TUFLOW Classic and GPU Solver 2016-03 Release Notes

Document Updates and Important Notices

January 30, 2017: Updated for the 2016-03-AE Build. Changes are highlighted in light orange starting at Item **153,** and are primarily minor enhancements and bug fixes. If using start times of less than zero or operational pumps using a pump curve, users should upgrade to this build.

October 26, 2016: Updated for the 2016-03-AD Build. Primary changes are highlighted in light grey. Also note the manual correction in Item **62(b).**

NOTE: For the GPU Solver, due to Items 143 below, which include important enhancements at water level boundaries and a bug fix, it is recommended that Build 2016-03-AD or later be used for GPU Solver simulations, unless for legacy reasons use of earlier versions of the GPU Solver is justified.

NOTE: For the TUFLOW Classic Solver, due to Item 147, any users utilising the CWF (cell width factor) feature via the "`Read GIS CWF ==`" **or** "`Read Grid CWF ==`" **commands should upgrade to 2016-03-AD unless for legacy reasons earlier versions need to be used.**

September 15, 2016: Updated for the 2016-03-AC Build. Primary changes are highlighted in light blue. Also note the manual correction in Item **62(b)**.

NOTE: Due to Item 133, any users utilising the new structure group output should upgrade to 2016-03-AC.

NOTE: Due to Item 139, any users utilising the SRF feature with TUFLOW GPU should upgrade to 2016-03-AC.

August 15, 2016: Updated for the 2016-03-AB Build. Primary changes are highlighted in light green.

NOTE: Due to the Items 103 and 104, any users running TUFLOW GPU should upgrade to 2016-03-AB.

Licensing

To run simulations using Builds 2016-03-AA to AD requires payment of the 2015/2016 annual software maintenance fee and for the TUFLOW licence to have been updated (ie. via RaC/RaU files). For the 2016-03-AE version, payment and an updated licence for the 2016/2017 year will be required. For tutorial and demo models, or if running in free demo mode (see dot point above), no licence is required. For any enquiries please contact sales@tuflow.com.

Table of Contents

2016-03 Release Overview

The 2016-03 release includes some major new features and range of enhancements. The most significant of these are:

- Numerous GPU enhancements:

  o Virtual Pipes. This feature provides a functional methodology to represent pipe drainage networks and cross drainage structures within a GPU model.

  o Improved Source Area (SA) boundary options: All of TUFLOW Classic's SA functionality is now supported by the GPU solver.

  o Maximum Outputs are now available. The GPU Solver now tracks the maximum value and time of maximum for water level, velocity and Z0 (VxD).

  o Improved RAM allocation. This means even larger GPU models are possible!

- New and improved structure options:

  o 1D Bridge options which automatically account for the losses due to flow contraction, expansion, and pressure flow.

  o New time varying weir options. These are proving useful to simulate structures such as fabri (inflatable) dams.

  o New grouped output options. Multiple structures (1D and 2D) can be associated with one another so model results can be output that are representative of the structure group.

- New Rainfall Control File framework. This addition creates a rainfall grid internally within TUFLOW based on point pluviograph rainfall data.

- Additional Map Output Formats: Now available for WaterRide, 12D, Delft-FEWS in NetCDF and also new GIS options for point datasets.

- A new free demo version of TUFLOW. The free version is fully enabled, though has the following limits:

  o 100,000 total cells and 30,000 active (potentially flooded) cells

  o 100 1D channels

  o There can only be one 2D domain (multiple 2D domains are not supported)

  o A simulation time of 10 minutes.

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.

1D Structures

1 **New BB Bridges**

BB bridges are a new bridge type. They differ from B bridges in that the losses due to flow contraction and expansion, and the occurrence of pressure flow is handled automatically. Only energy loss coefficients accounting for piers and the bridge deck once submerged and not under pressure flow are required.

A future release is planned to include a BB bridge feature that automatically calculates the variation in pier and deck loss coefficients with height. This will remove the need for manually defined BG table inputs. For more details, please refer the 2016 TUFLOW Manual.

2 **2013-12 Release Weirs**

2013-12 release weirs occasionally caused instabilities when they became drowned out. The weir approach has been enhanced, significantly improving model stability.

3 **Operational Weirs**

Weirs may now be operated to simulate structures such as fabri (inflatable) dams.

(a) This option is available for WB, WC, WD, WO, WR and WT weir types.

(b) 1d_nwk attributes are used to set the limiting dimensions of the weir, as follows:

(i) The "*Width_or_Dia*" attribute defines the width of the weir when fully open. This attribute is also used to set the fixed width of the weir if it remains unchanged for the entire simulation.

(ii) The "Height_or_WF" attribute sets the height of the weir when fully up. The invert of the weir (when fully down) is set by the maximum of the "*US_Invert*" and "*DS_Invert*" attributes. This attribute cannot be used to set the Weir Calibration Factor for operational weirs (a value of 1.0 will be used).

(c) New TUFLOW Operational Control (.toc) commands are:

Weir Height Speed [{} | min] == <speed>

Weir Width Speed [{} | min] == <speed>

Where <speed> is by default m/h or ft/h. The min option will treat the speed as m/min or ft/min.

Weir Height [{} | %] == [<height> | CLOSE | OPEN]

Where <height> is the height (not elevation) of the weir above its fully down (open) state. The % option allows for specification of the weir height that is up (0% indicates completely lowered and 100% completely raised).

Weir Width [{} | %] == [<width> | CLOSE | OPEN]

Where <width> is the width of the weir. The % option allows the specification of a percentage of width that is open.

Generic commands "Operation == NO CHANGE" and "Period Opening/Closing ==" also apply.

4 **Operational Structure Defaults**

Operational structure defaults were not set for some parameters possibly causing unexpected results in 2013-12-AD. Defaults have now been set. Refer to the Operational Structure section of the 2016 manual for more information.

5 **New Grouped Structure Reporting Feature**

This new feature allows output time-series and summary data for single and/or combined structures.

(a) 1D structures:

(i) Parallel 1D structures (i.e. they have the same upstream and downstream nodes) are automatically grouped together and treated as a single structure for this output. The group ID matches the structure with the lowest bed elevation.

The digitised direction of the channels is important for this feature. All channels must be digitised in the same direction to form a group.

(ii) A 1D structure without neighbouring parallel structures are also included in this output.

(iii) The below and above deck flow components are based on the logic that all structures except weirs are assumed to contribute to below deck and weirs to above deck.

(b) 1D/2D structures:

This feature can be used to group 1D/2D structure output.

(i) A 2d_po "*Type*" attribute of "QS" specifies that the grouped 1D/2D structure flow output be used.

- All 2D flow across this line (including multiple 2D domains) and any 1D structures that intersect the line are grouped together. The 1D structure's 1d_nwk line does not have to snap with the QS line; they only have to cross over each other. The ID assigned to the structure output group is the 2d_po "*ID*".

- If the QS line selects cell sides that are modified by the Layered FC feature, the output will split the flow into a below and above deck component. Layers 1 to 3 are considered below deck and Layer 4 above.

(ii) A 2d_po "*Type*" attribute of "HU" and "HD" is used to extract upstream and downstream water levels.

- "HU" objects should be located upstream of the structure and "HD" downstream. These can be point or line objects. If a line is used, the average 2D water level along the object will be used to populate the water level data in the output. Lines can have more than two vertices (ie. polylines are accepted).

- To associate the "HU" and "HD" objects with a "QS" line, all three (QS, HU and HD) must have the same 2d_po "*ID*". If a QS line has no "HU" object associated with it, output that cannot be produced will be given a value of -99999 in the _SHmx.csv output file described below.

(c) Grouped structure output files

The new grouped structure output is located in the plot/csv folder. It includes:

(i)   _SQ.csv file: Containing time-series data of the flow through the structure/s. This file is similar to other time-series .csv output, though it only contains structures (single structures or structure groups) as described above.

(ii)   _SHmx.csv: Containing a summary of each structure when the upstream water level reached its maximum. To generate this output the flow and other information is tracked every timestep for grouped structures. The output columns include: flow, area and average velocity for below and above deck; total flow, area and average velocity for the whole structure; upstream and downstream water levels; the head drop across the structure (ie. upstream minus downstream water level); and the time this data was recorded (ie. the time the upstream water level peaked).

(iii)   All 1D and grouped 1D structures are automatically output. As discussed above, output for 2D structures requires QS, HU and HD 2d_po objects.

6   **Primary Upstream and Downstream Channel Selection**

Selection of primary upstream and downstream channels now correctly takes into account bed slope. This may change results where the selection of channels, for example in determining the approach and departure velocities at a structure, has an effect. The method used prior to the 2016-03 release is applied if "`Defaults == PRE 2016`" is used.

7   **WW Weir Flow**

There has been a slight improvement to WW weir flow that is triggered with "Weir Flow == Method C". This update should not cause any significant change in results.

2D Structures and Form Losses

8   **Layered FC Enhancement:**

Two options are now available to specify the method in which form losses are applied. The original approach was to accumulate the losses with depth. The new approach proportions the losses with depth.

(a)   To specify the method on a structure by structure basis, populate the 2d_lfcsh Shape_Options attribute with either PORTION to proportion losses to the depth of water (this is the 2016-03 default), or CUMULATE to accumulate the losses as the depth of water increases.

(b)   The new .tcf command "`Layered FLC Default Approach == [ CUMULATE | {PORTION} ]`" can be used to set the default method to be applied to all structures in the model. The default approach prior to the TUFLOW 2016-03 release was CUMULATE, while for the 2016-03 release it is PORTION. If "`Defaults == PRE 2016`" is used, the default is set to CUMULATE.

(c)   The different approaches will produce different results, therefore either "`Defaults == PRE 2016`" or "`Layered FLC Default Approach == CUMULATE`" may be required for legacy models.

9   **FLC / Unit Length**

FLC values can now be specified as FLC (form or energy loss coefficient) per unit length (ie. per metre or foot) for .tgc FLC commands. Specify FLC/L to use this new option. For example:

`Set FLC/L ==`

`Read GIS FLC/L ==`

`Read GRID FLC/L`

The advantage to this approach is that it makes these inputs independent of the 2D cell size when using regions (polygons). As such, if the 2D cell size is changed the same energy loss will be applied to both models over the area of the region.

This per unit length this approach conforms with the existing FLC input for region objects in FC (flow constriction) and Layered FC GIS layers.

Initial Water Levels

10 **Auto Initial Water Level Command**

This is a new option to automatically set the model's initial water level to the downstream boundary water level.

<code>Set IWL == AUTO</code>

(a) Sets the IWL in 1D and 2D domains to the value of the water level boundary in the model at the start of the simulation.

(b) If the model has more than one water level boundary, and the starting level is different, an ERROR 0037 occurs.

(c) This feature only works for HT and HS boundaries.

(d) Use "Set IWL == AUTO" in the .tcf (the initial water level will be applied to both 1D and 2D domains).

(e) The initial water level is only applied to 1D nodes and 2D cells that have been allocated a zero IWL (this is the default value). This means that "`Read GIS IWL ==`" can still be used to set the IWL in other parts of the model, such as a lake. Note that if "`Read GIS IWL ==`" sets a zero IWL, then this will be overridden by the AUTO value.

This new command has been very useful during a Monte Carlo assessment that required over 11,000 simulations with varying initial water levels.

Boundaries and Links

11 **SX Flow Distribution Cutoff Depth**

Testing has found flow exchange may occur on "dry" cells in rare cases at SX 1D/2D locations. This occurred as a result of a numerical precision issue relating to the optimised compiling of the source code. This has been corrected by introducing a shallow depth cutoff for SX cells. The default value has been set to 5mm. This value can be changed using "`SX Flow Distribution Cutoff Depth ==`". This may change results very slightly, therefore for legacy models use "`Defaults == PRE 2016`" or "`SX Flow Distribution Cutoff Depth == 0.0`" for backward compatibility.

12 Default Boundary Type

It is now possible to specify a default boundary type in the .tbc. This is a command that can be repeated as per the below.

```
BLANK BC TYPE == SX

Read GIS BC == mi\2d_bc_M02_culverts_TD15006.MIF

BLANK BC TYPE == NONE  !will revert back to an error
```

13 Default HQ Slope

A default HQ slope can now be specified directly in the .tbc. This command is repeatable. This will only be used if no boundary name is specified for the HQ (as a slope is given preference over the name).

```
BLANK HQ SLOPE == 0.01
```

This can also be set as a variable:

```
Set Variable HQSlope == 0.02 !This would be in the .tcf or read file.

BLANK HQ SLOPE == <<HQSlope>>
```

14 Rainfall Control File

New gridded rainfall options are now available. The new format uses a rainfall control file (.trfcf) to manage the specification of gridded rainfall over the 2D cells based on point rainfalls locations. Three methods are available for interpolation:

- IDW (Inverse Distance Weighting)
- TIN (triangulation)
- Polygons

When the rainfall control file is processed during model initialisation a series of rainfall grids are output, as the simulation progresses the individual rainfall grids are read. This workflow has been chosen to reduce memory usage while TUFLOW is running.

Example .trfcf file are available on the wiki in the .trfcf examples.

Available optional rainfall control file commands are:

```
Maximum RF Locations == <maximum number of rainfall point locations> {1,000}

(Optional)
```

This controls the temporary memory allocated for reading / storing the rainfall data. If more the 1,000 point rainfall locations are used, this can be increased. Similarly, the default value can also be reduced to decrease temporary memory allocation.

```
Maximum Hyetograph Points == <maximum number of points in a hyetograph>
{1,000}
```

This controls the temporary memory allocated for reading / storing the rainfall data. If more the 1,000 points occur in the rainfall hyetograph, this can be increased. This can be reduced to decrease temporary memory allocation.

```
RF Grid Origin == <x coordinate, y coordinate> | {}
```

(Optional)

This optional command sets the origin for the output rainfall grid. If this command is omitted the rainfall grid origin is based on the dimensions in the TUFLOW model (the .tgc file).

`RF Grid Size (X,Y) == <x dimension, y dimension> | {}`

`RF Grid Size (N,M) == <number of rows, number of columns> | {}`

(Optional)

This optional command sets the size of the output rainfall grid, similar to the Grid Size (X,Y) or Grid Size (N,M) in the geometry control file. If this command is omitted the rainfall grid size is based on the dimensions in the TUFLOW model (the .tgc file).

`RF Grid Cell Size == <output cell size> | {10 times model cell size}`

(Optional)

This optional command sets the cell size for the output rainfall grid. If not set, a value of 10 times the model cell size is used. Spatial rainfall may vary on a larger scale than the hydraulic cell size and using a high resolution rainfall grid is typically not required.

`RF Grid Format == ASC | FLT | NC {}`

(Mandatory)

This mandatory command sets the output grid format. Options are:

- ASC (ESRI asc grid - extension .asc)

- FLT (binary float grid - extension .flt)

- NC (NetCDF - extension .nc).

The rainfall grids are output to a separate folder \RFG\ (rainfall grid) in the location off the .trfcf. If the .trfcf is in the TUFLOW\bc_dbase\ folder, a new directory TUFLOW\bc_dbase\RFG\ will be created containing the output grids.

ASC and FLT Format

If set to ASC or FLT a series of grids are written (one for each hyetograph timestep) in the ASC or FLT formats used for other TUFLOW outputs. Due to the large number of grids that may be written, these are separated into sub-folder under the RFG\ folder, e.g. TUFLOW\bc_dbase\RFG\<simulation_name>\<simulation_name>_t<time>.

An index file which contains a list of the times and rainfall grids names is written in .csv file format in the directory. E.g. TUFLOW\bc_dbase\RFG\<simulation_name>\<simulation_name>_rf_index.csv

A limit of 1,000 grids exists if using the ASC or FLT format rainfall grids, if more than 1,000 grids are required please use the NetCDF format instead.

NetCDF Format

If NC is specified a single netcdf file (.nc) containing all timesteps is output. This is given the simulation name using a *.nc file extension.

A total rainfall depth is also output, however, this is not used by TUFLOW during the simulation, though can be used for checking purposes. Refer to the TUFLOW wiki for more information on the [TUFLOW NetCDF Rainfall Format](#).

There is no limit to the number of rainfall timesteps that are included in the NetCDF format.

Rainfall Interpolation Options

Rainfall interpolation is performed each time a simulation is started with a .trfcf input, if preferred this can be simulated once for each event and the output file (.asc, .flt or .nc) can be read into future TUFLOW simulations. The output formats from the rainfall interpolation are the same as the input format used by the "`Read Grid RF ==` " command in the .tcf.

The method of rainfall interpolation must be set using the following command:

`RF Interpolation Method == TIN | IDW | Poly | {}`

`(Mandatory)`

This mandatory command sets the interpolation method. The options are described further below:

POLY Method

A series of GIS polygons are specified and the rainfall for each polygon comes from the point rainfall. This can be used to apply a distribution such as Thiessen polygons. A series of GIS polygons are read in the 2d_rf format, these polygons can either have the rainfall boundary Name (and F1 and F2 factors) specified on the polygon objects, or if these attributes are blank TUFLOW will look for rainfall points (specified with the "`Read GIS RF Point == <gis layer>`") that fall within the polygons. If the Name attribute in the polygon layer is blank and no points fall within the polygon an [ERROR 2619](#) will be returned.

This method is similar to using a series of rainfall polygons read in via "`Read GIS RF == <gis layer>`" in the .tbc. By pre-processing this using a rainfall control file is much more memory efficient, particular if a large number of rainfall boundaries are used.

IDW Method

An inverse distance weighting is used to calculate the rainfall depth based on the distance to the surrounding rainfall points (specified with `Read GIS RF Point == <gis layer>`").

$$\hat{v}_1 = \frac{\sum\limits_{i=1}^{n} \frac{1}{d^p_i} v_i}{\sum\limits_{i=1}^{n} \frac{1}{d^p_i}}$$

The exponent (p) within the above equation can be specified with the "`IDW Exponent == <value>`" command. The default value is 2. See also the commands below for more information.

`IDW Exponent ==`

`IDW Max Distance ==`

`IDW Max Point ==`

TIN Method

A TIN (Triangulated Irregular Network) is specified which connects the rainfall point locations.

`Read GIS RF Point == <gis_layer>`

`(Mandatory)`

Read the point rainfall locations in the 2d_rf file format. For each point the attributes are Name, F1 and F2 factors. If the rainfall factors F1 and/or F2 are zero (or less than zero), these are changed to 1 and [WARNING 2618](#) is issued.

`Read GIS RF Polygon == <gis_layer>`

`(Mandatory if using "RF Interpolation Method == Poly")`

A series of GIS polygons are read in the 2d_rf format, these polygons can either have the rainfall boundary Name (and F1 and F2 factors) specified on the polygon objects, or if these attributes are blank TUFLOW will look for rainfall points (specified with the "`Read GIS RF Point == <gis_layer>`" that fall within the polygons. If the Name attribute in the polygon layer is blank and no points fall within the polygon an [ERROR 2619](#) will be returned.

`IDW Exponent == <IDW exponent value> | {2}`

`(Optional)`

If using the "`RF Interpolation Method == IDW`" the exponent in the IDW equation can optionally be set. The default value if 2.0

`IDW Max Distance == <maximum distance>`

`(Optional)`

If using the "`RF Interpolation Method == IDW`" the maximum distance for a point to be considered in the interpolation can be set using this command. If not specified, no maximum distance is considered.

`IDW Max Point`

`(Optional)`

If using the "`RF Interpolation Method == IDW`" the maximum number of points considered in the interpolation can be specified. This may reduce memory usage if a very large number of rainfall points are used.

`Read GIS RF Triangle == <gis_layer>`

`(Mandatory if using "RF Interpolation Method == TIN")`

Reads in a GIS file in the .mif or .shp file format which is used for defining the triangulation of the rainfall points. The GIS objects should be polygons with three vertices and each vertex should be snapped to a rainfall point location. The attributes of the GIS layer are not used. For each grid cell in the rainfall output grid the rainfall depth is based on the planar (linear) interpolation of the three rainfall depths at the vertices of the triangle. An example of a triangulation (red) connecting the rainfall point locations (blue) is shown below.
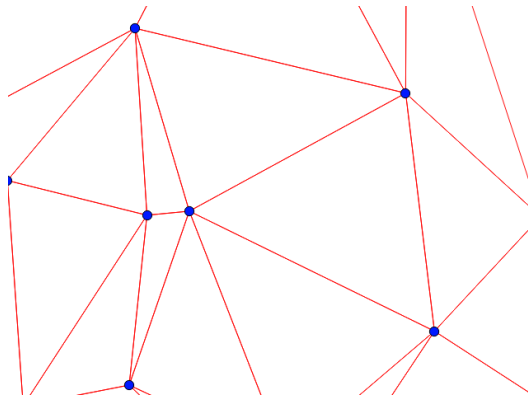
Bed Resistance and Materials

15 Log Law Bed Resistance

Log Law or "The Law of the Wall" option for varying Manning's n values with depth at very shallow flows is now available in both Classic and GPU solvers. A roughness height is used at shallow depths to derive an equivalent Manning's n value. As depth increases the user can set a limiting, conventional, n value. For further information see the 2016 TUFLOW manual in the Bed Resistance section.

16 Manning's n Scale Factor

A manning n scale factor can now be defined using the tcf command:

`Read Materials File == <file> | [ {1.0} | <n_factor> ]`

The `<n_factor>` value will factor all Manning's n values. For example, to increase all Manning's n values by 10% in a materials file (.tmf or .csv format), enter:

`Read Materials File == My_Materials.tmf" | 1.1`

2D Topography

17 New 2d_vzsh Restore Option

Variable Z Shapes can be restored once or repeatedly. Two new attributes have been added to the2d_vzsh layer for this feature:

- "*Optional Restore_Interval*", the time in hours between when the variable Z shape has finished altering the geometry and when to start restoring the Zpts back to their original values.

- "*Restore_Period* ", time in hours over which the variation in Zpt elevations occurs to restore the Zpts back to their original values.

Refer to the 2016 manual for more information.

18 Read GIS Z Shape Enhancement

The Read GIS Z Shape format has been enhanced to allow the "GULLY" and "ADD" flags to be specified in combination. If the "ADD" option is used with "GULLY" or "MIN" option and the add value is negative then the gully breakline formulation is used selecting cells centres and cell sides to form a continual flowpath).

19 **Record Gauge Data Output**

A new .tgc command has been created for associating gauge water level information with neighbouring receptors (buildings, infrastructures etc.)

Read GIS Objects [RECORD GAUGE DATA [{} | USE ZC | ZPTS]] == <gis_layer>

("Read GIS Receptors" may also be used). In the 2013-12 release this undocumented command was named "Read GIS Gauge Output =="; it has been enhanced for the 2016-03 release. It references GIS layer(s) containing points or polygons representing receptors such as properties or buildings. RECORD GAUGE DATA records the flood level and simulation time at one or more gauge(s) when the receptor is first inundated above a trigger inundation level (e.g. floor level). Gauges are defined as a point within a 2d_po GIS layer with type "G_". The levels from all gauges are recorded at each receptor once inundated. For more information refer to the TUFLOW 2016 manual.

At present the only available option is the RECORD GAUGE DATA feature. More options to record or value add information to receptors is planned for future releases.

Simulation Control

20 **Syntax Error Processing**

The 2016-03 release applies stricter rules when processing command line syntax within the control files (.tcf, .tgc, .ecf etc.). The new rules issue an ERROR if a "==" is not present in the command line syntax. For example: "If Scenario = Exg" will produce an error. The correct syntax is "If Scenario == Exg". A new command is provided to switch this new rule off.

Command Line Processing == [{2016} | Pre 2016]

Defaults == Pre 2016 will also turn this new syntax processing rule off.

21 **End After Maximum**

The "End After Maximum ==" command now supports an optional second argument to set a height tolerance for detecting whether a 1D node or 2D cell has reached its maximum.

End After Maximum == [<end after maximum> | <tolerance>]

For example, "End After Maximum == 0.25 | 0.01" will terminate the simulation once either the "End Time ==" has been reached, or there are no 1D nodes or 2D cells that have increased by 1cm (0.01m) in the last 15mins (0.25h). The % of 1D nodes and 2D cells that have reached their maximum are displayed after the "Mx" in the simulation console window and in the .tlf file. The default tolerance is 0.001m (or 0.001ft), unless "Defaults == Pre 2016" used, in which case it is 0.0.

22 **Batch Simulation**

The -b batch simulation run time option no longer stops with a message request if a simulation terminates prematurely. This means the batch process will proceed through all simulations regardless of whether there is an input ERROR.

Always cross-check whether the simulations have all completed by viewing the "_ TUFLOW Simulations.log" file in the runs folder, or the .tlf or .tsf files.

23 Output Drive

The Output Drive for a simulation can be specified with the –od command line option. For example –odC will redirect all outputs to the C:\ drive. This command line argument is given higher priority than the tcf command "`Output Drive ==` ".

24 Demo Mode

"Demo Model == ON" has been extended to include a free version of TUFLOW. All features are enabled except for restart files when running in demo mode! The limits are:

- 100,000 total number of cells
- 30,000 active (potentially flooded) cells
- 100 1D channels
- There is only one (1) 2D domain
- A simulation time of 10 minutes.

TUFLOW support is available to those users running in demo mode, however, support should not be used as a substitute for training. To contact TUFLOW Support use support@tuflow.com and for training options, please email training@tuflow.com.

25 Forward Slash Path Separator

This new command sets the operating system path separator to a forward slash as opposed to a backslash.

`Use Forward Slash == [ ON | {OFF} ]`

`(Optional)`

When set to "ON", forward slash (/) is used in the text output files contain filepaths (e.g .tlf, .qgis, .tpc, .wor). For example, the link to the GIS output file shown below will change from the default backslash version of:

`GIS Plot Layer Points == ..\gis\M04_5m_001_OSSep_BW_PLOT_P.mif`

To the line below for the forward slash version:

`GIS Plot Layer Points == ../gis/M04_5m_001_OSSep_FW_PLOT_P.mif`

This forward slash operator is used in Linux based operating systems and is required for TUFLOW users that are viewing results on Linux systems. The forward slash will also work in a Windows environment. This update is part of a wider task required to make a Linux compatible version of TUFLOW Classic / TUFLOW GPU. TUFLOW-FV is currently available as a Linux RPM download.

Simulation Map Outputs

26 New Map Output Formats

Several new map output formats are now available in addition to the previous documented formats that continue to be supported. The new formats include:

`Map Output Formats == WRC | WRR | NC | TGO | GIS`

The following map output formats are still supported as documented in the TUFLOW manual:

`Map Output Formats ==` DAT | XMDF | WRB | GRID | T3 | TMO | SMS TRIANGLES | HIGH RES | FLT | ASC

The new output formats are described below:

(a) **waterRIDE** by WorleyParsons is commercial software for visualising and post-processing hydraulic modelling results. For more information [click here](). Supported waterRIDE formats include: WRB, WRR and WRC. A combination of these can can be used.

For each waterRIDE output, one file is produced that contains the model's ground/bathymetric elevations, water levels, velocities (scalar and vector), and optionally the VxD product and one hazard category. If Z0 and/or a hazard category are not specified for WRB output, waterRIDE can optionally post-process these values. Also, if waterRIDE outputs specified, other data types specified using "`Map Output Data Types ==`" are ignored. Other data types such as depth are also post-processed by waterRIDE.

If maximums are tracked these are also added to the waterRide formats and are restricted to these data types. Note that if waterRIDE is used to post-process maximums the values will be different to those provided by TUFLOW. TUFLOW tracks maximums every timestep, whilst waterRIDE will calculate maximums using the values in the .wrb file which only occur every map output interval.

Each of the waterRide output is described below:

(i) **WRB Format.** This is unchanged from previous releases of TUFLOW. The .WRB is a triangulated output. 2D cells are represented as four triangles with a common vertex at the cell centre.

(ii) **WRR Format.** This outputs a single file for any multiple domains and 1D water level lines in raster format. The output grid is interpolated as a north-south aligned raster with a single cell size. The default cell size equals half the TUFLOW cell size. The output grid interpolation method used by the WRR format matches the approach used by the ASC, FLT, NC and TGO formats.

(iii) **WRC Format.** The WRC is a composite file and provides links to separate .wrb and .wrr files. These files adopt a different format to the above mentioned WRB and WRR formats. Any 2D domains are output as a rotated WRR format grid (ie. not north south aligned) with the cell size equal to the TUFLOW cell size. The output value is the cell centre value. For a multiple domain model, a separate .wrr file is created for each domain; these can each have a different rotation and cell size. Any 1D water levels are output in a separate .wrb file. The WRC "master" file is output in the specified results folder and the .wrr and .wrb files are separated into a waterRide sub folder.

(b) **NETCDF** output raster. The [NETCDF]() (Network Common Data Format) is a commonly used format for storing modelling and scientific data. TUFLOW supports the output of raster data into the netcdf file format. A single NetCDF file is created containing both the time varying and static output (e.g. maximums, and time outputs). The output is a north-south aligned raster and includes outputs from multiple domains and water level lines. The NetCDF interpolation method matches the approach used by the ASC, FLT, WRR and TGO formats.

A number of NetCDF specific commands are supported. These are described below.

`NETCDF Output Compression == [ OFF | {ON} | <compression_level_0-9> ]`

This command sets the compression for the output NetCDF file. A compression level of 1 is used if this command is set to "ON". The compression level can also be set to any number from 1 to 9. Higher levels of compression (greater than 1) result in smaller files but are slower to write / access. For example: `NETCDF Output Compression == 9`. If this command is set to "OFF" (default) a NETCDF "classic" format is used without compression. If "ON" a NetCDF4 file is used with compression. The compressed version can be read by ArcMap and Matlab, though not QGIS.

`NETCDF OUTPUT Start Date == [ { 2000-01-01 00:00} | OFF | <date in isodate format> ]`

This command sets the start date for the NetCDF output. If set to "OFF" or "NONE" the units attribute of the NetCDF file is simply the time unit (e.g. units = 'hours'). If a date is provided the time units will be in the format <unit> since <date>. For example, 'hours since 2000-01-01 00:00' or days since 2000-01-01 00:00'. The default is '2000-01-01 00:00', as not having a date appears to cause issues in ArcMap. If a date is specified, it is recommended that this be in isodate format. TUFLOW **does not** check the date is valid, it is simply added to the NetCDF time variable.

See also `NETCDF OUTPUT TIME UNIT` command below.

`NETCDF OUTPUT TIME UNIT = [ DAY | {HOUR} | MINUTE ]`

This command sets the output time unit for NetCDF outputs. The default is hours (as per other TUFLOW outputs). See also `NETCDF OUPUT Start Date` command above.

`NETCDF OUTPUT DIRECTION == [ {ANGLE} | BEARING ]`

When output in NetCDF raster format any vector outputs (e.g. flow or velocity) are output as two datasets: magnitude and direction. This command specifies whether the "direction_of" attribute is in arithmetic (0 degrees east) or geographic (0 degrees north) co-ordinates. The default output direction is arithmetic.

`NETCDF Output Format == [ Generic | {FEWS} ]`

Sets the output format for the NetCDF outputs. The FEWS format option outputs the maximum, time of peak, duration outputs with a timestamp at the beginning of the simulation to ensure the file can be loaded into FEWS correctly. The Generic output format does not assign a timestamp with the static datasets. For more information on the format please see the wiki page TUFLOW NetCDF Raster Format.

(c) **12D TGO** format is utilised by 12D Solutions for their TUFLOW interface. The output is a north-south aligned raster and includes outputs from multiple domains and water level lines. The output grid interpolation method used by the 12D TGO format matches the approach used by the ASC, FLT, NC and TGO formats.

(d) **GIS** format is a gridded output in either MapInfo (.mif) or Shapefile (.shp) format as either points or polygons. The GIS Format command is used to specify whether the outputs will use mif or shp format. This offers a similar functionality to TUFLOW_to_GIS. A separate GIS file is created for each output time. Scalar results are output as point features. Vector results can either be output as a point or region GIS file. Specific commands to this output type are:

(i) `GIS GRID VECTOR TYPE == [ {Region} | Point ]`

It specifies whether the output file should contain point or region objects. The default is regions (as per TUFLOW_to_GIS).

(ii) `GIS GRID VECTOR DIRECTION == [ {ANGLE} | BEARING | VERBOSE ]`

The magnitude and direction are output as attributes on the GIS object. This command sets the direction convention. "ANGLE" sets the direction output using in arithmetic format (0 degrees = East). "BEARING" sets the direction to a compass bearing notation with 0 degrees = North). If set to "VERBOSE", the x-direction component, y-direction component, angle and bearing are all output as attributes on the GIS layer.

(iii) `GIS GRID VECTOR SF == [ {1} | <scale factor> ]`

Factor for scaling of region objects (default =1). A value of 1 means a velocity of 1m/s is one cell long, with a scale factor of 2, a vector of magnitude 1, would be 2 cells long. A negative value outputs vectors of fixed length `<scale factor>` metres or feet.

(iv) `GIS GRID VECTOR TTF == [ {0} | <tail thickness factor> ]`

Factor for scaling the thickness of arrow tails (default =0). Thickness is TT_Factor times the arrow size.

The commands above can be applied to all vector outputs or can be specific to the output parameter by prefixing the command with V, Q or W, for velocity, unit flow or wind respectively. For example the below, will set the scale factor to 1 for all outputs except the unit flow Q which has a smaller factor of 0.1.

`GIS GRID VECTOR SF == 1`

`Q GIS GRID VECTOR SF == 0.1`

27 XMDF Compression

XMDF compression is now supported.

`XMDF Output Compression == [ OFF | {ON} | <compression level 0 – 9> ]`

Sets the compression for the output xmdf file. If set to "ON" a compression level of 1 is used. The compression level can also be set via a number from 1 to 9. Higher levels results in smaller files but are slower to write/access. For example, `XMDF Output Compression == 9`.

The default is for compression level 1, this should result in smaller .xmdf results files for the 2016 version of TUFLOW!

28 Hazard Time Output Cutoffs

Time output cutoffs can now be specified using any of the hazard routines by specifying the Hazard ID.
`Time Output Cutoff <Hazard ID> == [ <cutoffs> ]`

For example:

`Time Output Cutoff ZAEM1 == 1, 2, 3`

`Time Output Cutoff ZPC == 1, 2, 3`

Valid options are now:

`Time Output Cutoff Depths ==`

`Time Output Cutoff VxD ==`

`Time Output Cutoff <Hazard ID> ==`

29 **New Map Output Parameters**

New output parameter types are available for the DAT, XMDF, NC, TGO, ASC, FLT and GIS output formats.

Map Output Data Types == h v q RFR RFC VA

(a) RFR = Rainfall Rate (units mm/hr or inches/hr)

(b) RFC = Cumulative Rainfall (mm or inches)

(c) VA = Velocity Angle (angle of flow)

30 **XMDF Time Output Name Correction**

The display output name for time outputs in .xmdf file have been corrected for US Customary units (ft). Previously, the result dataset name included "metres" even if model was in US Customary (ft). For example, "Time of First Depth >0.10 m" will now be "Time of First Depth >0.10 ft".

31 **ArcMap Null Values**

Previous TUFLOW releases required that the dummy null value within the TUFLOW empty shp files be deleted prior to use as model inputs. This task is no longer required. ArcMap null values are now ignored in the .dbf file for numeric fields.

32 **TUFLOW AD Outputs**

TUFLOW AD (Advection Dispersion) outputs now compatible with Map Output Formats other than .dat or .xmdf.

Time-Series and Profile Output

33 **Combined 1D and 2D Reporting Locations**

A new 1D and 2D Reporting Locations feature allows water levels and flows from both 1D and/or 2D domains to be output in a combined manner as follows:

(a) Time-series results across both the 1D and 2D sections of the model are combined. For example it is possible to digitise a reporting location flow line across 1D and 2D domains, including multiple 2D domains, and TUFLOW will sum the flow across any 1D channels intersected by the line and all the 2D cells. Importantly this flow is tracked every timestep so that maximum flow reported is the true maximum, not the sum of the flow maximums for each component, which could occur at different times.

(b) Reporting location points and lines are digitised into a 0d_RL GIS layer containing only a single attribute (the name of the reporting location). The points and lines can be in the same layer or different layers. Points are treated as water level output and lines as flow output. For a point snapped to a 1D node, the 1D water level is used, if no 1D node is snapped a 2D water level is output.

(c) The Reporting Location outputs are contained in the new plot\csv\ folder. The following files are produced:

  (i) _RLL_Q.csv   = flow time-series;

  (ii) _RLL_Qmx.csv   = maximum flow information;

(iii) _RLP_H.csv   = water level timeseries; and

(iv) _RLP_Hmx.csv   = maximum water level information.

(d) As well as the maximum water level and flow information, the time when these occur, the water level at maximum flow and vica versa, and the maximum change between timesteps are also output to the mx.csv files.

(e) The reporting locations are also output to the plot\gis folder in the _PLOT GIS layers. Their time-series data can be easily viewed and displayed using the new TuPlot QGIS Plugin.

(f) Other plot output data types will be added to this feature in future releases.

34 **1D Output Type Command**

A new ecf command is available to control which type of 1D results are output.

`Plot Output Data Type == [ {H Q V S} | <string> ]`

This command is similar to the .tcf command "`Map Output Data Types ==`", but for 1D time-series output. Options are:

(a) A = Flow Area

(b) E = Energy

(c) H = Water Level (default)

(d) Q = Flow (default)

(e) S = Structure flow (default)

(f) V = Velocity (default)

35 **Output File Write Controls**

A new .tcf command is now available to control the writing of 1D output result files.

`Output File [ EXCLUDE | INCLUDE | {ALL} | NONE ] == <string>`

Options are presently confined to 1D output as all 2D output is already controlled by the user. Options that can be included in `<string>` are below.

(a) EOF to include/exclude the 1D output at the end of the .eof file.

(b) TS, TSF, TSL, TSMB, TSMB1D2D, 1D_MMH, 1D_MMQ, 1D_MMV and/or 1D_CCA to include/exclude these GIS output layer(s).

Note that the last occurrence of this command prevails, and all previous occurrences are ignored. For example:
`Output Files NONE`
`Output Files INCLUDE == TS, 1d_CCA`

Check Files and GIS Layers

36 **Additional Check and Result File Style Attributes**

Additional style attributes have been added for a range of check and results GIS files. These include rotation, magnitude and/or type attributes. This allows for advanced rendering when using the shapefile format in QGIS / Arc Map. The extra attributes are also output in MIF/MID files as they can be viewed in QGIS, however the style information is not shown. The changed files are:

(a) Check Files

  (i) _sac check now contains a column for boundary name and boundary group.

  (ii) _wllp_check contains a 1 character type field which contains "X" where the wllp intersects the 1d_nwk layer, B for a bridge channel, C for a culvert channel and A if WLL method A is used.

  (iii) _nwk_N_check contains two additional attributes "No_Channels" (the number of channels connected to the node) and "Pit_Connected" (Y if a pit is connected, otherwise "N").

(b) Results Files

  (i) _TS file now contains three additional fields

    • "Style_SF", a scale field from 0-1.

    • "Style_dir", a direction field. The direct value uses a compass bearing convention (0 degrees north).

    • "Style_type" a single character field containing Q, V or H depending on the result type.

    For the _TS output this change is only applied for shapefile format, this ensure that these can still be visualised with the mitools.

  (ii) _mmQ and _mmV contain two additional fields. A scale field from 0-1 to size the arrows and a direction field. The direct value uses a compass bearing convention (0 degrees north).

  (iii) _TSL contains a scale field.

  (iv) _TSF contains a scale field and also a type "C" for channel and "N" for a node.

XF Files

37 **1D XF Files**

XF files are now used to speed up the 1D tabular input (e.g. XZ, NA, HW, and BG) data. An xf folder is created under the input directory and the data is output there on the first reading of the files. Subsequent starts check the save date and if the xf file is newer speeding up the model initialisation

1d_tab layer .csv files containing data for 1D cross-sections (XZ) and nodal storage (NA) tables are now written to an XF file format by default. For large 1D model domains this can significantly decrease the start-up time for subsequent simulations.

ERRORs, WARNINGs and CHECKs

38 ERROR 2129 now occurs if no .tbc file has been specified for a 2D domain:

```
ERROR 2129 - No Boundary File Specified for 2D Domain <domain index>
```

Previously the message below was output to the screen / log, though the simulation window disappeared.

```
No Boundary File specified for 2D Domain Domain_001.  Bailing out.
```

39 If a landxml TIN file is written on a single line, an error occurred which caused the console window to close when looking for the start of the TIN data. TUFLOW checks for this and gives an ERROR 2315. Also refer to http://www.tuflow.com/forum/index.php?/topic/1294-landxml-from-autocad-civil-3d/

40 If using soils layer, inactive cells are no longer checked for input data, previously ERROR 2314 was returned even if the cell was inactive. The error only occurred if "Set Soil == " was not specified in the .tgc.

41 The intel Fortan compiler used by TUFLOW has limited support for files with UTF-16 encoding. ERROR 2416 is returned if a 12D Solutions TIN file (.12da) with unsupported file encoding is encountered.

42 A new ERROR message has been added if a Global Rainfall loss command occurs after the Global Rainfall BC. Previously these commands were ignored. For example, an error will occur for the following commands as the rainfall loss command occur after the rainfall BC command.

```
Global Rainfall BC == rainfall_name

Global Rainfall IL == 10

Global Rainfall CL = 2
```

43 ERROR 2010 previously applied to inactive cells for a TUFLOW GPU simulation. As such, previously models required a Material ID for all 2D cells (active or inactive required). This error now only applies to active cells.

44 ERROR 2520 message has been added if a GWL or GWD command occurs prior to a soil command. A "Set Soil", "Read GRID Soil" or "Read GIS Soil" command is required in order to activate the soil infiltration feature. If a ground water level or ground water depth command is specified prior to the soils being activated an error message is returned.

45 A new error message has been added if a 1d_bc layer is read into the .tbc file instead of the .ecf or within a 1D domain. The 1d_bc and 2d_bc GIS inputs have a different number of attributes. As such, reading a 1d_bc layer into the 2D tbc file previously caused ERROR 2041 to be returned, this error indicates that the issue is with a fixed field input. This has been replaced with ERROR 2340 :

```
ERROR 2340 - Less than 8 attributes for 2d_bc file <filename>
File appears to be a 1d_bc file and not a 2d_bc.
```

46 Reporting for ERROR 1025, which occurs if there is an inconsistency in the structure input data has been improved.

47 Previously, TUFLOW did not exit the simulation if "ERROR 1101 - Less than the mandatory 20 1d_nwk attributes found" occurred. This is now corrected.

48 A new ERROR message has been added:

```
ERROR 1164 - Bridge LC table elevations not in ascending order for bridge
channel
```

49 A new ERROR message has been added to cross-check the 1D timestep is a multiple of the timestep for <u>all</u> 2D domains:

```
ERROR 1299 - 1D timestep is not a multiple of timestep for Domain…
```

Previously, TUFLOW would proceed on the basis that the 1D timestep was a multiple of the smallest and largest 2D timesteps, or would be adjusted downwards so it was a multiple. If three or more 2D domains exist, it was possible that the 1D timestep of one or more of the 2D domain(s) was not a multiple, which could result in significant mass error values as the 1D timestep was out of sync with one or more of the 2D domains. For example, if there are three 2D domains and their timesteps are 3, 5 and 15s, and the 1D timestep is 1.5s, then the 1D domain would be out of sync with the 2D domain with a timestep of 5s. The new ERROR message now prevents this situation from occurring.

If `Defaults == Pre 2016` is set this check is not carried out.

50 WARNING 2460 has been escalated to an ERROR. Previously, TUFLOW issues one of the warning messages below if either of the rainfall factors (f1 or f2) were set to 0:

```
WARNING 2460 - RF f1 and/or f2 attribute set to zero - no rainfall will be
applied.
```

```
WARNING 2460 - SA RF Catchment_Area and/or Rain_Gauge_Factor attribute set to
zero - no rainfall will be applied.
```

This now produces an ERROR 2460 and halts the simulation. The message level can be controlled with the tcf command:

```
ZERO RAINFALL CHECK == [ {ERROR} | WARNING ]
```

If `defaults == Pre 2016` is set, this is error is reinstated as a Warning.

51 Changes have been made to the messaging options for handling unsupported GIS objects.

GIS software typically store vector data in three broad geometries: points, lines and regions. Within these geometry types, different GIS packages may offer a variety of digitising options. For example, when drawing a line object in MapInfo the user has the option for a line, a polyline and an arc. From left to right the editing buttons to digitise a line, polyline and arc in MapInfo are:

These different line types are stored differently in the MapInfo .mif file. An extract of a .mif file which showing a line object (red) a polyline object (green) and an arc object (blue) is provided below.

```
Line 340604.21 5782377 340612.07 5782369.48
    Pen (1,2,0)
Pline 3
340609.55 5782359.99
340614.35 5782361.73
340623.19 5782363.59
```

```
    Pen (1,2,0)
  Arc 340630.84 5782358.24 340640.43 5782382.01 180 270
    Pen (1,2,0)
```

Various GIS packages handle the advanced GIS geometries (such as arcs) differently. For example if converting a MapInfo arc object using QGIS, the arc object is converted to a polyline with vertices along the length. For consistency between packages and to provide better support across GIS platforms not all GIS geometries are supported by TUFLOW. For lines, arc objects are not supported (but line and polyline objects are both recognised). For region objects rectangles, rounded rectangles and ellipses are not supported.

Two special cases of unsupported geometries are "Text" objects used to annotate GIS layers and "None" or "Null" objects, which GIS software may add to the layer to indicate deleted objects (particularly if using the shapefile format).

A new .tcf command performs a check in the GIS routines for geometries not supported by TUFLOW.

`GIS Unsupported Object ==` [{ERROR} | WARNING]

This command sets whether the TUFLOW simulations stops with an error, or issues a warning and the simulation continues. There is a severity level component that can be specified using a vertical bar and value. The options are level 0, level 1 or level 2. For example:

`GIS Unsupported Object ==` ERROR | Level 1

`GIS Unsupported Object ==` WARNING | Level 2

The severity levels are:

- Level 0: No checks on unsupported geometries (i.e. previous behaviour)

- Level 1: Check for ellipses, rectangles, rounded rectangles and arcs (curved arcs)

- Level 2: All the level 1 checks as above plus checks for null and text objects.

The default for the unsupported objects is for `GIS Unsupported Object ==` ERROR | Level 1.

52 An additional command has been added to provide user messaging control when a supported GIS object is encountered, though is being ignored. Various TUFLOW inputs expect specific geometries. For example a 1D boundary (1d_bc) can contain points snapped to 1D nodes, or region objects to apply boundaries to multiple nodes falling within the polygon. So whilst a line or polyline is a supported object (see point above), any line objects in a 1d_bc layer are not used and TUFLOW will issue a WARNING message.

`GIS Supported Object Ignored ==` [ERROR | {WARNING}]

The above .tcf command controls if ignored GIS objects cause the TUFLOW simulation to stop with an ERROR, or issues a WARNING and the simulation continues.

53 Coincident points in a breakline layer now produce a WARNING message:

```
WARNING 2370 - Ignoring coincident point found in layer…
```

Only the first coincident point value is used.

54 A new WARNING message has been added:

```
WARNING 1132 - Channel <ID> assigned an unusually high form loss per unit
length via the Exit_Loss attribute
```

55 Added new message (WARNING 2508) if the initial soil moisture exceeds the soil porosity.

56 A new CHECK message has been added:

```
CHECK 1421 - Top level of bridge cross-section data differs to that in loss vs
height table
```

57 Messages 2314 through 2317 had been inadvertently assigned to both TIN and soil issues. The TIN related messages have been re-assigned to 2514 through 2517 to avoid duplication.

58 Non-spatial messages are now output to the _messages.mif or _messages_P.shp. These are written to the top-right spatial extent of the model.

59 The _messages.csv file now correctly loads into Excel. Previously some columns would load together as one column.

60 Wiki URLs are now embedded in ERROR, WARNING and CHECK messages to the .tlf file and as an attribute to the _messages GIS layer. This allows for direct links to the Wiki webpage with useful information about the message using a right click function in a text editor like UltraEdit. Note that this process may differ slightly between text editors.
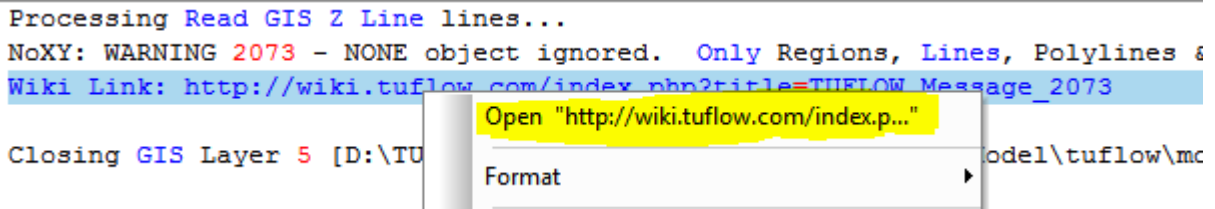
wiki.tuflow.com/index.php?title=TUFLOW_Message_2073

Page | Discussion — Read | View source | View history — Search

TUFLOW Message 2073

TUFLOW Message

WARNING 2073 - GIS object below ignored. Only Points, Lines, Polylines & Region Centers used.

Alternate Message

WARNING 2073 - Object below ignored. Only Lines, Polylines & Multiple Polylines used.

WARNING 2073 - Object below ignored. Only Points, Lines, Polylines & Regions used.

Message Type

WARNING

Description

GIS object below ignored. Only Points, Lines, Polylines & Region Centers used. Other GIS objects such as elipses, rectangles, etc should be avoided.

Suggestions

GIS objects should be points, lines, polylines and regions should be converted to valid objects. For example convert elipse objects to regions. In MapInfo this can be done by selecting an object and using Object >> Convert to Region or Object >> convert to polyline.

Links

- Discuss Message 2073
- Main page

Previous	Up	Next
⬅ Message 2072	⬆ 2xxx Messages	Message 2074 ➡

TUFLOW GPU Module

61 GPU Virtual Pipes

A new GPU Solver virtual pipe feature can now be used to account for flows in drainage networks, without modelling the complete pipe network. One or more pits layers are provided to simulate the inlets/drains/gullies and the flow into the pits. This flow can either exit the model (for example, if the pit flow can assume to never surcharge) or be redirected to one or more outlets based on a pipe network ID. Maximum capacities at the outlets can be specified to trigger surcharging of pits connected to the outlet. This feature is the first stage of providing full pipe and manhole flow modelling in the GPU Solver.   For more information, please see the section on Virtual Pipes in the 2016 TUFLOW manual.

62 GPU Boundaries

(a) GPU SA boundaries can now be distributed over wet cells, stream or pit cells using the standard TUFLOW Classic commands. All of Classic's SA functionality is now supported by the GPU solver. Previously only SA ALL and SA STREAMLINES ONLY were supported.

(b) **Correction to the highlighted text of the 2016-03-AA Manual extract below:**

Please see Item 143 belowfor changes to the TUFLOW GPU boundaries implemented in Build 2016-03-AD. This includes new options for a specified slope boundary.

A normal flow boundary in the GPU Solver is not activated by having the HT boundary level below the ground level. A normal flow boundary is invoked by specifying a HT boundary with a water level of -9999. The GPU Solver assumes uniform flow based on the ground slope of adjoining cells.

Note: for builds prior to 2016-03-AD the assumed water levels at these boundary cells are not presently output and the cells may appear as dry or with very low levels. Also the water levels at the common cell corners of adjoining active cells may be incorrectly low in the output. Improved output at automatic boundary cells for the GPU Solver was corrected for the 2016-03-AD build – see Item 143(e).

A water level boundary is required for water to exit the model. If the water level boundary value is less than the ground level, a normal flow boundary is applied at the exit point (this is effectively an automatic HQ boundary on a cell by cell basis). Water can enter the model via a water level boundary so the influence of tides, storm surges and tsunamis can be modelled.

Table 10-1 TUFLOW GPU Boundary Types

Boundary Type	TUFLOW Layer/Command	Notes
Water Level	2d_bc (HT): Read GIS BC	Multiple HT boundaries can be used and applied in a similar manner to TUFLOW Classic. If the water level is at ground level or below, normal flow is applied at the boundary.

63 GPU Maximum Outputs

The GPU Solver now tracks the maximum value and time of maximum for water level, velocity and Z0 (VxD). Maximums output for these data types will now be the tracked maximum.

64 GPU Time Output

The GPU Solver now supports the map based Time output option for first time of exceedance, and duration of exceedance. This can be done for one or more depths using:

`Time Output Cutoff Depth == <y1> [<y2>] [<y3>]`

65 GPU NaN Check

The GPU Solver now counts and records any NaN occurrences. NaN stands for "Not a Number" and can occur if the solution "bounces" or goes unstable. The GPU Solver winds back to the previous timestep, reduces the timestep and repeats the calculations, until the simulation stabilises using a shorter timestep. Viewing the location of any NaNs can be useful to check the model inputs for any erroneous data and to check the results. The location of any NaNs are reported to the .tlf file and _messages layer as a "WARNING 2550 - <number> instability timestep corrections recorded at cell".

66 GPU Thin Line as Thick

The .tgc command `Thin Line as Thick ==` default is now set to "ON" in the GPU Solver (the default is "OFF" for Classic simulations).

67 GPU Global Rainfall

Global Rainfall .tbc commands can now be used by the GPU Solver.

68 GPU Velocity Output

The GPU Solver velocity output interpolation to cell centres and corners has been improved. Previously, velocities along the wet/dry edge were not always well represented. This is a change to how the velocity vector results are output at the cell corners.

69 GPU Messaging

Improved messaging that alerts the user to commands not yet supported by the GPU Solver has been included in the 2016-03 release.

70 GPU RAM Optimisation

GPU Solver runs now do not allocate RAM for Classic features not yet supported by the GPU Solver. Previously when using the GPU solver, some of the memory arrays used in the TUFLOW Classic were still allocated. This removes redundant memory allocation for GPU simulations. The new .tcf command can be used to switch this feature on or off (on is the default).

`GPU RAM OPTIMISATION == [ {ON} | OFF ]`

(a) Most of the new map output formats consume a lot of memory due to the triangular mesh approach used to extract results. The new map output format mesh use the same configuration as XMDF SMS TRIANGLES, four triangles per cell with a common vertex at the cell centre. DAT and/or XMDF (not with the SMS TRIANGLES option) formats use less memory to producing map output and tracking maximums.

(b) Grid based output (.asc, .flt, .nc, .wrr) can also consume large amounts of RAM because mesh interpolation information is stored for each grid output cell (not 2D cell). This approach is used to increase the write speed. Consider increasing the "`Grid Output Cell Size ==`" value to reduce RAM requirements if using grid based output (the default is half the smallest 2D cell size). The example below from a .tlf file shows how the grid output memory and mesh memory can represent a large portion

of the overall memory.

```
Grid output dimensions: 2007 rows by 2007 columns
Grid output memory ...........................    64 Mb
TIN and output mesh memory ...................   116 Mb
Mass balance tracking memory .................     0 Mb

No (evacuation) routes found.
Route array memory ...........................     0 Mb

Number of Gauges ... 0

Number of GIS Objects/Receptors ... 0
Output Gauge array memory ....................     0 Mb
Miscellaneous array memory ...................     0 Mb
Maximum boundary table size ... 2304
Temporary arrays memory ......................     0 Mb

Total 2D Domain(s) Real SP Memory requested ...  343 Mb
Total 2D Domain(s) Real DP Memory requested ...   32 Mb
Total 2D Domain(s) Integer Memory requested ...  115 Mb
Total 2D Domain(s) Logical Memory requested ...   44 Mb
Total 2D Domain(s) Character Memory requested .    3 Mb
```

Minor Enhancements

71 XF Files are disabled if the copy all (-ca) switch is specified. This will copy the original datasets. For example, if a DEM is specified using `Read GRID Zpts ==` grid\dem.asc, if XF files were on (the default), previous versions of TUFLOW would copy the processed elevations (e.g. "grid\xf\dem.asc.xf4") instead of the original dataset.

72 If running TUFLOW in test mode (-t) the simulation now terminates later after the writing off all check files, including the DEM_Z and DEM_M.

73 A new .tcf command has been added to set the upper limit on the number of 1D channels during model start-up.

`Maximum 1D Channels ==` [{100,000} | <number of channels>]

This value is only used during the first pass through the model input files to allocate temporary space. After this pass, only memory required for the actual number of channels is allocated. The default is 100,000, but can now be increased (or decreased) if required. The upper limit on the number of nodes is set to twice the number of channels.

74 Improved memory (RAM) optimisation for Classic CPU solver and GPU solver has been undertaken for the 2016-03 release.

75 New "Verbose == LOW" option that outputs information to the screen and the .tlf file at a level of detail between the two other available options, "ON" and "OFF". The default setting for the 2016-03 release is "Verbose == LOW".

76 Information on number of 2D active and inactive cells has been added to the .tsf file.

77 Additional empty files for soils and output zones (2d_soil and 2d_oz) are now output if using the "Write Empty GIS Files ==" command.

78 Energy is now tracked every timestep, therefore if the energy (E) option in "`Map Output Data Types ==`" has been specified and maximums are being tracked, energy will appear as a dataset in the map output maximums.

79 An Evacuation Route (2d_zshr) can now be triggered using "Energy Depth". This is calculated as:

$$Energy\ Depth = d + \frac{V^2}{2g}$$

This is specified with the cut off type as "Energy". Available options now include Depth, VxD, Velocity or Energy.

80 Changes have been made to the `Read GIS Gauge Output` layer. See also point 19 above.

(a) Shape file format is now supported for Gauge Outputs.

(b) Previously TUFLOW would write a file for each gauge input layer using an output name based on the gauge layer. This caused the layer to be overwritten if running multiple events / scenarios. A single point file layer is now output with the output name based on the simulation name.

(c) An error was previously triggered if the "Zpt" flag was included in the command syntax. This has been fixed.

Bug Fixes

The following list of software bugs have been fixed:

81 Bug fix that caused error, "Should not be here conchn_2_pumps_iOC_3" to be returned when using a y-Q curve for a pump.

82 Bug fix that caused ERROR 1276 message to not appear in the messages layer.

83 Bug fix in the QGIS project (.qgs) file which caused input files to be incorrectly referenced. The geometry type (_P = points, _L = lines or _R = regions) was added to input filepaths.

84 Bug fix that would cause Warning 0317 to occur if "Write Timeseries Online" or "Write PO Online" options were enabled. Previously, "WARNING 0317 - Exceeded limit of input GIS layers that can be tracked (layer referencing may not be complete)" could be returned. This warning message does not affect results at all.

85 Bug fix in READ GIS Gauge Output when using the ZPT option. Previously ERROR 2343 was reported if the READ GIS Zpt Gauge Output == option was used. For example, ERROR 2343 - Number of Gauge Output objects exceeds number of attribute data records. Total number of data records = 0.

86 Bug fix for _sac check file that caused incomplete data to read into MapInfo. In the MID (data) file the description was output as:

"Lowest 2D cell for Inflow "33|Local" (ZC = 31.93; No. Cells = 1559)"

However, when imported to MapInfo the quotes around the name details caused the information after the underlined section to not import correctly. This did not affect QGIS or ArcMap.

Additional boundary name and boundary group for the _sac check are also output (see point 36 above).

87 Bug fix for multipoint shapefile causing error "Should not be here [gisWriteHeader_qgs]".

88 Bug fix for raster outputs (.asc or .flt) if the output zone extends outside the origin of the output grid. Previously the raster output was shifted.

89 Bug fix in operational pump wind down period.

90 Bug fix in rate of Infiltration. It could previously be incorrect if the soil became fully saturated.

91 Bug fix for 1D commands in 1D/2D models that were not activating the 1D/2D linkage correctly. This occasionally occurred when using commands prefixed by "1D".

92 Bug fix where .tcf lines starting with "1D" outside of an If Scenario block were not applied.

93 Bug fix where 2d_lfc Invert attribute sometimes did not recognise the 99999 (ignore) option.

94 Bug fix that did not clear flow regime flags (.TSF layer and .eof file) for new 2013 structure routines, resulting in the flags not always being correct.

95 Bug fix for the write procedure for the 1d_bc_check layer. Previously, in some situations it would not be closed until the end of the simulation, restricting the file from being opened by GIS software.

96 Bug fix for WLL velocity display for culverts.

97 Bug fix for FLCs that did not add (accumulate) if using "Read Grid FLC ==". All FLC commands (eg. Set FLC ==, Read GIS FLC == and Read Grid FLC ==) accumulate the FLC (i.e. they don't overwrite the FLC value as do most other related commands).

98 Bug fix in operational structure logic blocks. The name of an input variable was occasionally lost and appeared as blank in the output .csv files.

99 Bug fix for final result timestep writing in map output format. Occasionally, if a model was unstable it would not write the results for the final timestep when the instability occurred.

100 Bug fix in "CHECK 1206 - Pit occurs at the same location as a node, manhole or another pit [IDs:…". Previously, the check file would not report the correct ID.

101 Bug fix for restart files that did not work correctly due to there being a comma in a 1D Node ID. This has been corrected, however, it is strongly recommended not to use commas in any ID or label, as this may cause secondary issues with reading .csv output.

102 Bug fix for Read Grid Soil command, if it was specified before Read Grid Zpts in the geometry control file.

Build 2016-03-AB – New Features, Enhancements and Bug Fixes

Due to the Items 103 and 104, any users running TUFLOW GPU should upgrade to 2016-03-AB.

Note, any TUFLOW GPU models using virtual pipes will require the command "Virtual Pipes == ON" to be added to the .tcf for this feature to be utilised.

The new features, enhancements and bug fixes in Build 2016-06-AB are:

103 Bug fix for reading gridded rainfall data (using either Read Grid RF, or Rainfall Control File inputs) with GPU. Previously, rainfall boundary condition inputs were not updated correctly if the map output interval exceeds the interval used by the rainfall grid. This only affects TUFLOW GPU simulations, not TUFLOW "classic" simulations. This item is fully documented on the TUFLOW forum. Refer to the TUFLOW forum post for more details.

104 Fixed issue with TUFLOW GPU simulation when using SA inflows that are proportioned to depth (which is the default behaviour). NVidia driver update 368.81 (July 2016) was incompatible with the March TUFLOW GPU release 2016-03-AA. It caused inflow issues at boundaries that were wetting / drying. This item is documented on the TUFLOW forum. Refer to the TUFLOW forum post for more details.

105 Additional Empty (template) GIS layers now being written:

    (a) 1d_pit (virtual pipes)

    (b) 2d_gw (ground water level or ground water depth)

    (c) 2d_rf (rainfall) empty layer now has point geometry (_P) if using shapefile format with "`GIS Format == SHP`" defined in the .tcf

    (d) 2d_rl (reporting location)

    (e) 2d_zshr (evacuation route)

106 Maximums now tracked for Bed Shear Stress ("`Map Output Data Type ==`" command includes BSS) and Stream Power ("`Map Output Data Type ==`" command includes SP) and store maximums is on (default). For example the following would turn on water level, depth, velocity, bed shear and stream power outputs:

`Map Output Data Types == h d v BSS SP`

107 The option to ignore MapInfo Projection Bounds when the MIF Projection Check is processed. This requires the .tcf command "`MI Projection Check Ignore Bounds == ON`".

The project line in a MapInfo text file may look something like the below:

`CoordSys Earth Projection 8, 104, "m", 177, 0, 0.9996, 500000, 10000000 Bounds (-7500000.0, 2000.0) (8800000.0, 20000000.0).`

The portion of this projection line to the left of the "Bounds" command defines the parameters of the projection, the portion of to the right of the "Bounds" defines a rectangle in which the coordinate system is valid. Some GIS software will write these bounds differently for different layers within the same projection. For example, the two projections below are identical, though with slightly different projections bounds.

`CoordSys Earth Projection 8, 104, "m", 177, 0, 0.9996, 500000, 10000000 Bounds (-7500000.0, 2000.0) (8800000.0, 20000000.0).`

```
CoordSys Earth Projection 8, 104, "m", 177, 0, 0.9996, 500000, 10000000 Bounds
(-7499999.0, 2000.0) (8800000.0, 20000000.0).
```

When not using the ignore bounds option, the differences in the projection lines above would cause the model to stop, if ignoring bounds the model would continue. The command "`GIS Projection Check == {ERROR} | WARNING`" can be used to control whether TUFLOW will halt the simulation with an error or give a warning and continue the simulation if there is mis-match in the projection of the GIS file and the projection of the model.

108 Extra error messaging for GPU simulations if features are not supported. For example if rainfall losses are specified in the materials file during a GPU simulation, Warning 2783 message is returned.

109 The "`Read GIS Receptor Record Gauge Data ==`" command has been extended to include the reporting locations functionality. As well as outputting the water levels at gauge objects (defined as points with "G_" type in the 2d_po layer), the water levels at all point reporting locations (1D or 2D) as well as flows for all line reporting locations (1D and/or 2D).

110 `Map Output Cutoff Depth` is now supported in the 12D Civil Solutions .tmo output format.

111 New .tcf command "`WRR Write Method ==` Fast".

(a) WaterRide result file write speed has been improved. When finishing a TUFLOW simulation and writing the maximums to the .wrr file, the file needs to be closed and the maximums are written at the start of the .wrr file. For large models, this could be slow in the 2016-03-AA version of TUFLOW. If "`WRR Write Method ==` Fast" is specified in the .tcf, files are opened as a buffered file to write the maximums for faster processing.

112 Addressed an inconsistency in the way gridded rainfalls are applied. Gridded rainfall inputs were originally developed with forecast rainfall applications. As such, rain in the gridded rainfall at time t, is applied between time t and t + 1. This has been changed to be consistent with other rainfall boundaries, and the value at time t is assumed to have fallen in the period t - 1 to 1.

For example, if the rain falls at a cell are listed in the table below. For the 2016-03-AA version the value of 10mm at time 1 hour was applied as a forecast rainfall and applied between time 1.0 and 1.5 hours. For the 2016-03-AB version this rainfall depth of 10mm and time 1.0 hours is applied as a "recorded" rainfall that fell between time 0.5 and 1.0 hours.

Time (hours)	Rainfall depth (mm)
0.0	0
0.5	0
1.0	10
1.5	20

113 Bug fix when Read GIS <parameter> command was read in immediately after reading a soils layer. Could incorrectly interpret the GIS layer as soils layer.

114 Bug fix that would write a binary null to the .tlf when Check 2370 was output. This could cause issues with opening the log file in TRIM

115 NetCDF output options have been removed for 32 bit builds. A conflict in the underlying HDF5 compression libraries was causing issues with the .xmdf outputs and could cause the simulation to disappear with no error message written when trying to initialise the .xmdf file. The 64 bit versions of TUFLOW were unaffected.

116 Fix bug that would cause "`Write Check Files == OFF`" to be written to the ,tlf even if check files were on.

117 Bug fix that caused the message layer if using `GIS Format == SHP` not to be closed off correctly if running in test mode using the batch file –t option. This could cause the message_P.shp layer not to open in GIS.

118 Bug fix to TGO outputs.

119 Bug fix to `Read GIS Receptor Record Gauge Data ==` command.

120 Bug fix to reporting location line (RLL) output if the reporting location did not cross any 1D channels.

121 pBlockage attribute for Virtual Pipes now works as intended.

122 Bug Fix that could cause the WRR file not to be written correctly unless another gridded output (such as .asc or .flt) was also output.

123 Bug fix that could crash TUFLOW at first write if external water level lines were used.

124 Bug fix that could cause simulation to crash during write of plot output

125 Bug fix that could cause the rainfall grid dimensions to not match the TUFLOW model domain if the output grid size is not specified with either the "`RF Grid Size (X,Y) ==`" or "`RF Grid Size (N,M) ==`" commands.

126 Bug fix for TUFLOW GPU models with no temporal output specified. Previously this would cause the simulation to keep repeating the 1st timestep.

127 Bug fix with SMS tin write that could cause it not to be recognised in the latest version of SMS.

128 Fix bug that didn't assign "BB" as below deck flow for a structure group.

129 Additional information is output to the .tlf.

130 Error Message 1422 is now output for a "BB" Bridge channel if no loss vs height (LC or BG) table or 1d_nwk Form_Loss attribute is specified. Previously "Should not be here [jt,icc=] - please contact support@tuflow.com" was output and the simulation was stopped.

131 Fixed bug that would cause an error if using "`Read GIS X1D Network ==`" instead of "`Read GIS ISIS Network ==`" in the .tcf. Both options now work.

132 New Matrix Blockage Methodology. This allows for the blockage of culverts (I, C and R type culverts) to be varied based on the Average Recurrence Interval (ARI) of the flood simulation. To set this up the pBlockage attribute in the 1d_nwk GIS layer needs to be changed from a float (numeric) to a character type (maximum width of 50 characters). A blockage category is then specified in the 1d_nwk pBlockage attribute. A matrix file is provided which links the blockage category to the ARI of the flood event.

In the blockage matrix file, the blockage categories are listed in the columns, and each row of data represents the blockage for a specific ARI. For example for the 100year ARI, and blockage category "B" a 10% blockage will be applied. If the current ARI is between the values in the table it is interpreted (linear interpolation), for example for a 50year ARI event, blockage category "C" will have a blockage of 13.75%.

Example Blockage Matrix File

ARI	A	B	C	D	E
1	0	10	10	10	100
20	0	10	10	20	100
100	0	10	20	50	100
2000	0	20	50	70	100
10000	0	50	70	100	100

Two methods for applying the blockage are provided these are reduced area method (RAM) which reduced the culvert flow area the same a numeric blockage on the 1d_nwk blockage attribute and Energy Loss Method (ELM). For the energy loss method, the flow area of the culvert is not reduced rather an entry loss coefficient is calculated based on the blockage Ollett and Syme (2016).

New Commands
- Matrix Blockage == ON | {OFF}
    o Turns on or off the blockage matrix functionality
- Blockage Method == RAM | ELM
    o Sets the blockage method to one of RAM or ELM. If this command is not specified an error will be given and the simulation will terminate.
- Blockage ARI == <ARI in years>
    o Sets the ARI for the current event, this would typically be defined in the .tef but can be in the .tcf

- Blockage Default == <category>
  - Set the default blockage category to use, if one is <u>not</u> defined in the 1d_nwk, i.e. the pBlockage attribute is left blank.
- Blockage Override == <category>
  - Sets <u>all</u> culverts to use this blockage category (regardless of the attributes in the 1d_nwk)
- Matrix Blockage File == <link to blockage matrix>
- Blockage PMF ARI == <ARI for PMF> (only required if PMF is defined in the matrix).

Example TUFLOW control file

```
Matrix Blockage == ON
Blockage Method == RAM !Reduced Area Method
Blockage Default == A
Matrix Blockage File == Blockage_matrix.csv
```

Blockage ARI command specified in the .tef

```
Define Event == 050yr
  Blockage ARI == 50
  BC Event Source == _event1_
  Timestep == 1
End Define
```

Full documentation of this feature is provided in the next update of the 2016-03 TUFLOW manual.

Build 2016-03-AC – New Features, Enhancements and Bug Fixes

Due to Item <u>133</u>, any users utilising the new structure group output should upgrade to 2016-03-AC.

Due to Item <u>139</u>, any users utilising the SRF feature with TUFLOW GPU should upgrade to 2016-03-AC.

133 Fixed bug that did not always correctly assign channels above and below deck to structure groups, thereby not correctly allocating the below/above deck flow split for some structures. This bug fix has no effect on the hydraulic computations.

134 New check files for structure groups. Two new check files are created if the model contains any structure groups (either automatically created, or via a "QS" type line in a 2d_po layer). The check files are both .csv files as follows:

   (a) <simulation name>_Str_Grp_All.csv, contains information for all structure groups, including single 1D structures.

   (b) <simulation name>_Str_Grp_Multi.csv, contains information for structure groups that are comprised of more than one 1D channel or are generated from a 2d_po "QS" line.

For each structure group, the name, primary channel, number of channels above and below, and the channel list for above and below are output as per the example check file below.

Name	Primary Channel	Below	Channels Below	Above	Channels Above
TC_C_0410	TC_C_0410	2	TC_B_0400|TC_C_0410	2	TC_B_0400.W|TC_C_0410.W
TC_R_0420	TC_R_0420	1	TC_R_0420	1	TC_R_0420.W
TC_R_0430	TC_R_0430	1	TC_R_0430	1	TC_R_0430.W
TC_B_0440	TC_B_0440	1	TC_B_0440	1	TC_B_0440.W

135 For a structure group including 2D flows, a "QS" type object needs to be defined in the 2d_po layer (as per the TUFLOW manual). A new error message (ERROR 2551) is returned if the 2d_po type attribute is set to "SQ" as opposed to "QS". Previously, if "SQ" was specified the model would continue, but only flow (Type "Q_") was output, not the structure flow.

136 Fixed bug with "BB" channels introduced at Build 2016-03-AB where if an energy loss is specified on the 1d_nwk "Form_Loss" attribute and no "BG" or "LC" table is specified, Build 2016-03-AB incorrectly produces an ERROR during initialisation. This has been corrected for Build 2016-03-AC.

137 ZUK0 hazard outputs have been added to TUFLOW GPU. This maximum is tracked at every timestep and the functionality is the same as for TUFLOW Classic. At present, only one hazard output can be tracked (per simulation) if using the GPU Module, whereas the Classic solver supports up to 10. Therefore, for maximum hazard tracking using the GPU solver only one of Z0 or ZUK0 can be tracked per simulation at present. Extending this feature to allow more than one hazard category being tracked in the GPU Solver is planned for a future update.

138 Variable geometry has been implemented on TUFLOW GPU using the variable Z Shape command (`Read GIS Variable Z Shape == <2d_vzsh layer>`). The range of triggers and options for TUFLOW GPU are the same as for TUFLOW Classic. As per Section 10.6.2 of the TUFLOW manual, due to the cell centre approach of TUFLOW GPU all thin breaklines are treated as thick for the TUFLOW GPU solver.

139 A bug that did not apply the SRF (Storage Reduction Factor) feature to the GPU Solver has been fixed. **If using the SRF with the GPU Solver, all builds prior to 2016-03-AC should not be used.** This bug fix does not apply or affect the SRF feature when using the TUFLOW Classic 2D solver.

140 The GPU Solver now includes the tracking of maximum water level at cells along water level boundaries. In Build 2016-03-AB, these cells may have appeared as dry.

141 The character length of the "Inlet_Type" attribute for the 1d_nwk layer in the empty file has been increased from 12 to 50. Note that the length of this character field can be manually increased (or decreased) via your GIS package; the maximum number of characters is 256 as per Table 5.2 of the TUFLOW manual.

142 New .tgc command that modifies the elevations in the model based on a user specified change in the conveyance from a base case set of results. Command syntax is:

```
Read GIS Zpts Modify Conveyance == <gis layer> | <K_factor> | <Water_Level_Grid>
```

This command requires a GIS layer of one or more regions defining where the bed level is to be adjusted based on an increase or decrease in conveyance (recommended GIS layer prefix is 2d_mod). The attributes of the GIS layer are not used for this feature. The terrain levels are adjusted based on the depth of water relative the water surface provided by $depth*(1 – f^{(3/5)})$, which is effectively a change in conveyance of $K*f$ where K is conveyance and f is the factor <K_factor>. For example, f = 1.2 would be a 20% increase in conveyance resulting in a deepening of the waterway). <Water_Level_Grid> is a grid surface in a supported format (eg. ASC, FLT). The grid does not need to be in the same resolution as the 2D domains, but is typically the _h_Max grid output by TUFLOW from a previous run or generated by the TUFLOW_to_GIS utility. Both .asc or .flt formats can be used, but the .flt file format is preferred as it is much faster to access.

This feature is useful for sensitivity testing the changes in flood behaviour due to a deepening or accretion of a river's bed based on a change in conveyance.

This command would normally be applied in the geometry control (.tgc) file, after all elevation (Zpt) commands as the Zpt adjustment is based on the Zpt elevations processed up until the location of this command in the .tgc file. An example is provided below:

```
Read GIS Zpts Modify Conveyance == gis\2d_mod_river.mif | 0.9 | a_h_Max.flt
```

Where:
- 2d_mod_river.mif contains polygons of the areas to be modified, in this case along the river. This can be in .shp or .mif format.
- 0.9 is the factor change in conveyance compared to a base case, ie. 10% decrease in conveyance.
- a_h_Max.flt is a raster grid of the peak water level grid from a base case run. It is recommended to copy this grid layer from the results folder and place it somewhere under model\gis\.... If this grid layer varies with AEP, it is recommended to use the one .tcf with a variable to define the AEP, or alternatively a scenario variable.

Build 2016-03-AD – New Features, Enhancements and Bug Fixes

For the GPU Solver, due to Items <u>143</u> below, which include important enhancements at water level boundaries and a bug fix, it is recommended that Build 2016-03-AD or later be used for GPU Solver simulations, unless for legacy reasons use of earlier versions of the GPU Solver is justified.

For the TUFLOW Classic Solver, due to Item <u>147</u>, any users utilising the CWF (cell width factor) feature via the "`Read GIS CWF ==`" or "`Read Grid CWF ==`" commands should upgrade to 2016-03-AD unless for legacy reasons earlier versions need to be used.

143 Improvements to the treatment of water level boundaries using the TUFLOW GPU Solver have been made as documented below.

   (a)   General improvements to the handling of HT and HQ boundaries in the GPU Solver have been made where strong incoming flows occur (eg. an ocean tide or storm tide boundary). The previous approach in some situations was prone to generating unrealistic flow patterns and oscillations near the boundary on incoming flows. The new approach may cause slight changes in GPU Solver results near H boundaries of well-conditioned models. Due to the nature of the code changes, it has not been possible to provide a backward compatible switch that reverts to the previous approach.

   (b)   A HQ boundary water surface slope is now supported. The water level slope for a HQ boundary is specified in the same manner as TUFLOW Classic using the "b" attribute of the 2d_bc layer. For example a "b" attribute of 0.01 applies a 1% water level slope. Note that there is a difference in the underlying method in which this slope is applied between the two solvers:

      (i)   For a TUFLOW Classic simulation, the specified water level slope is used to generate a height versus flow (stage-discharge) curve at the start of the simulation. The total flow across the HQ boundary is evaluated each timestep and the corresponding water level from the curve is applied to all the HQ cells.

      (ii)   For the GPU Solver each HQ cell is treated separately and the specified water surface slope is applied to each HQ cell that joins a non HQ, active (wet) cell. The slope is assumed downhill in the direction of flow. Inflow from a HQ boundary is disallowed.

      (iii)   Whilst the underlying method used to apply the specified slope HQ boundary for TUFLOW Classic and the GPU Solver is slightly different, testing has shown that the results at the boundary are similar.

   (c)   The HT Auto (with level of -9,999), as described in Item <u>62(b)</u> of these release notes has been enhanced. Previously if the ground slope along the boundary varied significantly, this could cause issues and reverse flows, especially if the ground levels produced an adverse slope. The new method assumes constant water depth where flow leaves the model downhill. Where HQ boundary cells are uphill, water elevation is assumed constant. Inflow from a HQ boundary is disallowed.

   (d)   The "`Boundary Viscosity Factor ==`" command is now supported for HT boundaries in the GPU Solver, and is applied in the same manner as for TUFLOW Classic. Increasing this number from the default value of 1.0 can help to stabilise HT boundaries that experience unrealistic velocities and circulations, often associated with oscillations in the water levels. Sensitivity testing is recommended prior to adopting values greater than 1, but testing thus far indicates values up to 5 have very minor effects on the results of stable simulations.

(e) Water level values at H boundary cells for GPU Solver simulations are now correctly output. Previously, the assumed water levels at the boundary cells were not output and the cells may appear as dry or with very low levels. Also the water levels at the common cell corners of adjoining active cells could be incorrectly low in the output.

144 Improvements to how the GPU Solver handles glass wall boundaries. Previously artefact velocity patterns could occur near glass wall boundaries. The updated method provides a better zero-slope boundary condition and reduces artefact velocity patterns at glass walls.

145 The treatment of the eddy viscosity term in the GPU Solver has been enhanced with slightly improved results in areas of rapidly changing velocity patterns. The new method is the default for Build 2016-03-AD and will produce slightly different results compared with the previous method. The previous method can be invoked for backward compatibility using "GPU Viscosity Method == Method A". The default is the new "Method B".

146 A bug in the application of the soil infiltration by the GPU Solver that causes strange results has been fixed. The bug does not occur in most simulations and in the few known instances the bug has occurred, the results from the GPU Solver are unrealistic with rapid travel times and excessively high velocities. If using soil layers as inputs to a GPU Solver simulation it is recommended to use Build 2016-03-AD or later as the occurrence of this bug is unpredictable.

147 Fixed cell width factor bug that causes the cell width factor applied via the "`Read GIS CWF ==`" or "`Read Grid CWF ==`" commands) to be applied incorrectly around some edges of the GIS shape or grid. The values applied were same values as output to the _uvpt_check layer. As of Build 2016-03-AD the values around the edges are now correctly applied and the changes will be evident by the changes in the _uvpt_check layer generated by 2016-03-AD. Testing has shown that this bug fix causes only very slight changes to results. The "`Read GIS FC ==`" and "`Read GIS Layered FC ==`" commands, which also can adjust the cell width factor values are not affected by this bug fix. This bug fix also does not affect the GPU Solver cell width factor application.

148 Licence free mode support for new tutorial models:

(a) The new TUFLOW Tutorial Model 9 on setting up Multiple 2D Domains will need to be simulated using Build 2016-03-AD or later if the model is to be run in licence free mode.

(b) The new 1D/2D linked TUFLOW-Flood Modeller Tutorial Model can be simulated without licence of either software using Build 2016-03-AD or later.

149 Maximum number of soil types has been increased from 100 to 1,000 for TUFLOW Classic and 255 for the GPU Solver.

150 Fixed an issue with the rainfall output from a TUFLOW rainfall control file (.trfcf) if the "`Output Drive ==`" command is specified in the .tcf file. The output grid files, which are written during model initialisation are written to a "RFG" folder, which is created in the same location as the .trfcf. If the "`Output Drive ==`" command is specified, these grid files are now written to the specified drive (previously the requested output drive was ignored).

151 The "Pause" command is now processed earlier in the simulation start up when looking for Set Variable commands. Previously any occurrence of a Pause command that was affected by a Set Variable command (eg. to stop a simulation is an invalid scenario was specified) the simulation would not correctly start up.

152 Fixed an issue with GPU runs terminating early if there was not a Map Output Interval aligned with the simulation end time. Previously, the simulation would finish cleanly, but at the last map output time, rather than the end time, which would affect other results such as time-series outputs.

Build 2016-03-AE – New Features, Enhancements and Bug Fixes

153 The 2016-03-AE version requires that the 2016-2017 maintenance support has been paid and the licence has been updated. Please contact sales@tuflow.com if you have any queries about your licence.

154 New runtime option –nc (no console) which disables the console window when TUFLOW is running. TUFLOW simulations will run, but no window will be visible to check model progress. This feature has been built in primarily for running bulk simulations on blade machines or the cloud, or on Linux platforms using a Windows emulator. The console window output can be captured to a file, for example, by adding "> sim_dump.txt" to the end of the TUFLOW.exe batch line.

155 .ecf (1D) command "`Output Data Types ==`" now works as documented in the manual. Previously "`Plot Output Data Types ==`" was required.

156 Fixed issue with "`Read Grid Location ==`" if the input raster grid had an x or y origin coordinate less than -99,999; previously an Error 2347 (Insufficient Header Information) was output.

157 Bug fix that could cause "`Should not be here iwrt>jw`" to be output to the screen when writing time-series results. If the simulation start time is less than zero (0), this bug fix may cause a shift by one time-series output interval in time-series results – the fix does not affect the hydraulic calculations or map output results. Backward compatibility is not available for this bug fix.

158 Empty (template) file for reporting location (`Read GIS Reporting Location==`) is now output as 0d_RL as opposed to 2d_RL, to be consistent with documentation.

159 Bug fix for x1d_nodes_check.dbf (external 1D nodes) file that caused an error to occur and stop the simulation. This did not affect simulations using the MapInfo (MIF) output format.

160 Fixed bug that did not correctly close advection dispersion .dat format files at the end of the simulation.

161 Bug fix that did not output the _TS format (GIS format time-series) data for a GPU Solver virtual pipe model if plot output (2d_po) data that contained only lines was specified. If a point object existed in a 2d_po layer, or there was no 2d_po layer, this bug does not occur.

162 Fixed bug that causes unpredictable results when SMS High Res is specified and the DEM_M or DEM_Z is being output. The two are incompatible, so the DEM_Z and DEM_M are no longer output if high resolution SMS output is specified.

163 Fixed bug which in some situations could cause the WLLp elevations from processed XS data to be incorrect. This does not affect the hydraulic computations and does not affect elevations read in via the `Read GIS WLL Points ==` command.

164 The 1d_nwk_C_check Height attribute is now correctly populated with the weir height for operating weirs.

165 Fixed bug that causes an operated pump to not switch off if using a pump curve. Constant flow rate pumps were not affected.

166 Additional output in the _O.csv (1D operational structure) output for operational pumps. The status column now provides a wider range of conditions, including: Off; Starting; Stopping; Dry; Below Soffit; Constant (flow) and Pump Curve (flow). "Below Soffit" indicates the intake water level has fallen below the soffit of the pump intake (invert plus diameter), in which case the pump is turned off (this behaviour is as previously applied, but there was no Status description indicating this behaviour).

167 Fixed bug that caused a negative constant value after an operand giving an ERROR 1513. For example, the line below previously would result in an ERROR.

```
Pump_Start == -0.5
```

168 For a pump, the upstream (intake) and downstream (outlet) inverts were not correctly shown in some output such as the .eof file. This has been corrected.