this new feature can be turned off with the command “Index 1D2D Links == OFF”.

### 10.2 Build 2018-03-AC versus Builds 2018-03-AA and AB

Build 2018-03-AC may cause a change in results from prior builds due to several bug fixes. Testing of a large range of models indicates that for those models affected by these bug fixes that the changes are typically very minor. Very slight changes in results to HPC results may also occur due to the enhancement described in Item 8.6.1.

Changes that may affect results are described in the following Items:

- 3.1.2 Bug fix for Gully Z Shapes (Classic and HPC)
- 3.1.3 Bug fixes/enhancements for FC Shapes and Layered FC Shapes (Classic and HPC)
- 4.9.1 Bug fix for rainfall initial loss in TUFLOW Classic
- 4.9.2 Bug fix for soil infiltration in TUFLOW Classic
- 4.9.3 Bug fix for SX Region with “Z” flag (Classic and HPC)
- 8.6.1 Precision enhancements (HPC)

## 11 TUFLOW-AD (Advection Dispersion) Module

The 2018-03-AD version of TUFLOW introduces several improvements and fixes for the TUFLOW-AD (Advection Dispersion) module. These are outlined below, the documentation for the AD module is currently being updated.

The 2019 release of TUFLOW will enable support for AD models using the HPC solver.

### **11.1 Fix issues with large models causing memory issue**

Build 2018-03-AD fixes a bug that could cause very large 2D models with TUFLOW Advection-Dispersion module turned on to crash during initialisation. This would cause a “Stack Overflow” issue and the simulation window to disappear.

### **11.2 SX lines now supported**

Build 2018-03-AD adds support for SX / CN lines as well as SX points for 1D/2D connections. Prior to this, SX line objects were ignored and reported in the AD log file (.adlf).

### **11.3 Wetting Enhancements**

As outlined in Section 6.3.1 of the TUFLOW manual TUFLOW Classic uses an Alternating Direction Implicit scheme for performing the hydraulic calculations. As an implicit scheme this can run with Courant numbers of greater than 1, meaning that water can travel across more than 1 cell within a timestep, this wetting can introduce (typically minor) mass balance issue for AD models. Build 2018-03-AD introduces an improvement in mass conservation when wetting. See also point 11.4

### **11.4 AD Calculation frequency**

As outlined in Section 6.3.1 of the TUFLOW manual, the TUFLOW Classic computational procedure for solving the hydraulics involves two stages per timestep. Prior to 2018-03-AD the AD calculations were performed at the end of stage 2, i.e. at the end of each hydraulic timestep. This method remains the default for the 2018-03-AD version of TUFLOW, however, from 2018-03-AD onwards the user can specify the timing on the AD calculations. This can be set to the end of stage 1, stage 2 or after both stages 1 and stages 2. Performing the AD calculations at both stages 1 and stages 2 can improve mass balance on models with significant wetting / dry, but will also increase the simulation time.

The new .adcf command “AD Calculation Frequency == Stage 1 | {Stage 2} | Both” can be used to control the timing on the AD calculations in relation to the hydraulic calculations.

### **11.5 Additional Error Checking**

Build 2018-03-AD introduces more rigorous error checking when unsupported features are encountered in model configurations. These include:

#### **11.5.1 If Event / If Scenario**

TUFLOW-AD does not currently support event / scenario logic in the AD control file or boundary control file (.tbc). If this is encountered the TUFLOW model will be halted with an ERROR message.

#### **11.5.2 Overlapping SA boundaries**

When applying AD concentrations, at source boundary cells, overlapping SA boundaries cause issues with tracer mass. For build 2018-03-AD update, a check for overlapping SA cells is performed



## 11.6 Units for Write Mass can now be specified

As outlined in the TUFLOW-AD documentation, the default output unit for constituent mass if using the setting “Write Mass == ON” is in tonnes (if the concentrations are specified in mg/L). This can now be set with the .adcf command:

```
Write Mass Unit == Gram | Kg | {Tonne}
```

The default unit for 2018-03-AD remains in tonnes.

## 11.7 Other Changes

### 11.7.1 GIS / output enhancements

A range of improvements have been made to the GIS outputs from TUFLOW-AD these include:

- AD checks / messages files now output in shapefile format if “GIS Format == SHP” specified in the TUFLOW Control File (.tcf)
- QGIS workspace now closed cleanly at the end of the simulation
- Messages and GIS workspace are now written to the “Log Folder” specified in .tcf. Previously these outputs were output in the same location as the .tcf.

### 11.7.2 Removal of longitudinal dispersion coefficient limit

Prior to Build 2018-03-AD if a longitudinal dispersion coefficient of greater than 500 was specified this would cause TUFLOW-AD to close with an ERROR. From 2018-03-AD if a value of greater than 500 is specified a WARNING 4005 is generated, but the TUFLOW simulation will continue.

**NOTE: As with all model parameters it is strongly recommended that these are calibrated where possible and parameters kept within literature values.**