



## TUFLOW 2013-12 Release Notes

We are pleased to announce the 2013-12 release. Notes on the changes, new features and enhancements to TUFLOW, including getting started information for the TUFLOW GPU Module are provided within this document.

### Document Updates:

**June 12, 2015: Updated for Build 2013-12-AE. Primary changes to the document are highlighted in light grey.**

**November 13, 2014: Updated for Build 2013-12-AD. Primary changes to the document are highlighted in light red.**

**June 2, 2014: Updated for Build 2013-12-AC. Primary changes to the document are highlighted in light blue.**

**February 26, 2014. Updated for Build 2013-12-AB. Primary changes to the document are highlighted in light green.**

And yes, we are underway with updating the 2010-10 TUFLOW Manual to include the new features outlined in these release notes, plus the 2011-09 / 2012-05 release notes. We've also decided to give the manual a spring clean, remove/edit much of the legacy content for builds prior to 2010-10 and rework the structure. The manual will be available during 2014.

Should you require clarification or more detail on any of the points below, please email [support@tufLOW.com](mailto:support@tufLOW.com).

## Table of Contents

2013-12 Release Overview .....	3
Build 2013-12-AA Changes, New Features and Enhancements .....	4
New Map Output Formats .....	13
Output Zones .....	16
New 1D Structures.....	19
Operating Controls (.toc File).....	25
Build 2013-12-AB – New Features, Enhancements and Bug Fixes .....	32
Build 2013-12-AC – New Features, Enhancements and Bug Fixes .....	38
Build 2013-12-AD – New Features, Enhancements and Bug Fixes .....	42
Build 2013-12-AE – Bug Fixes .....	47
TUFLOW GPU Description.....	49
Introduction .....	49
Conservativeness.....	49
Stability and Adaptive Timestepping.....	50
Will the Results be the Same as TUFLOW “Classic”?.....	51
Compatible Graphics Cards .....	52
Compute Capability.....	55
Updating NVIDIA Drivers.....	55
Running TUFLOW GPU .....	56
TUFLOW Control File Commands.....	56
Cell Codes.....	57
Topography .....	57
Bed Resistance .....	57
Initial Water Levels.....	57
Boundary Conditions.....	58
Infiltration.....	59
Output.....	59
Time-Series Output (2d_po) .....	59
Excluded TUFLOW Functionality.....	59
Single and Double Precision .....	59
Troubleshooting .....	60
FAQ.....	60

## 2013-12 Release Overview

The 2013-12 release includes some major new features and range of enhancements. They include:

- GPU Module allowing 2D models of hundreds of millions of cells to be run hundreds of times faster.
- New map output formats including direct writing of 12D, waterRIDE, Blue Kenue and GIS Grids.
- New triangular mesh format for SMS .dat and .xmdf formats that includes the results at the cell centres.
- Ability to assign different map output settings to different output formats.
- Output Zones that allow you to produce map output in different formats, different time resolutions, different data formats, etc for different regions within your model.
- Much improved interpolation/extrapolation to the 2D cell corners at the wet/dry front that resolves issues associated with tracking maximums and inconsistencies between events of different probability.
- New .csv time-series format for our GIS based time-series results viewer.
- New 1D structures including pumps, sluice gates, radial gates, Q (stage-discharge) channels, spillways and eight types of weirs.
- Operational controls of gated rectangular culverts, pumps, sluice and radial gates, Q channels and gated spillways via simple logical scripts.
- Direct reading of land .xml TINs that can be exported from software such as AutoDesk's Civil3D.
- SA inflows via GIS streamline layer(s).
- New \_DEM\_Z ASCII grid check layer of your final model ground/bathymetric elevations.
- Control of which GIS check layers are output, and output of the large check layers (DEM\_Z, 2s\_zpt, 2d\_grd, 2d\_uvpt) for output zone(s).
- User defined variables in your control files.
- You can now use "If Event ==" in the same way you can use "If Scenario ==".
- .qgs project file that allows you to open all input and output (check) layers in QGIS.
- Specification of a road crossfall slope to improve the capture of water at pit inlets.
- \_mmQ GIS layer has two additional attributes for the drop and slope in peak water level along channels.
- 2D variable Z shapes can now be triggered based on 1D water level(s).
- New folders are now automatically created (no dialog query prompt anymore).
- TUFLOW Override files that allow you to specify computer dependent .tcf and .ecf commands.
- Issues relating to 2D PO flow lines slowing down simulations resolved.

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 (using Data, Data Calculator... in SMS and subtracting the two maximum h data sets is an easy way to do this). If you have any queries on the comparison outcomes, please email details and the .tlf files to [support@tuflow.com](mailto:support@tuflow.com).

## Build 2013-12-AA Changes, New Features and Enhancements

The changes, new features and enhancements in Build 2013-12-AA are:

- 1 There have been no changes to the 2D solver from the 2012-05 release. Similarly, there have been no changes to the 1D solution except that the 1D structure code has been reworked to accommodate the new 1D structures in the 2013-12 release. There are, however, some changes to the default settings as listed below. Backward compatibility to the 2012-05 default settings can be achieved in the usual manner by using “**Defaults == PRE 2013**”, noting that the changes below do not apply if “Defaults == PRE 2013” is set.
  - (a) A more advanced approach to extrapolating values to the 2D cell corners is the default for this release. This will cause changes in results at the cell corners – see the discussion at Item 5.
  - (b) The default for the 2013-12 release is to track and output the maximums every timestep. To not track the maximums, “Maximums and Minimums == OFF” will need to be specified.
  - (c) For W (weir) channels, “Weir Approach == METHOD A” has been reinstated as the default setting for the 2013-12 release. METHOD A utilises the Bradley submergence approach, which is after further analysis and feedback from users a more preferred approach. Note: this change does not apply to the new weirs, which have a more detailed approach to submergence and flow calculations. “Defaults == PRE 2013” will reset “Weir Approach ==” to METHOD B.
  - (d) “Write Check Files ==” in the .tcf file will now automatically also write the 1D check files. There is no need to specify “Write Check Files ==” in the .ecf file unless a different folder path for the files is desired.
  - (e) Output intervals for map and time-series data are now required to be entered. This is because the default values for these intervals is the computational timestep, and by not specifying values can cause excessive amounts of memory to be allocated, sometimes causing undesirable results!
  - (f) The substantially faster approach for 2d\_po flow lines is the default for the 2013-12 release. Should any issues arise with this change, you can revert back to the old approach using “PO Approach == METHOD A”.
  - (g) A slightly modified .csv format for 1D output is the default for the 2013-12 release.
  - (h) To use the new 1D structures described in these release notes, do not set “Defaults == PRE 2013”.
  - (i) A few WARNINGS have been changed to ERRORS. If “Defaults == PRE 2013” is set, they will remain as WARNINGS.
- 2 **New GPU Module:** For details on TUFLOW’s GPU 2D solver refer to the section [TUFLOW GPU Description](#) starting Page 49.
- 3 **New Map Output Formats**
  - (a) See the section on [New Map Output Formats](#) on the following pages for details on the new triangular mesh output option that utilises the 2D cell centre values, and for direct outputting in native formats for Blue Kenue, WaterRIDE, GIS Grids and 12D.
  - (b) Also note that different start/end times and intervals can be defined for different output formats by including the output format extension on the left of the command. The following commands can be

made output format specific by setting <format> to one of DAT | XMDF | WRB | T3 | GRID. For Build 2013-12-AC, instead of GRID, ASC and/or FLT can be specified. See Item 69.

**<format> Start Map Output == <time\_hr>**

**<format> End Map Output == <time\_hr>**

**<format> Map Output Interval == <interval\_sec>**

**<format> Map Output Data Types == <list\_of\_types>**

For example, to set the Map Output Interval for XMDF output to 6 minutes use:

**XMDF Map Output Interval == 360**

#### 4 New Output Zones Feature

- (a) The new Output Zones feature allows users to generate map and check output for sub-regions of models. Different output zones can have different output formats, start/end times, frequency of output, and output data types. Any number of output zones can be defined for a model, and all, a selection or none of the output zones can be activated when the model is run.
- (b) Output zones are very useful, particularly for large models, where higher time resolution map output is required for a part of a model to create an animation or to examine more closely the time varying changes. For a problematic section of model that requires closer examination, generate output on a fine time interval during the period in question. Another benefit is simply reduce the size of the output files by only outputting where information is required, and disabling the output for the whole model using "Map Output Entire Model == OFF".
- (c) For more detail see the section on [Output Zones](#).

#### 5 New interpolation/extrapolation methods to the 2D cell corners

- (a) There have been issues with tracking the maximum hazard and other depth related outputs, particularly on steep slopes, at the wet/dry interface where the depth extrapolated to the cell corners was exaggerated as it was extrapolated horizontally. On steep slopes this can cause high depths, and therefore, an exaggeratedly high hazard to be recorded as the maximum. Peak flood levels and depths are not so affected by this issue as the peak flood level is tracked at the cell centres then extrapolated to the corners when the maximums are output (hazards are tracked at the cell corners).
- (b) The other consequence of the pre 2013 approach to extrapolating to cell corners is that in some situations a higher probability flood can have a slightly higher peak flood level than a lower probability flood in the vicinity of the wet/dry interface.
- (c) For the 2013-12 release these issues are resolved with an improved, smarter, extrapolation of values to the cell corners at the wet/dry front. Two new methods are available, however, it is recommended that METHOD C be used as it is simpler and extensive testing has indicated it resolves the above issues. METHOD B largely resolves the above issues and has the added advantage that the effect of thin breaklines is better taken into account whilst they are dry or upstream controlled flow occurs.
- (d) To use the pre 2013 approach specify "Map Output Corner Interpolation == METHOD A" (this is invoked if "Defaults == PRE 2013" is specified). The new approach, which is the default for the 2013-12 release, is "Map Output Corner Interpolation == METHOD C". More details on the approach will be provided in the updated TUFLOW manual.
- (e) For model results where the water has risen upwards (eg. river flooding), the new approach should cause no significant changes in results (ie. fractions of a mm). Where the flow is downwards over

steep slopes, some changes in results at cell corners will occur, but usually only slightly although maximum hazard values on very steep slopes may experience a more significant change.

## 6 New 1D Structures

- (a) The new 1D structures available are
  - (i) Pumps
  - (ii) User defined stage-discharge (Q) channels.
  - (iii) Sluice gates
  - (iv) Radial gates
  - (v) Spillways with or without gates.
  - (vi) Weirs (Broad-crested, Crump weir, User-defined, Ogee, Rectangular, Trapezoidal or Cippoletti, V-notch, and the original W weir with options to customise). Other weirs such as side-spill weir and alternative methods from the literature are planned.
- (b) For information on the new structures, see the section [New 1D Structures](#).

**7 Operational Controls of Structures:** Gated rectangular culverts, pumps, sluice and radial gates and gated spillways can be operated using simple to complex logical scripts. See the section [Operating Controls \(.toc File\)](#) for details.

**8 “Read TIN Zpts ==” now recognises land .xml TINs** that are readily exported from 3D surface TINning software such as AutoDesk’s Civil3D. The format of the TIN file is determined by the file extension, so ensure the extension is “.xml”.

## 9 Streamlines (“Read GIS Streams ==”)

- (a) Streamlines allow the user to apply SA inflows along the waterways rather than to the lowest cell (when all cells are dry within the SA region). If streamlines have been specified using the .tbc “Read GIS Streams ==” command, SA inflows are distributed along the 2D cells selected by the stream lines within each SA region.
- (b) “Read GIS Streams ==” can be used one or more times in the .tbc file to define streamline cells. Streamlines are polyline or line objects, usually representing the path of the waterways. One attribute is required being the Stream Order as an integer. GIS and other software have the ability to generate streamlines from DEMs, and usually assign a stream order to each stream polyline. If needed, rearrange (or copy) the attributes so that the first attribute is the stream order one.
- (c) Note, only streams with a stream order greater than zero (0) are used by TUFLOW. Therefore, streams that are not to be used for applying SA inflows can be assigned a stream order of 0 or deleted from the layer.
- (d) By default, any wet cells that are not streamline cells are also included in the distribution of the SA inflow. Options for controlling streamline inflows are:
  - (i) “Read GIS SA STREAM ONLY ==” will only apply the SA inflows to the streamline cells, ie. no non-streamline wet cells in the SA region will receive an inflow. Note this is the default approach adopted by the GPU solver.

- (ii) “Read GIS SA STREAM IGNORE ==” will ignore all streamline cells within the SA region(s) and distribute the inflows using the standard approach for SA inflows, ie. lowest cell if all wet, otherwise distributed over the wet cells.
- (iii) If “SA Proportion to Depth == ON” (default setting), the distribution of SA inflows according to depth only applies to wet non-streamline cells. The approach adopted is as follows:
  - The total inflow assigned to streamline cells within a SA region is proportioned according to the number of streamline cells versus wet non-streamline cells.
  - The distribution of the inflow allocated to streamline cells is weighted equally between cells.
  - The distribution of the inflow allocated to wet non-streamline cells is weighted according to the depth of water in the cells.
- (e) The 2d\_sac\_check layer will show those cells selected as streamline cells.

## 10 \_TUFLOW\_OVERRIDE files

- (a) The override files allow the user to apply .tcf commands after TUFLOW has finished processing the .tcf file. This can be useful where, for example, you wish to change the output drive using “Output Drive ==” for a set of runs on a different computer.
- (b) Two files, if either or both are present, are processed after processing of the .tcf file is finished. The first file that TUFLOW checks for in the same folder as the .tcf file must be named “\_TUFLOW\_Override.tcf”. If this file exists, .tcf commands within this file are processed after all other .tcf commands.
- (c) The second file TUFLOW looks for must be named “\_TUFLOW\_Override\_<computer\_name>.tcf” where <computer\_name> is the name of the computer running the simulation (this appears near the top of the .tcf file if you’re unsure of the computer’s name). If this file exists TUFLOW will process any commands within it after any commands from the “\_TUFLOW\_Override.tcf” file (if it exists).
  - (i) An override file specific to a particular computer can be particularly useful where, for example, different output drives or results folders, are to be used for runs using different computers. This will allow you to run TUFLOW simulations on different computers without having to change the .tcf file.
  - (ii) For example, if a run is started on one machine that only has a C drive, the output can be set just for that machine to be directed to the C drive using “Output Drive == C”.
  - (iii) Another example is if using the GPU solver and one machine only has a single GPU, while another has four GPUs, for the machine with four GPUs the command “GPU Device IDs == 0, 1, 2, 3” can be specified in the override file specific to that machine.
- (d) Nearly all .tcf commands can be placed within the override files with the exception of Read GIS and some other similar commands that involve processing of data layers. It is recommended that the override files are only used for global changes to settings such as relocating the results folder.

## 11 “Pit Default Road Crossfall == <>” ....

- (a) New .ecf command “Pit Default Road Crossfall == <slope>” that increases the depth at Q pits based on the height of an imaginary triangle of the road cross-section with a crossfall slope of <slope>. <slope> is Vertical/Horizontal as a fraction (not percentage).

- (b) The imaginary triangle has the same area as the vertical flow area in the 2D cell the pit is connected to (ie. the triangle's area is the depth in the 2D cell times the width of the cell). Once the horizontal width of the triangle is greater than the width of the 2D cell, the formula changes to give an equivalent area based on a trapezoid consisting of the triangle plus a rectangle for the remaining area in excess of the triangle.
- (c) Improves the ability to capture water from 2D cells into pits feeding the pipe network. Especially for larger cell sizes, the shallow depth that can occur in the cell can be unrepresentative of the depth at the entrance to the pit, therefore restricting the amount of flow entering the pit. The larger depth using an equivalent triangle will push more water into the pit providing a more realistic representation of the pit's capture.

## 12 DEM\_Z Check Grid

- (a) The one that many of you have been waiting for! When writing check files, TUFLOW now produces a ASC or FLT DEM of the final ground/bathymetry elevations, including those from any 1D WLL mesh. The file is given a DEM\_Z extension, and can be readily opened by most GIS and other GUIs. The default size of the grid cells is half the smallest 2D cell size. This can be changed using "Grid Output Cell Size ==". To exclude writing this file, include "DEM\_Z" in the "Write Check Files EXCLUDE ==" list. The "Grid Format ==" command introduced in Build 2013-12-AB controls whether ASC or FLT format is used.

## 13 New options for controlling which check layers are written.

- (a) "Write Check Files ==" now has the additional format as follows:  
Write Check Files [ {ALL} | NONE | EXCLUDE | INCLUDE ] == <prefix\_list>
- (b) If EXCLUDE or INCLUDE occur to the left of "=", the entry to the right of the "=" is a space delimited list of file prefixes used to exclude or include GIS layers from being written. Prefixes must be the same as those used by TUFLOW. For example, "zpt" would apply to the zpt\_check layer. To exclude/include more than one layer ensure there are spaces between the prefixes. If EXCLUDE or INCLUDE occurs more than once, the latter occurrence prevails.
- (c) The ALL and NONE options require no prefix list. Specification of the ALL or NONE options will nullify any prior occurrence of an EXCLUDE or INCLUDE list; this is useful if you wish to write no or all check files for one particular run – simply add Write Check Files ALL (or NONE) to the end of the .tcf file.
- (d) Provided "Defaults == PRE 2013" is not specified, specification of the "Write Check Files ==" command will also write the 1D check files, so there is no need to specify "Write Check Files ==" in the .ecf unless a different folder path or check file prefix is desired for the 1D check layers.
- (e) Examples are:  
Write Check Files EXCLUDE == zpt uvpt ! excludes writing of the zpt\_check and uvpt\_check layers  
Write Check Files INCLUDE == dem\_Z ! will only write the Grid of the model's ground elevations

- 14 **Two additional attributes have been added to the \_mmQ GIS layer** containing the difference in maximum water level drop (dHmax) and the slope as a percentage of the water surface along the channel (pSmax). Useful for quickly identifying any untoward behaviour along 1D networks by viewing/searching for any negative (adverse) slopes or large unexpected changes in flood level.



## 15 New “Set Variable ==” command (also see Item 56)

- (a) In any TUFLOW control file use “Set Variable <name> == <value>” to define a variable’s name and value. Wherever you want to refer to the variable, the variable’s name must be bounded by “<<” and “>>” characters. Note, that when using “Set Variable ==” do not use <<...>> to bound the variable’s name. The variable’s <value> may include one or more events and/or scenarios if you so choose.
- (b) Variables may be set to any number or text and can be referred to as often as needed in any control file and in other files such as the .tmf and new .toc files. At present you can’t use variables in the bc\_dbase.csv files, but future builds may offer this feature.
- (c) For example the lines below give some idea of how you could use variables (also see Item 57) that explains how scenarios and events are automatically set as available variables. The commands below set the model’s cell size, timestep and output interval, and sets the folders for outputting check files and results according to Scenario 1 which is the model’s grid resolution (cell size), ie. one of “0.5m”, “1.0m” or “2.0m”.

### **.tcf file entries:**

```

If Scenario == 0.5m
  Set Variable 2D_CELL_SIZE == 0.5
  Set Variable 2D_Timestep == 0.25
  Set Variable LogInt == 60
Else If Scenario == 1.0m
  Set Variable 2D_CELL_SIZE == 1.
  Set Variable 2D_Timestep == 0.5
  Set Variable LogInt == 60
Else If Scenario == 2.0m
  Set Variable 2D_CELL_SIZE == 2.
  Set Variable 2D_Timestep == 1.0
  Set Variable LogInt == 30
End If

! Set times
Start Time == 0
End Time == 0.5
Timestep == <<2D_Timestep>>

Write Check Files == ..\check\<<~s1~>>\
Output Folder == ..\results\<<~s1~>>\
Screen/Log Display Interval == <<LogInt>>

```

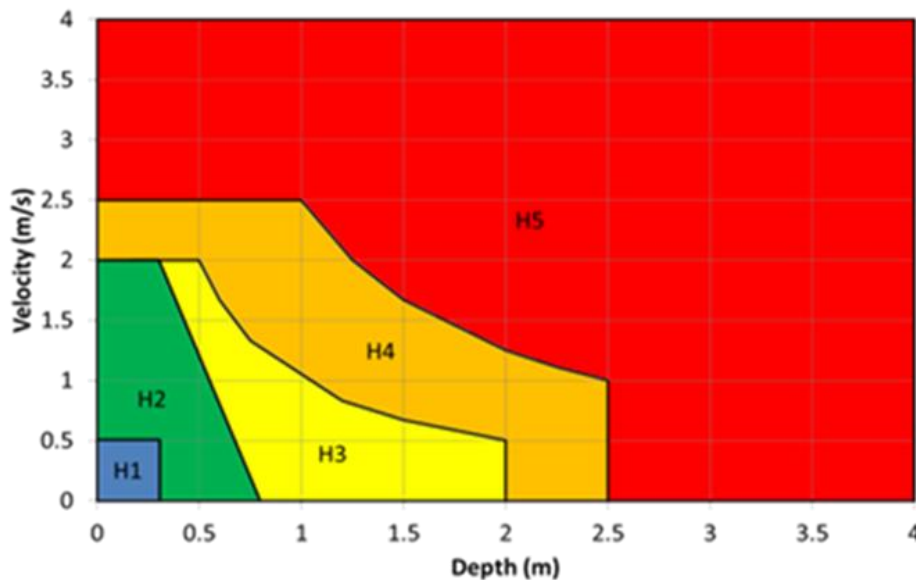
### **.tgc file entry:**

```

Cell Size == <<2D_CELL_SIZE>>

```

- 16 Can now use “If Event ==” / “End If” logic blocks in the same manner as “If Scenario ==”.
- 17 “Time Series Output Interval ==” or “Output Interval (s) ==” must now be specified if any 1D or 2D plot output exists. Similarly, “Map Output Interval ==” must also now be specified. If either of these cases occur an ERROR 0046 occurs. If “Defaults == PRE 2013” is used, a WARNING 0046 is issued and the simulation will continue to proceed (and will use the computational timestep for the output interval). As of Build 2013-12-AC if “Map Output Interval == 0” is specified time based map output is switched off – see Item 6969(c).
- 18 “Time Series Output Format == [ PRE 2013 | {2013} ]”
  - (a) The default for the 2013-12 release is to output 1D .csv files to a csv folder in a slightly new format that works with our new GIS 1D chart viewer. The GIS 1D viewer will be made available early 2014, and works as a standalone utility, or can be integrated with GIS. A beta version is available for those interested in testing/contributing (please email [support@tuflow.com](mailto:support@tuflow.com)).
  - (b) 2D PO .csv output is not yet incorporated, and continues to be output in the same format as prior releases.
  - (c) To output in the old formats, specify “Time Series Output Format == PRE 2013” in the .tcf file.
- 19 New hazard output ZMBRC, used by Moreton Bay Regional Council for their flood hazard categories, can now be used and is described below.



Category Name	Suitability/Description	Hydraulic Criteria
H <sub>1</sub>	Hydraulically suitable for parked or moving cars	$v < 0.5\text{m/s}$ and $d < 0.3\text{m}$
H <sub>2</sub>	Hydraulically suitable for parked or moving heavy vehicles and wading by able-bodied adults	$v < 2\text{m/s}$ , $d < 0.8\text{m}$ and $v < 3.2 - 4*d$
H <sub>3</sub>	Hydraulically suitable for light construction (e.g. Timber frame and brick veneer)	$v < 2\text{m/s}$ , $d < 2\text{m}$ and $v*d < 1$

Category Name	Suitability/Description	Hydraulic Criteria
H <sub>4</sub>	Hydraulically suitable for heavy construction (e.g. steel frame and reinforced concrete)	$v < 2.5\text{m/s}$ , $d < 2.5\text{m}$ and $v*d < 2.5$
H <sub>5</sub>	Generally unsuitable	$v > 2.5\text{m/s}$ , $d > 2.5\text{m}$ and $v*d > 2.5$

- 20** Similar to the .wor file, **TUFLOW now also writes a .qgs file**. Opening this file in QGIS will open all GIS input and output check layers (.mif and/or .shp). Note that the visibilities of the output layers are unchecked so that the display time is quick. If using .shp files for the check layers use Build 2013-12-AC or later.
- 21** **1D node water levels can now be used to trigger a 2D variable Z-Shape**. In the 2d\_vzsh layer the trigger point must be snapped to the 1D node or channel end. For the Shape\_Options attribute the entry must be "Trigger 1D" (not just "Trigger" – "1D" needs to occur within the attribute so TUFLOW knows to search for a snapped 1D node). Use the 2d\_vzsh\_zpt\_check layer to cross-check that the 1D node trigger has been correctly selected.
- 22** **"Read GIS SA RF =="** and **"Read GIS RF =="** now treat negative rainfall values as a loss (eg. evaporation). No IL/CL values that apply to positive rainfall are applied to negative values. Previously negative values were treated as zero.
- 23** **The approach for processing 2d\_po flow lines has been reworked and will now not slow down a simulation** if numerous 2d\_po flow lines exist as was occurring with prior releases. To use the original approach "PO Approach == METHOD B" can be specified in the .tcf file, however, this should only be necessary in the event of a problem arising in which case please email [support@tuflow.com](mailto:support@tuflow.com).
- 24** Water level boundaries that do not incrementally change along the boundary line by using the f, d, td, a, b attributes are now processed much faster.
- 25** **Non-existent folders are now created automatically** (equivalent to using the -acf run time option). If you would prefer to have the Create folder query dialog to appear, you can specify the -qcf run time option (stands for query create folders). Therefore, if you're expecting the Create Folder query dialog to appear at it doesn't, this is why!
- 26** **-pu<id> run time option** to select which processing units to direct the simulation towards. At present this only applies to the GPU solver where a simulation is to be directed to a particular GPU card or cards. -pu must be specified once for each device. For example, to direct the simulation to GPU devices 0 and 2, specify -pu0 -pu2.
- 27** For **BW or IW channels that automatically insert a weir** over the B or I channel, the weir width if not manually specified using the EN1 attribute is now based on the maximum width of the B or I channel, rather than the width at the channel's invert. Previously, it was problematic if the B or I channel represented an arched bridge or egg shaped culvert as the width at the top would be zero! Backward compatibility is available by specifying "Structure Routines == ORIGINAL".
- 28** **BSS and SP map output may change slightly** due to now taking the surrounding average Manning's n value at cell corners (previously the Manning's n value at the cell side below the cell corner was used. MB1 and MB2 values at cell corners are now the average of the 4 surrounding cell centres (previously the value at the lower left cell centre was used). For cell centre values use the new triangular mesh output for 2D domains.

- 29** **Structure output to the .eof file is now in alphabetical order** according to the channel ID (as per other output to the .eof file).
- 30** “**ERROR 1276** - Could not find any channel ends for the pit within the Pit Search Distance” is now included in the \_messages layer pointing to the pit channel in question. Previously the message was only output to the .tif file.
- 31** “**WARNING 2320** - GIS layer for "Read GIS Code ==" command might be a 2d\_bc layer” was treated as an ERROR causing TUFLOW to exit – this has been rectified.
- 32** A bug that could incorrectly apply “**Global Rainfall ==**” values has been corrected.
- 33** Horizontal (flat) channels would always be flagged as having adverse flow using the “A” flag in the .eof file – this has been corrected.

## New Map Output Formats

34 Several new map output formats are now available in addition to the SMS .dat and .xmdf formats. A new output resolution is also available for the SMS formats that incorporates output at the cell centres. The new options are discussed below.

- (a) **Map Output Formats == DAT | X MDF | WRB | GRID | T3 | TMO | SMS TRIANGLES | HIGH RES**  
Sets the map output format(s). Some or all of the formats can be specified including the new formats .wrb (waterRIDE), .t3s/.t3v (BlueKenue), .tmo (used by 12D) and GIS grids. The SMS TRIANGLES option outputs 2D cells as four triangles so as to include the cell centre values, which is the approach for the approach adopted for WaterRIDE .wrb and BlueKenue .t3 formats. The only restrictions are that the SMS HIGH RES option cannot be specified in combination with any of the other formats. For Build 2013-12-AC, instead of GRID, ASC and/or FLT can be specified. See Item 69.

(b) **New Cell Centred Triangular Mesh Option**

- (i) A new triangular mesh output format is available that incorporates output at the cell centres, so the cell centre elevation and the exact water level calculated by TUFLOW can be viewed in a GUI. 2D cells are represented as four triangles with a common vertex at the cell centre, giving a higher resolution spatial output than just using the corners of the cells.
- (ii) Note that the number of nodes in the mesh for the 2D domains will increase by 20% and the number of elements by a factor of 4 compared with using square elements, therefore the output file sizes will be larger and the viewing times possibly slower. Any triangular elements from 1D WLLs are not affected and remain the same.
- (iii) To utilise this format using the SMS .dat and .xmdf formats, specify “SMS TRIANGLES” anywhere in entry for “Map Output Formats ==”. Note, “X MDF” will also be required if using the .xmdf format, and “DAT” if any other format is specified (.dat format remains the default output format if no other format is specified).
- (iv) The waterRIDE (WRB) and Blue Kenue (T3) output formats described below only use this format as these products don’t support quadrilateral elements in a mesh.

(c) **waterRIDE** by WorleyParsons is commercial software for visualising and post-processing hydraulic modelling results. For more information [click here](#).

- (i) TUFLOW waterRIDE output for this release utilises the .wrb format. The inclusion of the waterRIDE grid based formats is being incorporated and is planned as an update to the 2013-12 release.
- (ii) 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. The .wrb format is restricted to these data types, so for waterRIDE output other data types specified using “Map Output Data Types ==” are ignored. The only “Map Output Data Types ==” data types recognised (besides h and V) are Z0 (VxD) and one hazard category (the first one is used if more than one is specified). If Z0 and/or a hazard category are not specified for WRB output, waterRIDE can optionally post-process these values. Other data types such as depth are also post-processed by waterRIDE.
- (iii) If maximums are tracked these are also added to the .wrb file for the data types mentioned above. 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 .wrp file which only occur every map output interval.

(iv) As mentioned above, the .wrp output utilises the new triangular mesh output for 2D domains.

(v) If using TUFLOW double precision (DP), Build 2013-12-AC or later will need to be used for .wrp output – see Item 69(d).

(d) **Blue Kenue** from the National Research Council Canada is free software for visualising and animating time varying hydraulic model map output. For more information [click here](#).

**Note: Use Build 2013-12-AB or later for Blue Kenue output.**

(i) TUFLOW Blue Kenue output utilises the .t3s and .t3v formats. .t3s contains fixed or temporal scalar data and .t3v contains vector data.

(ii) Five files are produced per simulation as follows:

- .Z.t3s: Contains the ground elevations/bathymetry of the model. This file is the ASCII form of the .t3s format so can be viewed using a text editor.
- .T.t3s: Contains the temporal scalar map output for all data types specified using the binary format of the .t3s format. Note for Builds 2013-12-AA and AB the file extension was set to .Ts.t3s.
- .T.t3v: Contains the temporal vector map output. At present this format can only support one data type, so if, for example, V and q data types have been specified, only V will be included (V and q as scalar output will appear in the .Ts.t3s file). Discussions with Blue Kenue developers are underway to allow this format to support multiple data types. The file is a binary format. Note for Builds 2013-12-AA and AB the file extension was set to .Tv.t3v.
- .M.t3s: If maximums are being tracked, this file contains the maximums for scalar data types in binary format. Note for Builds 2013-12-AA and AB the file extension was set to .Ms.t3s.
- .M.t3v: If maximums are being tracked, this file contains the maximums for the vector data type in binary format. The comment above for .Tv.t3v files also applies to this file. Note for Builds 2013-12-AA and AB the file extension was set to .Mv.t3v.

(iii) All of the above files, including the binary files, have an ASCII header which is useful as meta data on the file can be read by a text editor. If the editor opens the file in hex format, switch the editor to ASCII (text) viewing format to view the header information.

(iv) Blue Kenue only supports triangular meshes, so all Blue Kenue output at present utilises the new triangular mesh output described above.

#### (e) TMO 12D Map Output

(i) The .tmo format is utilised by 12D Solutions for their TUFLOW interface. If using 12D to display/view results this format will need to be specified using “Map Output Format ==”

#### (f) GIS Grids

(i) The simple GRID “Map Output Format ==” option provided in the 2012-05 release has been replaced for the 2013-12 release. The new GRID approach handles rotated 2D domains, multiple 2D domains and 1D WLL meshes. All “Map Output Data Types ==” are supported, and the resolution of the output grids can be set by the user (the default is half the smallest 2D cell size).

The new approach utilises a similar method to that used by TUFLOW\_to\_GIS using the -asc or -flt option.

- (ii) TUFLOW pre-processes and stores in memory the interpolation factors at each GIS Grid cell. This significantly reduces the computational time to write each grid so will appear much faster than TUFLOW\_to\_GIS. This approach does, however, consume more RAM.
- (iii) ~~All grids are presently written as ASCII grids, but there are plans to extend to the .flt binary format and other formats.~~ .flt grids are now also supported as of Build 2013-12-AB – see Item 58.
- (iv) **“Grid Output Cell Size == <grid\_cell\_size\_m\_ft>”** sets the cell resolution of all grid outputs with the default being half the smallest 2D cell size. At present only one output grid resolution is possible, so the last occurrence of the command will prevail. It is planned to allow different resolutions to be specified for different map output formats and different output zones. Also note that a DEM of the final Zpts is now automatically written using this resolution if writing check files unless it is excluded using “Write Check Files Exclude == DEM\_Z”.
- (v) **“Maximums and Minimums Only for GRID == [ ON | OFF ]”** can be used to control whether only grids of the maximums are required.
- (vi) For SMS users, note that a nice feature of v11.1 or later is the ability drag and drop ASC or FLT grids into SMS for reviewing alongside the standard mesh based output in .dat or .xmdf files.

#### (g) GIS Layers

- (i) Direct outputting of GIS layers of results utilising a similar approach to TUFLOW\_to\_GIS is close to being finished and will be provided as an update in 2014.
- (h) Different start/end times and intervals can be defined for different output formats by including the output format extension on the left of the command. The following commands can be made output format specific by setting <format> to one of DAT | XMDF | WRB | T3 | GRID. For Build 2013-12-AC, instead of GRID, ASC and/or FLT can be specified. See Item 69.

**<format> Start Map Output == <time\_hr>**

**<format> End Map Output == <time\_hr>**

**<format> Map Output Interval == <interval\_sec>**

**<format> Map Output Data Types == <list\_of\_types>**

For example, to set the Map Output Interval for XMDF output to 6 minutes use:

**XMDF Map Output Interval == 360**

## Output Zones

### 35 Output Zones Overview

- (a) Similar to using events, any number of Output Zones can be defined. The “Model Output Zones ==” command is used to select which output zones are to be used.
- (b) Output zones currently only apply to TUFLOW’s map outputs (ie. does not apply to time series and other outputs).
- (c) Unlike events which use an event (.tef) file, there is no requirement to have a separate file containing the output zone definitions, however, if there are numerous output zones, it would be good practice to place these in one or more separate files and use the “Read File ==” to reference these file(s) so as to keep the size of the .tcf file minimal.

### 36 Define Output Zone ==

Each output zone is defined using a definition block as follows:

**Define Output Zone == <oz\_name>**

.....

**End Define**

The following commands can be used within an output zone definition. With the exception of “Read GIS Output Zone ==”, all commands are optional.

Note: If one of the commands below is not specified within the output zone’s definition, the latest setting of that command prior to the output zone’s definition, or the global default if the command has not been used prior to that location with the .tcf file, will be used. If, for example, all output is to be in the .xmdf format, only specify “Map Output Format == XMDF” once and prior to any output zone definitions.

- (a) **Read GIS Output Zone == <gis\_layer>**  
The GIS layer contains one or more polygons that define the regions to be output. The attributes of the layer are not used.
- (b) **Map Output Formats == DAT | XMDF | WRB | GRID | T3 | SMS TRIANGLES**  
Sets the zone’s map output format(s). Some or all of the formats can be specified including the new formats .wrb (waterRIDE), .t3s/.t3v (BlueKenue) and GIS grids. The SMS TRIANGLES option outputs 2D cells as four triangles so as to include the cell centre values, which is the approach for the approach adopted for WaterRIDE .wrb and BlueKenue .t3 formats. The only restrictions are that the SMS HIGH RES option cannot be specified in combination with any of the other formats, and the TMO format used by 12D is not yet available for output zones.

For Build 2013-12-AC, instead of GRID, ASC and/or FLT can be specified. See Item 69.

- (c) **Start Map Output == <time\_hr>**  
**End Map Output == <time\_hr>**  
**Map Output Interval == <interval\_sec>**  
**Map Output Data Types == <list\_of\_types>**  
Use the above commands to control the output zone’s default start, finish, interval and output data types.



Note: If a different setting of one the above commands is required for a specific map output format, then repeat the command(s) and include the output format to the left of the ==. For example, to set the Map Ouput Interval for GRID output to one hour use:

**GRID Map Output Interval == 3600**

(d) **Output Folder == ..\results\zones**

Sets the output zone's output folder if different to the default setting. Note that, by default, the map output for an output zone is placed in a sub-folder with the same name as the output zone.

(e) **Maximums and Minimums Only for GRID == ON**

Only outputs maximums and/or minimums if GRID output for the output zone is switched on.

(f) **Maximums and Minimums == [ ON | OFF | {ON MAXIMUMS ONLY} ]**

Sets whether maximums and/or minimums are to be written for the output zone. Note that for the 2013-12 release the new default for this command is ON MAXIMUMS ONLY. To suppress tracking of maximums set to OFF.

(g) **Map Cutoff Depth == <depth\_m\_ft>**

~~Sets the cutoff depth for the output zone.~~

(Apologies: The above command was documented as being available for different output zones. This was incorrect, the command is still a global command, ie. all output zones will use the same map cutoff depth if specified.)

(h) **Write Check Files [ INCLUDE | EXCLUDE | ALL | NONE ] == <prefix\_list>**

The large check layers (2d\_zpt, 2d\_grd, 2d\_uvpt, DEM\_Z) are also written for any active output zones if writing check files. The name of the output zone is included in the filename. The "Write Check Files == " command utilising the new Include/Exclude/All/None options can also be used to change the write check file settings for each output zone.

### 37 Example of Defining an Output Zone

The example below defines Output Zone “ZoneA”. In the example, a DEM\_Z check grid will be the only check layer written for the output zone, and Grid, WRB and XMDF outputs will be produced. The Grid output will consist of water level (h) and VxD (Z0) results, starting at time 0.5h and ending at 4h at an interval of 0.5h. The WRB and XMDF output files will contain water levels (h), velocities (V) and the new ZMBRC hazard categories starting at the simulation beginning and finishing at 6h at an interval of 6 minutes.

```
Define Output Zone == ZoneA
  Read GIS Output Zone == ..\model\mi\2d_oz_ZoneA.mif
  Write Check Files Include == DEM_Z
  Map Output Formats == GRID XMDF WRB
  End Map Output == 6.0
  Map Output Interval == 360
  GRID Start Map Output == 0.5
  GRID End Map Output == 4.0
  GRID Map Output Interval == 1800
  XMDF Map Output Data Types == h V ZMBRC
  GRID Map Output Data Types == h Z0
End Define
```

### 38 “Model Output Zones == <oz\_name\_1> | <oz\_name\_2> | .....

- (a) The .tcf “Model Output Zones ==” command controls which output zones are to be used. If this command is omitted, no output from output zones is written. Separate multiple output zones using a “|”. For example, to output from zones ZoneA and ZoneC specify:

```
Model Output Zones == ZoneA | ZoneC
```

### 39 Miscellaneous Output Zone Related Commands

- (a) **Map Output Entire Model == [ {ON} | OFF ]**

The default is to produce map output for the whole model irrespective of the number of output zones being written. Map output commands that occur outside Output Zone definitions apply to the entire model (ie. as is the case in previous releases). If map output for the entire model is not required specify “Map Output Entire Model == OFF”.

## New 1D Structures

40 **New 1D structures** in the 2013-12 release are described below. The new structures include:

- (a) Pumps (P)
- (b) User defined stage discharge (Q) channels
- (c) Radial gates (RG)
- (d) Sluice gates (SG)
- (e) Gated or un-gated spillways (SP)
- (f) A range of new weirs and the ability to adjust the weir equation parameters and manually define your own submergence curve. The available weir types are:
  - (i) W: Broad-crested weir unchanged from previous releases, except the default submergence approach has been reverted back to “Weir Approach == METHOD A” – see discussion below.
  - (ii) WB: Broad-crested weir
  - (iii) WC: Crump weir
  - (iv) WD: User-defined weir
  - (v) WO: Ogee-crested weir
  - (vi) WR: Rectangular weir
  - (vii) WT: Trapezoidal weir or Cippoletti weir
  - (viii) WV: V-notch weir
  - (ix) WW: Similar to the original W weir channel, but has more options

41 The 1d\_nwk Attribute changes/descriptions for the new structures are provided below for the attributes that are affected.

### (a) Type

- (i) Note: Up to 8 characters can now be used to define a channel type. If 4 characters are not sufficient, the Type attribute’s length can be increased using your GIS software from 4 to 8.
- (ii) Use the one or two letter acronym for the structure as described above. For example, a V-notch weir would be entered as “WV”, and a sluice gate would be entered as “SG”.
- (iii) For sluice and radial gates, if they are located on top of a weir the type of weir can be specified by adding one of the new weirs. For example, a sluice gate set on top of a rectangular weir would have a Type of “SG WR” (noting that the space is optional).
- (iv) “O” flag is required for structures that are to be operated using an operating control definition (see section on Operational Controls below). For example, an operated pump would have a Type attribute of “PO”.
- (v) “E” and “H” flags for RG, SG, SP and all new weirs set whether to use energy (E) or water surface (H) levels for the calculations. The default is to use energy (E), unless the global .ecf command “Structure Flow Levels == WATER” is used, in which case the default is to use water level (H). For example, if a broad-crested weir is to use water surface levels in the equations, specify WBH (a space can be used so “WB H” may be preferred for clarity).

**(b) n\_nF\_Cd (previously n\_or\_n\_F)**

- (i) **RG, SG, SP:** Discharge coefficient for the structure if using a fixed coefficient. If the value is less than or equal to zero, the default Cd value of 0.7 for RG, 0.6 for SG and 0.75 for SP is used.

**(c) US\_Invert and DS\_Invert**

- (i) **Q:** US\_Invert sets the level from which the upstream depth is to be calculated for interpolation into the depth-discharge curve. DS\_Invert not used other than it may need to be set to the invert of the joining channel downstream.
- (ii) **RG, SG:** The higher of US\_Invert and DS\_Invert is used to set the structure crest or invert.
- (iii) **SP:** US\_Invert sets the spillway crest and DS\_Invert sets the level of the gate seat (if SP is operated, ie. a SPO channel).
- (iv) **All weirs:** The higher of US\_Invert and DS\_Invert is used to set the structure crest or invert.
- (v) **WO:** For the Ogee Weir (WO), the absolute difference in height between the US\_Invert and DS\_Invert is used to set the height of the weir above its sill (usually denoted as P), which is used for recalculating the weir's discharge coefficient each timestep. If the US\_Invert and DS\_Invert are the same value the primary upstream channel bed will be used to set the value of P.

**(d) Form\_Loss**

- (i) **RG, SG:** If no weir is specified in the Type attribute, it is assumed that the gate seats on to the bottom of the channel. For this case the flow calculations where the gate is not surcharged uses the Form\_loss attribute to apply an energy loss to the structure to represent contraction/expansion losses.

**(e) pBlockage**

- (i) **P, RG, SG, SP:** The channel's dimensions are reduced as per the pBlockage value.
- (ii) **WB, WC, WD, WO, WR, WS:** The weir width is adjusted proportionally by the % blockage.
- (iii) **WT:** The base width of the weir is adjusted proportionally by the % blockage.
- (iv) **WV:** The V-notch angle is adjusted proportionally by the % blockage.

**(f) Inlet\_Type**

- (i) **Q:** For non-operated Q channels the name of depth-discharge curve in a depth discharge database (identical approach to inlet Q pits). A Q channel is distinguished from a Q pit, in that a Q channel is a polyline in the 1d\_nwk layer and a Q pit is a point object. A depth-discharge database is the same as a pit inlet database as used for Q pits, and the same database file can be used for both Q channels and Q pits. The new .ecf command "Depth Discharge Database == " and the existing command "Pit Inlet Database == " perform the same function. Note that the depth discharge curve is depth (not water elevation) and the depth is taken as the upstream water level less the Q channel's invert.
- (ii) **QO:** For operated Q channels the filename of a .csv file containing a flow matrix table in the same format as used for M channels. The vertical axis is the depth above the channel's invert and the horizontal axis is the percentage opening as shown in the example below. In the example, Column A contains the depth above invert values and Row 2 the % opening values. Note that the value in cell A2 is a flow multiplier. If this value is empty, negative or set to zero (0),

a multiplier of one (1) is used. The flow through the channel is interpolated from this table each timestep based on the structure's opening and upstream depth above invert.

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	<b>Opening (%)</b>												
2	<b>1</b>	<b>0</b>	<b>5</b>	<b>10</b>	<b>20</b>	<b>30</b>	<b>40</b>	<b>50</b>	<b>60</b>	<b>70</b>	<b>80</b>	<b>90</b>	<b>100</b>
3	<b>0.0</b>	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	<b>7.0</b>	0.00	2.90	5.87	10.97	16.14	19.90	23.64	26.82	29.24	32.06	33.90	35.36
5	<b>7.2</b>	0.00	2.91	5.89	11.00	16.18	19.94	23.69	26.88	29.30	32.14	33.98	35.45
6	<b>7.4</b>	0.00	2.92	5.90	11.02	16.22	19.98	23.74	26.94	29.37	32.21	34.06	35.53
7	<b>7.6</b>	0.00	2.92	5.92	11.05	16.26	20.03	23.79	27.00	29.43	32.28	34.14	35.61
8	<b>7.8</b>	0.00	2.93	5.93	11.07	16.30	20.07	23.84	27.06	29.50	32.36	34.22	35.69
9	<b>8.0</b>	0.00	2.94	5.94	11.09	16.33	20.11	23.89	27.12	29.56	32.43	34.30	35.78
10	<b>8.2</b>	0.00	2.94	5.96	11.12	16.37	20.15	23.94	27.18	29.63	32.51	34.37	35.86
11	<b>8.3</b>	0.00	2.95	5.97	11.13	16.39	20.18	23.96	27.21	29.66	32.54	34.41	35.90
12	<b>8.4</b>	0.00	2.95	5.97	11.14	16.41	20.20	23.99	27.24	29.69	32.58	34.45	35.94
13	<b>8.6</b>	0.00	2.96	5.99	11.17	16.45	20.24	24.04	27.30	29.75	32.65	34.53	36.02
14	<b>8.8</b>	0.00	2.96	6.00	11.19	16.48	20.28	24.09	27.36	29.82	32.72	34.61	36.10
15	<b>9.0</b>	0.00	2.97	6.01	11.21	16.52	20.33	24.14	27.41	29.88	32.80	34.68	36.18

#### (g) Width\_or\_Dia

- (i) **P**: The diameter of the pump offtake.
- (ii) **Q**: A multiplier applied to the flow interpolated from the depth-discharge curve. Particularly useful if the discharge values are per unit width (ie.  $m^2/s$  or  $ft^2/s$ ). If set to zero (0), the multiplier is set to one (1).
- (iii) **QO**: Used to determine the flow area at different stages by multiplying by the upstream depth. The area is used to determine the velocity that is output, otherwise, it does not play a role in the hydraulic calculations.
- (iv) **RG, SG, SP**: The width of the gate/spillway. The flow area is assumed to be rectangular in shape.
- (v) **WW**: The width of the weir unless a cross-section is assigned to the weir using a 1d\_xs line.
- (vi) **WB, WC, WD, WO, WR, WS**: The width of the weir. The weir is assumed to be rectangular in shape.
- (vii) **WT**: The base width of the weir.
- (viii) **WV**: Angle of the V-notch in degrees. Must be between 20° and 100°.

#### (h) Height\_or\_WF

- (i) **P**: Can be used to set the pump flow capacity for operated pumps (ie. a PO channel). The pump capacity can also be set and changed during the simulation using the operating control commands. Useful where the operating control definition is generic, ie. pump control is used for more than one pump of different flow capacities.

- (ii) **RG, SG:** The vertical height of the gate in its fully open position. If an underlying weir is specified, the weir is assigned a coefficient adjustment factor of 1.0, which cannot be modified using this attribute.
- (iii) **SPO:** For an operated (gated) spillway (SPO), sets the vertical height of the gate in its fully open position. The fully open height can also be changed during the simulation using the operating control commands (this is useful where the operating control definition is generic, ie. non-structure specific). Not used if channel is non-operational, ie. a SP channel.
- (iv) **All Weirs:** Weir coefficient adjustment factor to be primarily used for model calibration or sensitivity testing. The weir coefficient is multiplied by this value. The resulting weir coefficient can be viewed in the .eof file and over time in the \_TSL GIS layer. If zero or negative an adjustment factor of 1.0 (ie. no adjustment) is applied.

**(i) HConF\_or\_WC (previously Height\_Cont)**

- (i) **All new weirs:** Weir coefficient,  $C_d$ , in its dimensionless form as in the equation below. If less than or equal to zero the default value for the weir type in the table further below is used. Note that for WO and WV weirs the default is to recalculate  $C_d$  every timestep. Entering a value greater than zero (0) will override this and apply a fixed  $C_d$ . For WD weirs the user must specify a non-zero positive value.

Note that published weir coefficients may be based on other non-dimensional or dimensional forms of the equation below, therefore care should be taken in ensuring the coefficient is converted to comply with the form below if this is the case.

$$Q = \frac{2}{3} C_{sf} C_d B \sqrt{2g} H^{Ex}$$

- (ii) **RG, SG:** As above if a weir has been specified (eg. a SG WB channel).
- (iii) **SP:** As above for non-operated spillways.

**(j) WConF\_or\_WEx (previously Width\_Cont)**

- (i) **All new weirs:** Weir flow equation exponent  $Ex$  in the equation above. If less than or equal to zero the default value for the weir type in the table further below is used. The default value is 1.5 for all weir types except for WV which is 2.5.
- (ii) **RG, SG:** As above if a weir has been specified (eg. a SG WB channel).
- (iii) **SP:** As above for non-operated spillways.

**(k) EntryC\_or\_WSa (previously Entry\_Loss)**

- (i) **All new weirs:** Sets the submergence factor “a” exponent in the equation below for calculating the weir submergence factor  $C_{sf}$ . If less than or equal to zero the default value for the weir type in the table further below is used.

$$C_{sf} = \left( 1 - \left( H_d / H_u \right)^a \right)^b$$

(ii) **RG, SG:** As above if a weir has been specified (eg. a SG WB channel).

(iii) **SP:** As above for non-operated spillways.

**(l) ExitC\_or\_WSb (previously Exit\_Loss)**

(i) **All new weirs:** Sets the submergence factor “b” exponent in the equation above for calculating the weir submergence factor  $C_{sf}$ . If less than or equal to zero the default value for the weir type in the table below is used.

(ii) **RG, SG:** As above if a weir has been specified (eg. a SG WB channel).

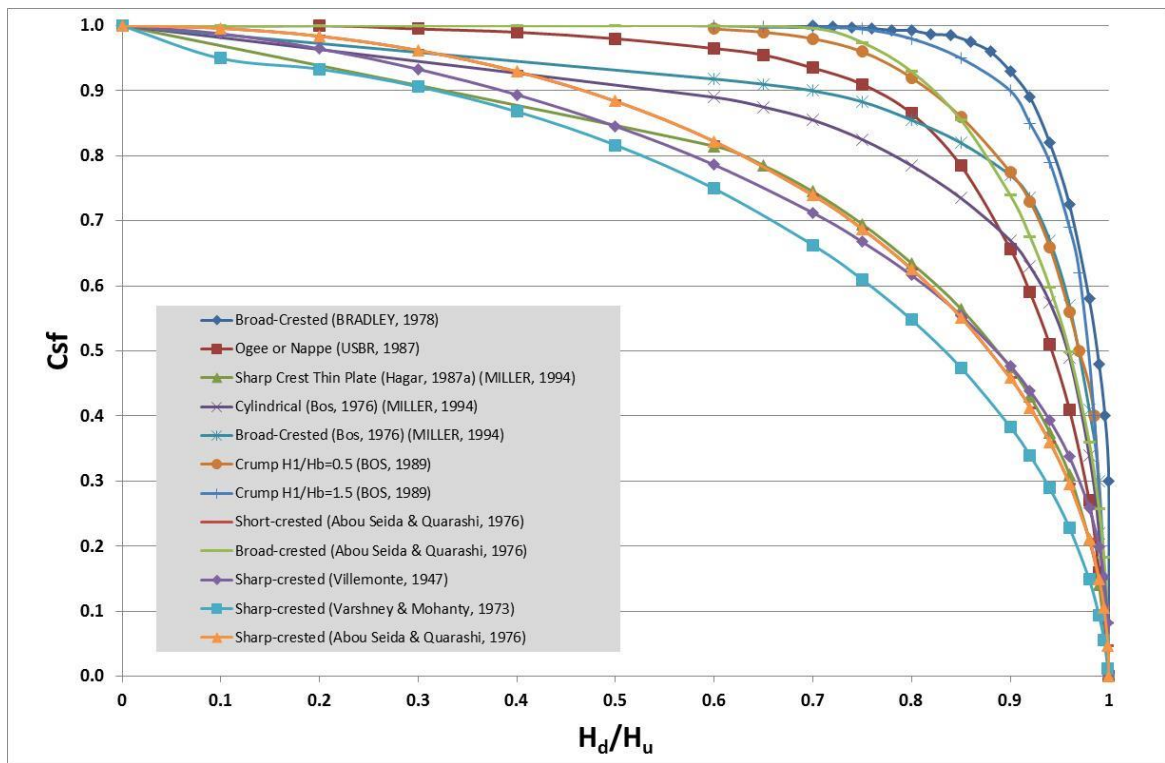
(iii) **SP:** As above for non-operated spillways.

**Default Attribute Values for the Weir Equation**

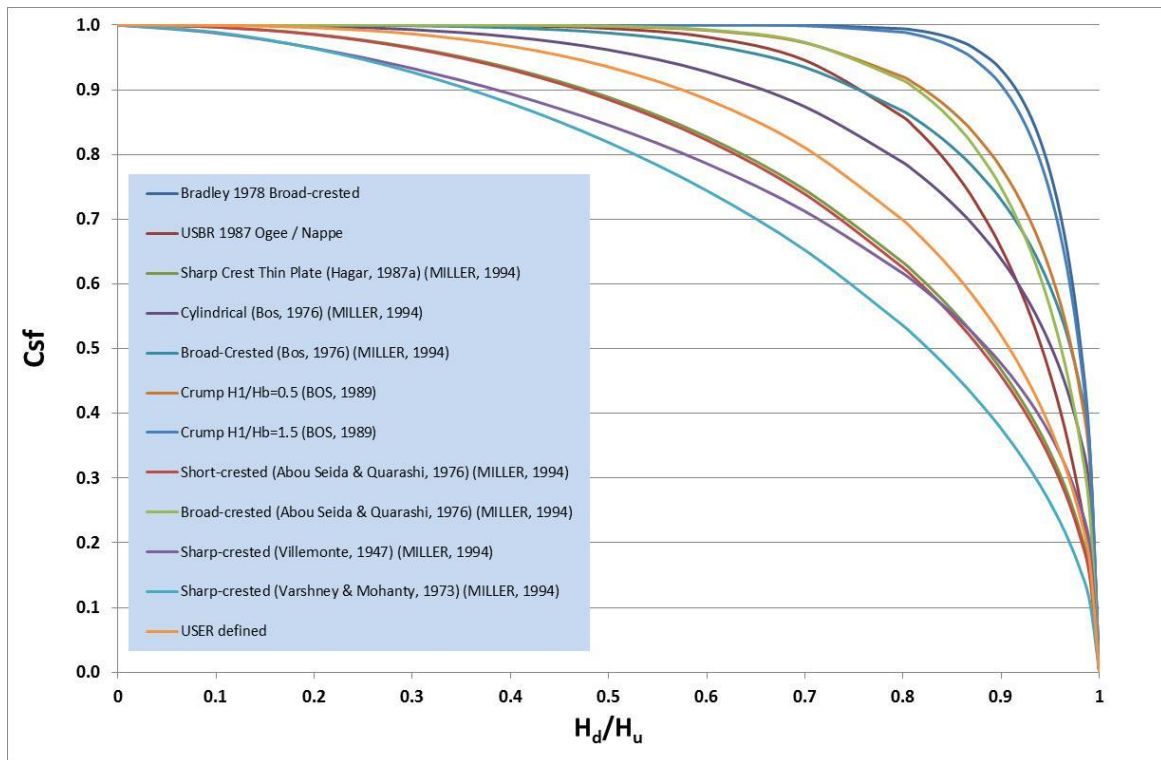
(Note that some of these values are derived from dimensional forms of the weir equations, hence the number of significant figures!)

<b>Channel Type</b>	<b><math>C_d</math> (HConF_or_WC)</b>	<b>Ex (WConF_or_WEx)</b>	<b>a (EntryC_or_WSa)</b>	<b>B (ExitC_or_WSb)</b>
<b>SP</b>	0.75	1.5	6.992	0.648
<b>WB</b>	0.577	1.5	8.55	0.556
<b>WC</b>	0.508	1.5	17.87	0.59
<b>WD</b>	User Defined	1.5	3.0	0.5
<b>WO</b>	Calculated	1.5	6.992	0.648
<b>WR</b>	0.62	1.5	2.205	0.483
<b>WT</b>	0.63	1.5	2.205	0.483
<b>WV</b>	Calculated	2.5	2.205	0.483
<b>WW</b>	0.542	1.5	21.15	0.627

(m) For weir submergence curves, the curves shown in the chart below were sourced from the literature and best fit curves as shown in the second chart were fitted. The equation found to provide the best fits is that as shown above as originally derived by Villemonte. Further details will be provided in the updated manual.



**Submergence Curves from the Literature**



**Submergence Curves using Equation**

(n) Prior to the 2013 release a variable weir channel could be specified as a WV channel. As of the 2013 release WV channels are processed as a V-notch weir, so any WV channels will need to be changed to VW.



- (o) **For the old W channels**, “Weir Approach == METHOD A” has been reinstated as the default setting for the 2013-12 release. METHOD A utilises the Bradley submergence approach, which, after further analysis and feedback from users, is a more preferred approach. Note: this change does not apply to the new weirs, which use the submergence approach described previously. “Defaults == PRE 2013” will reset “Weir Approach ==” to METHOD B.

## Operating Controls (.toc File)

- 42 The .toc file contains operating rules that can be applied to hydraulic structures, pumps and other controllable devices. Each set of operating rules is contained within a Control definition. More than one structure/device can use the same control definition.

The .toc file is referenced via the .ecf (not .tcf) file using the command below:

**Read Operating Controls File == <.toc\_filename>**

- 43 The operating rules for a control can only occur within a .toc file. More than one .toc file can be set up and accessed should there be a need to break the control definitions into several files (for example, all pump controls could be placed in one file and sluice gate controls in another).
- 44 A .toc file can only contain “Define Control ==”...“End Define” blocks.
- 45 Each “**Define Control ==**” must include a keyword indicating the type of structure/device as per below:

Define [ Culvert | Pump | Q Channel | Sluice | Spillway ] Control == <control\_id>

....

End Define

where <control\_id> is a unique control definition name. For a 1d\_nwk channel to use the control, <control\_id> must be entered into the 1d\_nwk Inlet\_Type attribute. As mentioned above, more than one channel can reference the same control. For example, several pumps may utilise the same operational control logic.

- 46 Within the control definition, commands specific to the type of structure/device can be used to adjust the structure/device’s operation. The commands available for each type of control are described below.
- 47 Each “**Define Control ==**” block consists of three sections.:
  - (a) The default settings for the control’s commands, which are usually placed at the top of the definition and prior to the logical rules. The default settings are the values used for a command in the event the command is not used within the logic rules.
  - (b) User defined variables as described further below.
  - (c) One or more logical rules as described further below.

### 48 “**Define Culvert Control ==**” Commands (Rectangular, R, channels only)

- (a) Gate Type == [ VERTICAL UNDERFLOW | VERTICAL OVERFLOW | HORIZONTAL SINGLE | HORIZONTAL DOUBLE ]

Sets the type of gate arrangement. VERTICAL/HORIZONTAL indicate the direction of the gate movement. SINGLE is a single gate, while DOUBLE are two gates that move in/out from either side.

- (b) Gate Speed [ {} | (min) ] == <speed>  
The speed at which the gate moves. Units are m/s or ft/s or if "(min)" is specified in m/min or ft/min.
- (c) Gate Period [ {} | (min) ] == <period>  
The time taken to fully open a closed gate or to fully close an open gate. Units are in hours or minutes if "(min)" is specified.
- (d) Gate Height Fully Open == <height>  
For vertically moving gates the height (not elevation) of the gate when fully open above the gate's seat. Units are in m or ft. If not set, the 1d\_nwk "Height" attribute is used.
- (e) Gate Width Fully Open == <width>  
For horizontally moving gates the width of the gate(s) when fully open. Units are in m or ft. If not set, the 1d\_nwk "Width\_or\_Dia" attribute is used.
- (f) Gate Opening [ {} | (%) ] == [ [ ++ | -- | {} ]<opening> | CLOSE | OPEN | NO CHANGE ]  
The position the gate is to be operated towards. A "++" or "--" before <opening> will incrementally open or close the gate by the value of <opening>, otherwise <opening> is taken as the absolute position. The units of <opening> are in m or ft, unless "(%)" is specified where it is the percentage of the fully gate open position. CLOSE will start moving the gate to the fully closed position, while OPEN will start moving the gate to the fully open position. NO CHANGE means that the gate operation remains unchanged.
- (g) Cd == <Cd>  
Sets the discharge coefficient.
- (h) Method == [ NAIR 2003 ]  
Sets which method to use for the hydraulic calculations. At present only that of Nair, 2003 is available. Note: for Nair 2003 the culvert must be drowned at all times.

#### 49 "Define Pump Control == " Commands

- (a) Pump Operation == [ ON | OFF ]  
Turns the pump on or off.
- (b) Pump Period [ {} | (min) ] == <period>  
The time taken to start the pump up or shut it down. Units are in hours or minutes if "(min)" is specified.
- (c) Pump Capacity == [ <flow> | <discharge\_curve> ]  
The flow capacity of the pump. If a constant flow rate, specify <flow>. If a head-discharge curve specify the name of the curve in the Depth Discharge Database.
- (d) Pump Number == <no\_pumps>  
Number of pumps in parallel.

#### 50 "Define Q Channel Control == " Commands

- (a) Gate Speed [ {} | (min) ] == <speed>  
The speed at which the gate moves. Units are m/s or ft/s or if "(min)" is specified in m/min or ft/min.
- (b) Gate Period [ {} | (min) ] == <period>  
The time taken to fully open a closed gate or to fully close an open gate. Units are in hours or minutes if "(min)" is specified.

- (c) Gate Opening [ {} | (%) ] == [ [ ++ | -- | {} ]<opening> | CLOSE | OPEN | NO CHANGE ]  
 The position the gate is to be operated towards. A “++” or “--” before <opening> will incrementally open or close the gate by the value of <opening>, otherwise <opening> is taken as the absolute position. The units of <opening> are in m or ft, unless “(%)” is specified where it is the percentage of the fully gate open position. CLOSE will start moving the gate to the fully closed position, while OPEN will start moving the gate to the fully open position. NO CHANGE means that the gate operation remains unchanged.

## 51 “Define Spillway Control ==” Commands (Rectangular, R, channels only)

- (a) Gate Speed [ {} | (min) ] == <speed>  
 The speed at which the gate moves. Units are m/s or ft/s or if “(min)” is specified in m/min or ft/min.
- (b) Gate Period [ {} | (min) ] == <period>  
 The time taken to fully open a closed gate or to fully close an open gate. Units are in hours or minutes if “(min)” is specified.
- (c) Gate Height Fully Open == <height>  
 The height (not elevation) of the gate when fully open above the gate’s seat. Units are in m or ft. If not set, the 1d\_nwk “Height” attribute is used.
- (d) Gate Seat Vertical Offset == <offset>  
 The difference in height in m or ft between the spillway crest and the seat of the gate.
- (e) Gate Opening [ {} | (%) ] == [ [ ++ | -- | {} ]<opening> | CLOSE | OPEN | NO CHANGE ]  
 The position the gate is to be operated towards. A “++” or “--” before <opening> will incrementally open or close the gate by the value of <opening>, otherwise <opening> is taken as the absolute position. The units of <opening> are in m or ft, unless “(%)” is specified where it is the percentage of the fully gate open position. CLOSE will start moving the gate to the fully closed position, while OPEN will start moving the gate to the fully open position. NO CHANGE means that the gate operation remains unchanged.
- (f) Cd Spillway == <Cd>  
 Sets the discharge coefficient of the spillway.
- (g) Cd Gate == <Cd>  
 Sets the discharge coefficient of the gate.
- (h) Method == [ USACE 111-21 ]  
 Sets which method to use for the hydraulic calculations. At present only that documented by USACE Hydraulic Design Criteria (HDC) Sheet 111-21 is available.

## 52 User Defined Variables

- (a) If a line in the control definition cannot be processed as one of the commands described above, and it is not within an If...End If block, it is treated as a variable definition using the syntax:

<variable> == <variable\_value>

where <variable> is the variable name, while <variable\_value> must conform to one of the options below. Any characters can be used for the variable name, but it is recommended to use only letters, numbers and underscores and to avoid spaces. Note that variables can be redefined at any point with

the control definition. Also note that these variables only apply to the control. For global variables, use the new “Set Variable ==” feature.

- (b) <variable> == <constant>  
Sets the variable to the value of <constant>. Must be a number.
- (c) <variable> == TIME of MODEL  
Sets the variable to the simulation time in hours.
- (d) <variable> == TIME in 24H  
Sets the variable to the simulation time in hours on a 24 hour clock (ie. will always be between 0 and 24 hours). A simulation time of zero is equivalent to midnight.
- (e) <variable> == DAY of WEEK  
Sets the variable to the day of the week where Sunday is 1 and Saturday is 7. The keywords “Sun”, “Mon”, “Tue”, “Wed”, “Thu”, “Fri” and “Sat” can also be used when using the variable in a logic rule.
- (f) <variable> == PERIOD NO CHANGE  
Sets the variable to the time in hours since the last time there was a change in operation.
- (g) <variable> == H1D <node\_id>  
Sets the variable to the water level at the 1D node named <node\_id>.
- (h) <variable> == Q1D <channel\_id>  
Sets the variable to the flow in the 1D channel named <channel\_id>.
- (i) <variable> == H2D <x>,<y>  
Sets the variable to the water level at the 2D cell located at the XY coordinates <x>,<y>
- (j) <variable> == HU  
Sets the variable to the water level at the upstream node based on the digitised direction of the channel.
- (k) <variable> == HD  
Sets the variable to the water level at the downstream node based on the digitised direction of the channel.
- (l) <variable> == dHUD  
Sets the variable to the difference in water level between the upstream and downstream nodes based on the digitised direction of the channel. Will be negative if flow is in opposite direction to digitised direction.
- (m) <variable> == H1  
Sets the variable to the upstream water level of the channel based on the flow direction.
- (n) <variable> == H2  
Sets the variable to the downstream water level of the channel based on the flow direction.
- (o) <variable> == dH12  
Sets the variable to the difference in water level between the upstream and downstream nodes based on the flow direction of the channel. Will always be positive.
- (p) <variable> == YU  
Sets the variable to the depth above the structure invert of the upstream node based on the digitised direction of the channel.

- (q) <variable> == YD  
Sets the variable to the depth above the structure invert of the downstream node based on the digitised direction of the channel.
- (r) <variable> == Y1  
Sets the variable to the upstream depth relative to the structure invert based on the flow direction.
- (s) <variable> == Y2  
Sets the variable to the downstream depth relative to the structure invert based on the flow direction.

### 53 Logic Rules

- (a) The logic rules consist of using If...End If blocks using the construct below.
- (b) If <condition\_1> [ [ and | or ] <condition\_2> ] [ [ and | or ] <condition\_3> ]....  
...enter one or more commands or variable definition lines  
[ Else If <condition\_1> [ [ and | or ] <condition\_2> ] [ [ and | or ] <condition\_3> ]....  
...enter one or more commands or variable definition lines]  
[ Else If...repeat as needed]  
[ Else  
...enter one or more commands or variable definition lines]  
End If
- (c) <condition\_1> must be a conditional operation that includes one of the symbols "=", ">", ">=", "<" or "<=". The left and right sides of the condition must be a single value or a variable name. Optionally the variable may be operated on by following with a "+", "-", "\*" or "/" and a constant value. For example, a condition could be "x + 2 < 3".
- (d) If more than one condition is to be applied, the conditions must be separated by an "and" or "or". <condition\_2>, <condition\_3>, etc have the same format as for <condition\_1> above.
- (e) If...End If blocks can be nested inside other If...End If blocks (use indenting!).

54 The example below shows the control definition for a pump that operates between 6am and 6pm Monday to Friday and 10am and 4pm on the weekends.

```

!
Define Pump Control == P_8_to_1

! Default settings
Pump Operation == OFF ! Must be ON or OFF

! Set user variables
t == TIME 24h ! Must be between 0 and 24. Simulation time of 0 is midnight.
wday == DAY of WEEK ! Must be one of Sun, Mon, Tue, Wed, Thu, Fri, Sat. Treated as an integer from 1 to 7.

! Provide logic
if wday >= Mon and wday <= Fri
    if t > 6 and t < 18
        Pump Capacity == 0.1 ! Must be single value which is the pump flow in m3/s or refer to a pump curve
        Pump Operation == ON
        !Pump Number == 2 ! Number of pumps (default is 1)
    else
        Pump Operation == OFF
    else
        Pump Flow == Pump_A ! specify an efficiency curve
    end if
else
    if t > 10 and t < 16
        Pump Capacity == 0.1 ! Must be single value which is the pump flow in m3/s or refer to a pump curve
        Pump Operation == ON
    else
        Pump Operation == OFF
    end if
end if

End Define

```

- 55 The example below shows the control definitions for a gravity released discharge to a power station (modelled as a pump) and the gated discharge through the reservoir (courtesy of Department Natural Resources, Wales).

```

!
Define Pump Control == PG_Pump

    ! Default settings
    Pump Capacity == 19.3
    Pump Operation == OFF ! Must be ON or OFF
    Pump Period (min) == 10

    ! Set user variables
    h_Reservoir == H2D 275300, 286250 ! 2D water level in reservoir

    ! Provide logic
    if h_Reservoir > 337.4
        Pump Operation == ON
    else if h_Reservoir < 337.3
        Pump Operation == OFF
    else
        Pump Operation == NO CHANGE
    end if

End Define

!
Define Q Channel Control == Q_Release

    ! Set default structure values
    Gate Period (min) == 600 ! Time taken to open gates from closed to fully open
    Gate Opening == CLOSE

    ! Define and initialise input variables to logic controls
    h_reservoir == H2D 275300, 286250 ! 2D water level in reservoir
    period_no_change == PERIOD NO CHANGE
    wait_period_rise == 1 ! Time interval before next check
    wait_period_fall == 1

    ! Logic Blocks
    if period_no_change < wait_period_rise
        Gate Opening == NO CHANGE
    else if h_reservoir > 337.5 and h_reservoir == HIGHER
        Gate Opening == ++10 ! ++ indicates to increase by 10 the gate opening
        wait_period_rise == 1
        wait_period_fall == 3
    else if period_no_change < wait_period_fall
        Gate Opening == NO CHANGE
    else
        Gate Opening == --10 ! -- indicates to decrease by 10 the gate opening
        wait_period_rise == 1
        wait_period_fall == 1
    end if

End Define

```

## Build 2013-12-AB – New Features, Enhancements and Bug Fixes

The new features, enhancements and bug fixes in Build 2013-12-AB are:

### 56 “If Scenario” / “If Event” and “Set Variable”

- (a) “If Scenario ==” and “If Event ==” now operate at a higher level than “Set Variable ==”, so that they can be used to set different values for the same variable. Previously, variables were set prior to processing any “If Scenario” or “If Event” logic blocks, therefore, variables could not vary with different scenarios/events. For more information see Item [15](#) in these notes.
- (b) For example, in the case below the variable “2D\_CELL\_SIZE” is used to set both the grid output resolution and the cell size of the model.

In the .tcf file we have:

```
If Scenario == 2m
    Set Variable 2D_CELL_SIZE == 2
Else If Scenario == 5m
    Set Variable 2D_CELL_SIZE == 5
End If
...
Grid Output Cell Size == <<2D_CELL_SIZE>>
...
```

And in the .tgc file we have:

```
Cell Size == <<2D_CELL_SIZE>>
```

### 57 Scenarios and Events automatically set as Variables

- (a) Any scenarios and events are now automatically set as a variable that can be used within your control files.
- (b) For example, if your model results are to be output to different folders depending on Scenario 1 (~s1~), enter the following into the .tcf file noting the use of << and >> to delineate the variable name as described in Item [15](#) in these notes.

```
Output Folder == ..\results\<<~s1~>>
```

In the case above if Scenario ~s1~ is set to “OpA”, TUFLOW automatically sets a variable named “~s1~” to a value of “OpA”, and the output will be directed to ..\results\OpA.

- (c) As an extension to the example above if the output folder is to also include the first event name, which, for example, is the return period of the flood, the following could be used.

```
Output Folder == ..\results\<<~e1~>>_<<~s1~>>
```

If Event ~e1~ is set to “Q100”, the output will be directed to ..\results\Q100\_OpA.



## 58 Support for Binary Grid (.flt format)

- (a) Binary grids (.flt format) can now be used instead of ASCII (.asc) grids. .flt grids are readable by most mainstream GIS software and TUFLOW visualisation software such as SMS. The .flt format is orders of magnitude faster than .asc grids for reading and writing, and is viewed much more rapidly in the GIS.
- (b) When TUFLOW reads a grid using the .tgc “Read Grid ==” command, the format is controlled by the file extension (use .flt for binary grids, otherwise an ASCII grid is assumed regardless of the file’s extension).
- (c) Use the new command “Grid Format == [ ASC | {FLT} ]” to set the format that TUFLOW uses to write output grids. **Note that the default for the 2013-12-AB build is FLT, therefore, “Grid Format == ASC” must be specified to write output grids using the ASCII format.**
- (d) Should you want to convert from .asc to .flt or vice versa, the TUFLOW utility asc\_to\_asc.exe is being updated to convert from one to the other (note that the utility maybe renamed grid\_to\_grid.exe or similar).

59 “Grid Approach == METHOD B” is the default setting for 2013-12-AB when interpolating DEM grids. See Item 121 in the [2011-09 / 2012-05 release notes](#) for more information.

## 60 New .tgc commands “Set CWF ==”, “Read GIS CWF ==” and “Read Grid CWF ==”.

These commands can be used to adjust the 2D cell flow widths (in the same manner as 2D flow constrictions (FC)), noting that the changed flow width applies to all depths. The CFW is a factor, for example 0.1 will limit the flow width to 10%.

## 61 Can now select which attribute to use for .tgc “Read GIS ==” commands.

- (a) For .tgc “Read GIS” commands that require only one attribute (eg. Code, Mat, Soil, GWL, GWD, IWL, CnM, WrF, FLC, CWF, SRF, Zpt), this attribute no longer needs to be the first attribute. If the attribute is not in the first field position, the position can be specified by including it as an argument before the GIS filename.
- (b) For example, a buildings’ outline GIS layer called “2d\_buildings.shp” has four attributes: “Prop\_ID”, “Material”, “SRF”, “Pad\_Depth”  
Rather than copy the layer three times and set up the appropriate attribute as the first column, the original layer can be used without modification using the following three .tgc commands.

```
Read GIS Mat == 2 | gis\2d_buildings.shp
Read GIS SRF == 3 | gis\2d_buildings.shp
Read GIS Zpt ADD == 4 | gis\2d_buildings.shp
```

If no number exists then the first attribute is assumed.

**62 New Horton Soil Infiltration Feature**

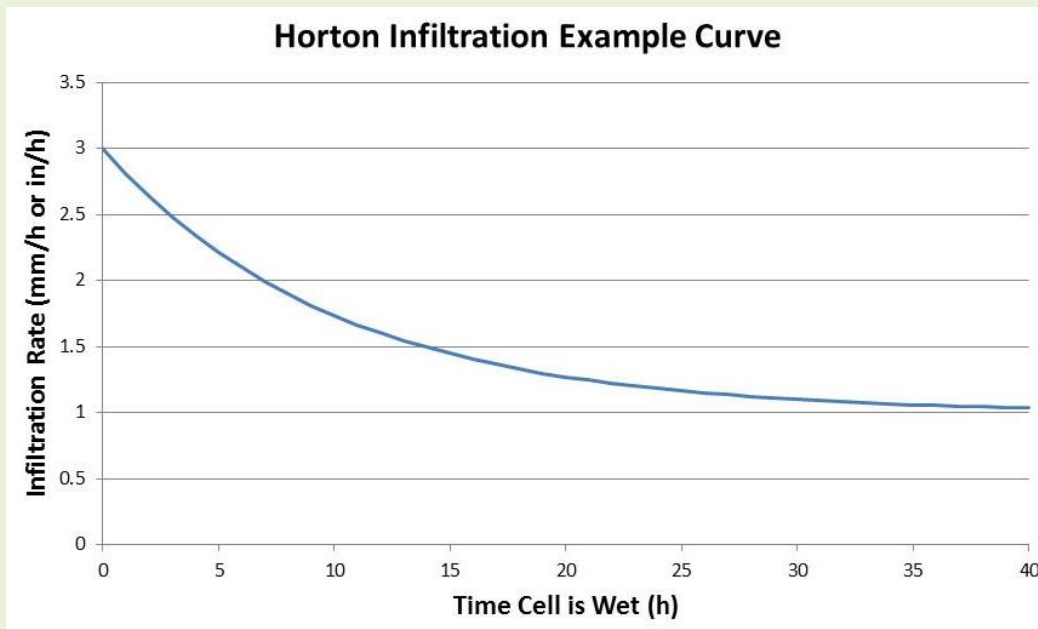
(a) The Horton approach utilises the equation

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

where  $f_0$  is the initial infiltration rate in mm/h or in/h,  $f_c$  is the final (indefinite) infiltration rate,  $t$  is time in hours and  $k$  is the Horton decay rate. For the TUFLOW implementation, the time  $t$  is the period of time that the cell is wet.

(b) If an initial loss (IL) is specified, the initial loss is applied first, followed by the Horton infiltration.

(c) The chart below shows an example of how the infiltration rate varies over time for  $f_0$  equal to 3,  $f_c$  equal to 1 and  $k$  equal to 0.1.



(d) The table below shows the table from the 2011-09 and 2012-05 release notes with the Horton parameters added.

Method	.tsoilf Parameters							
	1 <sup>st</sup>	2 <sup>nd</sup>	3 <sup>rd</sup>	4 <sup>th</sup>	5 <sup>th</sup>	6 <sup>th</sup>	7 <sup>th</sup>	8 <sup>th</sup>
No Infiltration	Soil ID							
Initial Loss / Continuing Loss	Soil ID	ILCL	Initial Loss (mm or inches)	Continuing Loss (mm/h or in/h)	Porosity (Fraction)	Initial Moisture (Fraction)		
Green-Ampt	Soil ID	GA	USDA Soil Type	Initial Moisture (Fraction)	Max Ponding Depth (m)			
			Suction (mm or in)	Hydraulic Conductivity (mm/h or in/h)	Porosity (Fraction)	Initial Moisture (Fraction)	Max Ponding Depth (m or ft)	
Horton	Soil ID	HO	Initial Loss (mm or inches)	Initial Loss Rate ( $f_0$ ) (mm/h or in/h)	Final Loss Rate ( $f_c$ ) (mm/h or in/h)	Exponential Decay Rate ( $k$ ) ( $h^{-1}$ )	Porosity (Fraction)	Initial Moisture (Fraction)

**63** In the **2d\_grd\_check layer** any soils check information is now confined to just the Soil ID at each cell. The .tlf file contains the parameters for each Soil ID.

**64 Restart Files**

- (a) New .tcf command “Write Restart Filename == [ {INCLUDE TIME} | OVERWRITE ]” that controls whether restart files are overwritten or are time-stamped.
- (b) If INCLUDE TIME (the default) is specified, .trf and .erf filenames are time-stamped and written for each restart output time, and the files are placed in a folder called “trf”.
- (c) OVERWRITE is the case for all prior versions of TUFLOW where the .trf and .erf files are overwritten each time restart files are written. If “Defaults == PRE 2013” is set the default setting is OVERWRITE.
- (d) .tcf command “Write Restart File Version == 2” uses a more detailed dump of the 2D domains that will result in a more precise restart.
- (e) To output restart files at regular intervals use the command “Write Restart File Interval ==”.

**65 New option to make writing of XF files dependent on the simulation settings.**

- (a) If different models utilise the same DEM(s) that are automatically dumped as .xf files to speed up processing for subsequent runs, the DEM is reprocessed each time a different model is run. To prevent this there is a new .tcf command “XF Files Include in Filename == <text>” to add text to the end of the .xf filenames. This also negates the need to make a copy of the DEM(s) to get around this issue.
- (b) For example, if two models use the same DEMs for setting Zpt values, use this command as per below.

In Model 1’s .tcf file:

```
XF Files Include in Filename == M1
```

In Model 2’s .tcf file:

```
XF Files Include in Filename == M2
```

- (c) If the same .tcf file is being used to run both models, use the scenario name as follows.

```
XF Files Include in Filename == <<~s1~>>
```

TUFLOW will add to the end of any .xf filenames the value of ~s1~ (see the previous item on the automatic setting of scenarios and events as variables). Thus, when ~s1~ changes from one simulation to the next, the .xf filenames are unique to that scenario and will not need reprocessing.

## 66 Bug fixes:

- (a) Bug fix that incorrectly applied the IL/CL and manually specified Green-Ampt infiltration methods for US Customary (English) Units. The bug does not apply where pre-defined Green-Ampt soil types were specified. **If using US Customary units and soil infiltration Build 2013-12-AB should be used.**
- (b) Bug fixed that output **unrealistic velocities in Q pits as they become submerged** due to the flow from the depth-discharge curve being divided by the water level difference rather than the approach depth. Whilst this does not affect the hydraulic calculations, very slight (fractions of a mm) may result compared with prior builds due to numerical precision issues.
- (c) If **“GIS Projection Check == ERROR”** is set (the default for 2012-05 builds and 2013-12-AA), an ERROR 0305 followed by termination of the simulation now occurs; previously a WARNING 0305 was issued and the simulation would continue.
- (d) The new .tcf **“Model Output Zone ==”** command (see Item 38) can now be embedded within “If Scenario” or “If Event” blocks. In Build 2013-12-AA, the last occurrence of this command would prevail irrespective of any If Scenario/Event blocks.
- (e) **Blue Kenue .t3 output files are now written correctly.** In Build 2013-12-AA if writing T3 output TUFLOW would crash.
- (f) The new **“Map Output Format ==” SMS TRIANGLES option now works** correctly if 1D WLLs are used.
- (g) Build 2013-12-AA may produce **“WARNING 1100”** messages for models that previously did not issue this WARNING. WARNING 1100 is a new WARNING introduced for the 2013-12 release, but for Build 2013-12-AA it is incorrectly issued. This has been corrected for Build 2013-12-AB. The WARNING is issued where a 1D structure’s invert/bed lies below the bed of the primary upstream and/or downstream channel. **The issuing of this WARNING has no effect on the results, although for Build 2013-12-AB onwards, if this WARNING is issued, a review of the 1D topography and structure invert/dimensions is recommended.**
- (h) A bug that incorrectly generated NA tables for nodes connected to a new weir channel has been fixed. TUFLOW would previously stop with a “...values in Column 1 not ascending message.”
- (i) The new **“\_TUFLOW\_Override\_<computer\_name>.tcf”** file for specific computer names introduced for 2013-12-AA **now works correctly.**
- (j) For the 2012-05 builds and 2013-12-AA, for 2D only models using multiple 2D domains stitched together using 2d\_bc 2D lines, TUFLOW would occasionally crash when writing output at the end of the simulation if **“Reveal 1D Nodes == ON”** has been set.
- (k) If **“Output Drive ==”** is specified, the **“\_ TUFLOW Simulations.log” files and any .xf files are not redirected** to the output drive. They will be written or appended to in their appropriate locations.
- (l) The .tgc file’s **“Write GIS Grid ==” and “Write GIS Zpts ==” commands** for the 2013-12-AA build incorrectly write empty layers. These commands **now work correctly** for Build 2013-12-AB.
- (m) Several minor bugs relating to the .2dm.info files and other map output features that work incorrectly for Build 2013-12-AA when using Output Zones.
- (n) Bug fixed if using **“Meshparts == ON”** with the new Output Zones feature.

## 67 GPU Module Changes in Build 2013-12-AB.

- (a) **If using the GPU Module provided with Build 2013-12-AB, it is likely that you will need to update your NVidia graphics drivers to their latest version. If you receive a message that the GPU card is incompatible, the most likely reason is that you will need to update the drivers.**
- (b) Improved performance at deep water level (HT) boundaries.
- (c) The default limiting Courant criteria for propagation is now set at 1.6 (previously 2.0). This may extend run times, however, in applications to very deep fast flowing water occasional instabilities could occur using 2.0. Also, the step rejection limit of 1.25 has been reduced to 1.1 to minimise the risk of overshooting the solution.
- (d) Now supports form loss coefficients using the "Read GIS FLC ==" .tgc command.
- (e) Now supports storage reduction factors using the "Read GIS SRF ==" .tgc command.
- (f) Supports the new "Read GIS CWF ==" .tgc command for adjusting cell widths.
- (g) Now supports use of setting Manning's n (or M) values directly using commands such as "Read Grid CnM ==" and "Read GIS CnM ==" instead of using materials.
- (h) The Horton soil infiltration approach is also implemented in the GPU solver.
- (i) Bug fix that incorrectly applied the IL/CL and manually specified Green-Ampt infiltration methods for US Customary (English) Units also applies to the GPU Solver. The bug does not apply where pre-defined Green-Ampt soil types were specified. **If using US Customary units Build 2013-12-AB should be used.**
- (j) ZH values around the perimeter of the 2D domain used for map output at the cell corners are now correctly extrapolated from the cell centre (ZC) values.
- (k) If different output formats have different output intervals for a GPU simulation (see Item 3 in these release notes), map output from the GPU may not perform as expected. This has been fixed for Build 2013-12-AB.

## Build 2013-12-AC – New Features, Enhancements and Bug Fixes

The new features, enhancements and bug fixes in Build 2013-12-AC are:

**68 Multiple 2D Domain 2D/2D Linking Enhancements.** A major feature of Build 2013-12-AC is significant enhancements to the 2D/2D linking of different 2D domains as described below. **It is recommended that models utilising 2D/2D linking upgrade to Build 2013-12-AC or later where possible.**

- (a) The default method for Build 2013-12-AC is “**Link 2D2D Approach == METHOD D**” and incorporates the enhancements and options discussed below. Prior to this build the default was METHOD B (METHOD C was an option and offered some benefits, but these were only minor).
- (b) **Note: Any models using the 2d\_bc 2D link lines will give different results using METHOD D. For backward compatibility specify “Link 2D2D Approach == METHOD B”.**
- (c) **The occurrence of a water level drop that is sometimes observed across 2D/2D links no longer occurs with METHOD D.**
- (d) **New .tcf command “Link 2D2D Distribute Flow == [ {ON} | OFF ]”** that improves the distribution of flow between 2D cells along 2d\_bc 2D link lines. The default is ON. For prior 2D/2D linking methods OFF applies.
- (e) **New .tcf command “Boundary Viscosity Factor ==”** multiplies the eddy viscosity coefficient by the factor along all open (external) boundaries and 2D / HX links. For these boundaries the eddy viscosity coefficient was previously set to zero for the boundary cells (this is because land boundaries require this). The default factor is kept at zero for backward compatibility, except for “**Link 2D2D Approach == METHOD D**” **the default is set to 1.0**, ie. the standard eddy viscosity coefficient is applied as this can provide an improved performance in flow patterns along the 2D link lines. Changing this value in the range of 0.0 to 5.0 (possibly higher) usually has little effect on results, however, increasing the factor can help “stabilise” unrealistic circulations along a boundary or 2D / HX link line without adversely affecting results. As with all new features, sensitivity test prior to adopting, especially for larger factors. Setting this value to 1.0 as the default and being able to vary this value for different boundaries/links will be considered for the TUFLOW 2014 release, so please send any feedback, good or bad, to [support@tuflow.com](mailto:support@tuflow.com).
- (f) **New .tcf command “Link 2D2D Global Stability Factor ==”** will globally factor all 2D link hidden 1D node storages to easily carry out a quick sensitivity test on the effect of increasing the storage of the hidden 1D nodes or to globally improve stability along 2D link lines if needed. The default value is 1.0. Usually, increasing the storage of the hidden 1D nodes by up to around a factor of 5 has only a slight effect on results. Note that any factor specified using the 2d\_bc “a” attribute is also applied on top of this factor. Also note that the default value for the 2d\_bc “a” attribute using METHOD D is 1.0 (prior methods use 2.0).
- (g) **New .tcf command “Link 2D2D Adjust Velocity Head Factor ==”** that sets the proportion of the velocity head by which to adjust the water levels along the 2D link line. A value of zero applies no adjustment whilst a value of 1.0 adjusts (downwards) the level by  $V^2/2g$ . The default is 0.0 (no adjustment). Some tidal models demonstrated improved performance by setting to 1.0, however, where there is rapid changes in velocities along a 2D link line, for example in an urban situation where velocities vary considerably from road to garden to between buildings, this feature may not improve the 2D link performance, therefore sensitivity test before adopting.

## 69 Enhancements to Map Outputs

- (a) **“ASC”** and **“FLT”** have been added to the **“Map Output Format ==”** command. This means that different map output settings (see Items 3 and 36) can be applied for ASC and FLT grid output. It also means that you can output both ASC and FLT grids during the same simulation (previously only one grid format was allowed as specified by the “Grid Format ==” command – see Item 58). Use of the “GRID” option and the “Grid Format ==” command can still be used, noting that “Grid Format ==” is used for other grid output such as the new DEM\_Z check grid of the model’s ground/bathymetric elevations.
- (b) **New .tcf command “Grid Output Origin == MODEL ORIGIN”** that forces grid output (eg. ASC or FLT) to have its origin exactly at the model’s lowest left coordinates for map output. If this command is not specified, TUFLOW adjusts the origin for the output grids by rounding to the **“Grid Output Cell Size ==”** so that all grids produced from different simulations using different model extents, and between different output zones, are all aligned. If “Grid Output Origin == MODEL ORIGIN” is specified, output grids of different origins (due to a change in the model’s schematisation or addition of 1D WLLs), or from different output zones, may not be aligned.
- (c) If **“Map Output Interval ==” is now set to zero**, no time based map output is produced. As described in Item 3, by preceding this command with the map output format, different map output intervals can be specified for different map output formats. For example, if only the maximums in ASC format are required, specify “ASC Map Output Interval == 0” (note tracking of maximums must be switched on – see Item 1(b)).
- (d) **WaterRIDE (WRB)** map output now works for TUFLOW double precision (DP) simulations.
- (e) For **BlueKenue (T3)** map output there has been a slight change in the file extensions used – see Item 34(d)).

## 70 Weir Flow over HX Lines.

- (a) The 2d\_bc layer’s “b” attribute can now be used to adjust the WrF value along HX lines to adjust or calibrate the flow rate across a 1D/2D HX link when upstream controlled weir flow occurs. This is similar to using the “a” attribute for applying additional energy (form) loss along the HX line.
- (b) To force the weir flow equation to be applied across all active HX cell sides when the flow is upstream controlled, the **new .tcf command “HX Force Weir Equation == ON”** can be specified (the default is OFF). This command lowers the HX cell centre elevation by 0.002m below the lowest active HX cell side elevation to force an adverse slope. If a value is specified the feature is turned ON and the value is the amount by which to set the HX cell centre elevation below the lowest active cell side, for example, “HX Force Weir Equation == 0.1” would use 0.1 instead of 0.002.

**Note that the default approach uses either weir flow or super-critical flow when the flow is upstream controlled, depending on whether the ground surface gradient from HX cell centre to cell side is adverse (weir flow) or not adverse (super-critical flow). When the flow is downstream controlled, regardless of the ground surface slope, the full 2D equations are applied including allowance for momentum across the HX 1D/2D link.**

- (c) New ERROR 2306 checks that WrF values are greater than zero – a value of zero would cause a crash.

**71 .dbf Deleted Record Flag Now Supported.** If a .shp file object's .dbf record (attribute data) is flagged in the .dbf file as a deleted record this is now supported in that TUFLOW ignores the GIS object and treats it as non-existent. This enhancement arose due to QGIS Version 2.2, rather than removing the object and its attribute data from the .shp and .dbf files if it was deleted by the user, sometimes retains the data and uses the .dbf file deleted flag to mark the object as being deleted. If using TUFLOW builds prior to 2013-12-AC with QGIS Version 2.2 the presence of the .dbf deleted flag would cause TUFLOW to terminate with "ERROR 0408 - End of .dbf file encountered whilst reading record". Prior versions of QGIS (as far as we can tell) and our testing of ArcGIS thus far indicate that these software remove the object and attribute data rather than use the .dbf deleted flag.

**72 USA FMA Challenge Models can be run without a licence by specifying "Demo Model == ON".** These models will be uploaded to the TUFLOW Wiki in a new Demonstration Model Zone where users can download models, simulate them and see how TUFLOW models can be set up in addition to using the TUFLOW Tutorial Models.

**73 Minor bug fixes:**

- (a) Bug fix that sometimes incorrectly applied the grid's origin if using "Read Grid" with an .asc or .flt grid that uses xllcenter/yllcenter to define the origin. Most grids use the xllcorner/yllcorner notation.
- (b) If no elevation points were written to an .xf file (eg. a 2d\_zsh polygon doesn't modify any Zpts), TUFLOW would crash when reading the .xf file on the next simulation. This has been fixed.
- (c) The .tbc "Global Rainfall == " command occasionally not working correctly has been fixed.
- (d) The "Read RowCol == " or "Read MID == " commands were broken for Build 2013-12-AB – these are now working again.
- (e) Minor bug fix that did not correctly apply the global setting for commands not specified within an Output Zone definition.
- (f) Output folders for Output Zones now work correctly if not specified. This also fixes a bug that would place the \_TS output layer in the last Output Zone folder instead of the main results folder.
- (g) Filenames to GIS layers in the QGIS .qgs workspace file are now written correctly with their prospective "P", "L" or "R" extension.

**74 GPU Module Changes in Build 2013-12-AC.**

- (a) **New .tcf command "GPU DP Check == [ {ERROR} | OFF ]".** The default setting of ERROR, causes TUFLOW to stop with ERROR 2420 advising that it is recommended to use the GPU solver via a single precision exe of TUFLOW. (The GPU solver due its explicit formulation and being depth based does not usually require to be run in DP mode.) There can also be substantial speed gains using SP on some GPU cards, and there is a significantly less memory footprint. **If DP is desired or required for the GPU Module specify "GPU DP Check == OFF" in the .tcf file, and run using a DP TUFLOW exe.**
- (b) More ERROR and WARNING messages have been built in to advise users if a feature they are applying is not available or ignored by the GPU Solver. As always, check your messages and use the check files/layers to quality control or review a model.
- (c) Water level boundary cells experiencing an inflow are now allowed to wet and dry. Previously, dry water level boundary cells could not generate an inflow.



- (d) The full cell surface area is now used for determining the volume of rainfall and infiltration applied to the cell regardless of the SRF value. This is the same approach as adopted in TUFLOW “Classic”. Previously, if a SRF was applied to a cell using the GPU Solver the rainfall/infiltration was based on the adjusted cell’s surface area.
- (e) FLC values were incorrectly applied in Build 2013-12-AB – this is now fixed.
- (f) Limitations on the allowable range in SRF and CWF values removed/expanded.
- (g) Restructure during the initialisation process so that the amount of memory required is ascertained more quickly and a message issued sooner if insufficient memory on the GPU card(s).
- (h) Minor bug fixes including:
  - (i) The .gpu.tlf log file is now correctly written to the new drive if “Output Drive ==” is specified.
  - (ii) Unsupported map output types are not written.
  - (iii) If different output zones or map formats use different map output intervals, these are now handled correctly for GPU simulations.
  - (iv) If all start map output times were after the simulation start time, the GPU solver would terminate – this is now fixed.

## Build 2013-12-AD – New Features, Enhancements and Bug Fixes

The new features, enhancements and bug fixes in Build 2013-12-AD are provided below. Please note that Build 2013-12-AD requires that licences have been updated to the 2014/2015 support year (2014 Feature Map). For any queries please contact [sales@tuflow.com](mailto:sales@tuflow.com).

### 75 Time Map Outputs:

- (a) Time output cut-offs can now also be specified as VxD (hazard) values. Instead of specifying “Time Output Cutoff Depths ==” use “Time Output Cutoff Hazards ==”. The associated map outputs will now be based on exceedance of the VxD value rather than depth value.
- (b) Fixed bug introduced to Build 2013-12-AA that incorrectly outputs time based map data (eg. time of maximum water level, time of first depth cutoff, duration of inundation).

76 **Gauge Level Output** now works for the 2013-12 release, and is now supported by the XMDF format

77 **RC Map Output** (used for animating route closures and levee overtopping):

- (a) The maximum RC category value is now tracked every timestep and output (if tracking maximums is switched on, which is the default).
- (b) The RC values are now always output as an integer representing the closure category specified by the user.
- (c) Fixed bug that would cause TUFLOW to crash or produce unpredictable map outputs if the RC map output data type was specified but no routes were specified using the “Read GIS Z Shape Route ==” .tgc command.
- (d) RC map output data type now works correctly for multiple 2D domain models (previously in some instances the RC values for a domain would be set to zero).

### 78 GRID (ASC or FLT) Map Output Format:

- (a) Time of maximum water level is now output correctly for grid formats.
- (b) All time outputs are now supported by the grid map output formats. The grid file extensions used are:
  - (i) “\_TDur\_<cutoff>” for duration of inundation
  - (ii) “\_TExc\_<cutoff>” for time <cutoff> is exceeded
  - (iii) “\_TMax\_h” for time of peak water level
  - (iv) “\_TMax\_V” for time of peak velocity
- (c) Gauge Level (GLO) map outputs are now available for grid formats.
- (d) Grid map output hazard categories are now output as integer grids except for Z0 and ZUK0, which continue to be output as real numbers.
- (e) Fixed bug that would occasionally not correctly produce grid output from multiple 2D domain models.
- (f) Fixed bug would incorrectly output grid formats. This bug occurred if GRID, ASC or FLT was the only map output format specified.
- (g) Fixed bug that would sometimes output the maximums for grid output when “Map Output Interval == 0 is used” as the values at the last timestep rather than the tracked maximums.

**79 External 1D Schemes (ie. ISIS, XP-SWMM and 12D-DDA):**

- (a) Fixed bugs introduced for Build 2013-12-AA that would produce erroneous output where WLLs are used for models that contain both TUFLOW (ESTRY) 1D channels and external 1D channels, and did not always correctly apply WLLs from external 1D schemes (ie. ISIS, XP-SWMM and 12D DDA).
- (b) Fixed bugs introduced in Build 2013-12-AA that did not output the x1d\_chans\_check, x1d\_nodes\_check, xWLLo\_check and xWLLp\_check layers. These layers are check layers for any channels and nodes, and any WLLs, for an external 1D scheme provided to TUFLOW.
- (c) Another attribute, ID\_DS, has been added to the x1d\_nodes\_check layer. ID\_DS contains the ID of the downstream node (for ISIS river units only). The attribute can be used to help detect a mismatch between the node IDs in the 1d\_isis\_nodes layer and the connectivity sent through from ISIS via the ISIS-TUFLOW link causing TUFLOW to stop with an ERROR 2042.

**80 New Hazard Categories:**

- (a) **ZAEM1** as presented in Figure 6 (shown below) in “*Technical flood risk management guideline: Flood hazard - Supporting document for the implementation of Australian Emergency Management Handbook 7, Managing the floodplain: Best practice in flood risk management in Australia*” by Australian Emergency Management Institute. ZAEM1 output values are 0 (zero) for no hazard and 1 to 6 for H1 to H6 respectively.

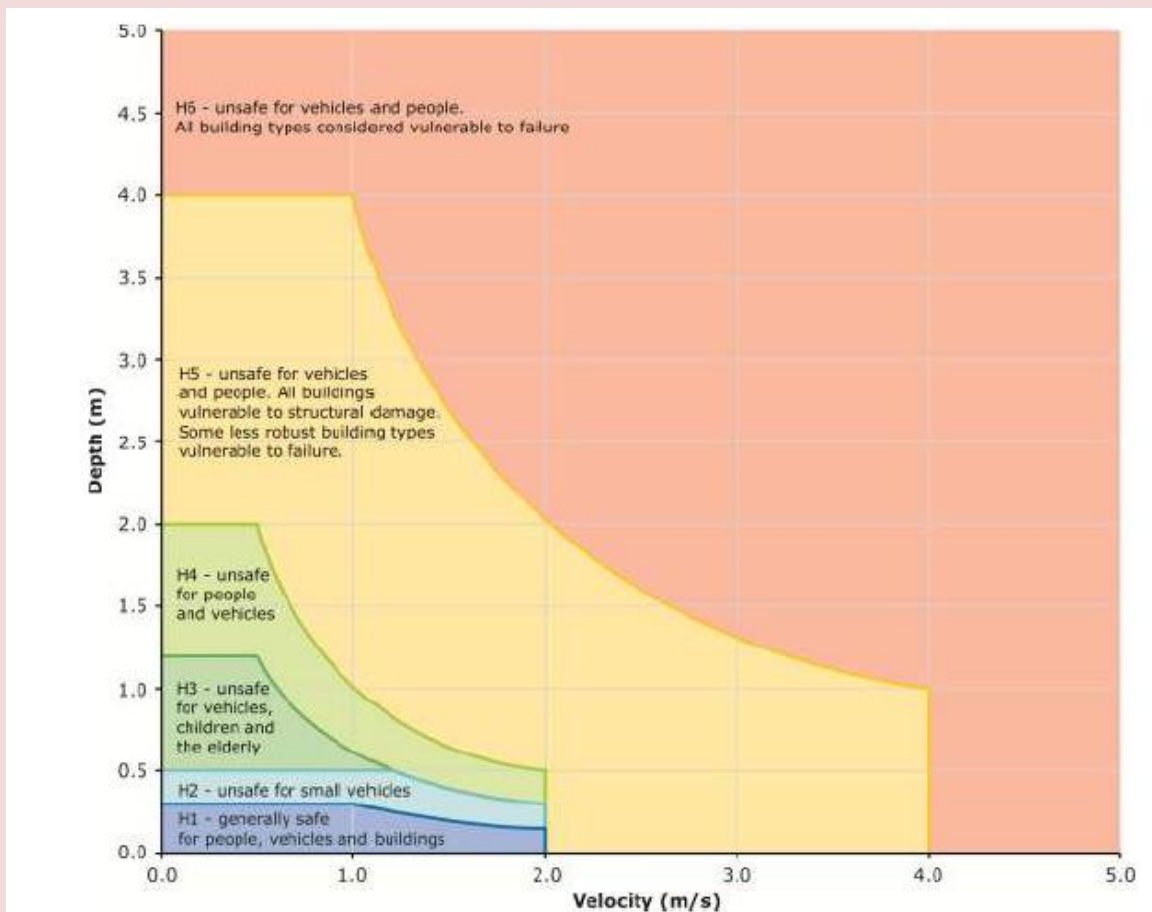
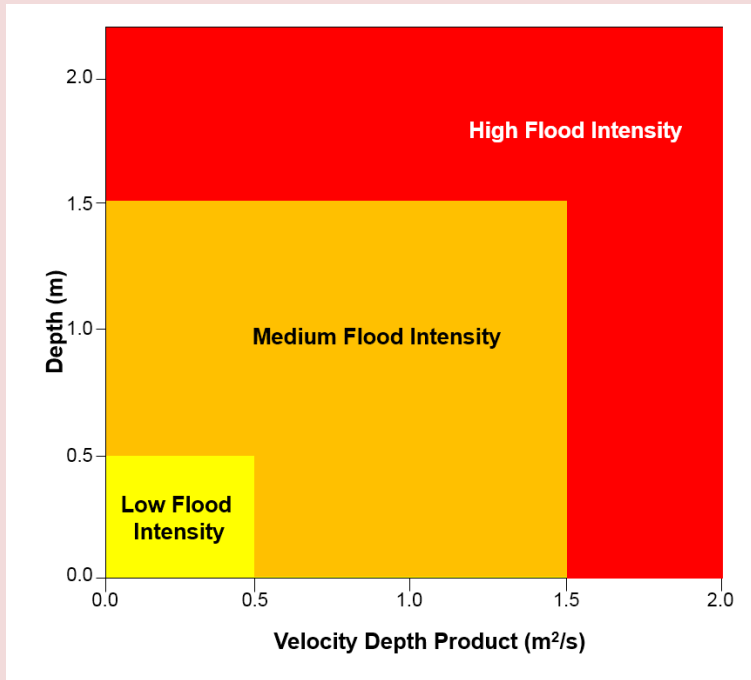
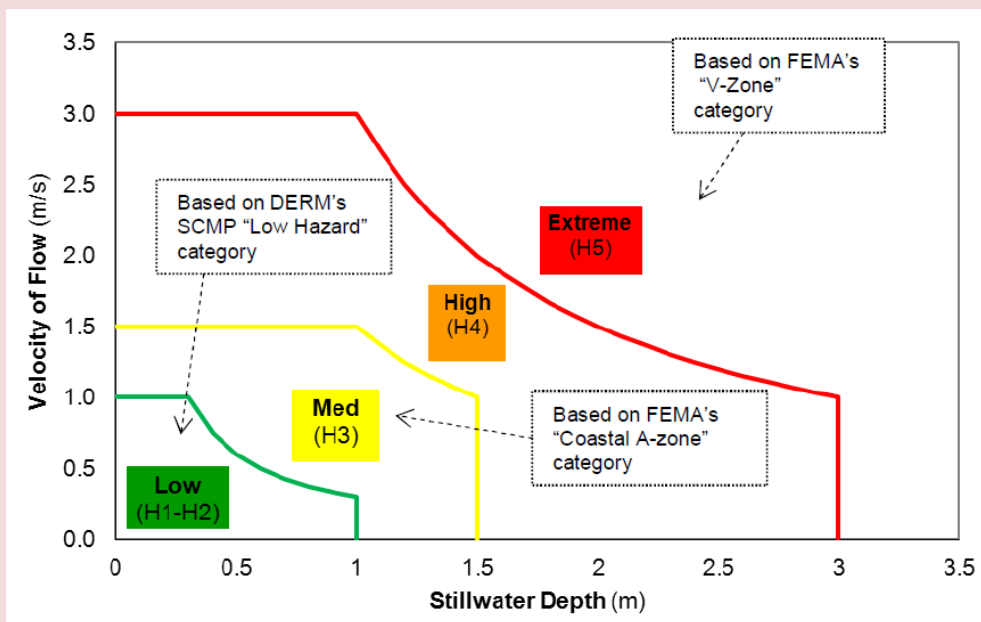


Figure 6 General flood hazard vulnerability curves

(b) **ZUSA1** which is based on the chart below created from the logic documented in the Flo-2D Mapper Manual, Version 2009. ZUSA1 output values are 0 (zero) for no hazard, 1 for Low Intensity, 2 for Medium and 3 for High. Note that as of this build for all hazard categories for models run using English Units the depth and velocity are converted to metric to correctly calculate the hazard category.



(c) **Z9**, which is based on the draft storm tide hydraulic hazard categorisation developed for Moreton Bay Regional Council by GHD. Z9 output values are 0 (zero) for no hazard and 1 to 5 for H1 to H5 respectively.



## 81 Check Grids:

- (a) New DEM\_M check grid containing the final material values in grid (ASC or FLT) format. Similar to the DEM\_Z check grid. The DEM\_M check grid can be included/excluded from being written using the DEM\_M flag for "Write Check Files INCLUDE or EXCLUDE ==".
- (b) The DEM\_M and DEM\_Z check grids are now also written if the model start up is forced to only process the .tgc file. **Tip:** To only process the .tgc file don't specify, or comment out, the "BC Control File ==" command.

82 Map Output data type "ZH", to output the bed elevations, now works correctly for the 2013-12 release.

83 Fixed bug that occurs if the soil initial moisture value exceeds the soil's porosity causing excessive infiltration. If the initial moisture exceeds the porosity the soil is now assumed to be fully saturated and unable to absorb any water.

84 Fixed bug introduced for the 2013-12 release that does not correctly apply 2D HT boundary values if the start of the HT polyline has two or more duplicate vertices. Note that this bug only occurs if the first vertex is duplicated.

85 If using **US Customary (English) Units** please note the following:

- (a) The Initial Loss (IL) and Continuing Loss (CL) values for materials were previously not converted from inches and inches/h to ft and ft/h as stated in Item 49 of the [2011-09 and 2012-05 release notes](#). As of Build 2013-12-AD the values are correctly converted. Note that for prior builds the values will need to be entered as ft and ft/s. Note that the IL/CL values referred to here are for the IL/CL applied to direct rainfall (RF) inflows. The IL/CL values used for soil infiltration are not affected.
- (b) For determination of hazard categories these are now correctly based on the equivalent metric charts/equations. For Z0 and ZUK0 the values are based on using feet and seconds.
- (c) Some output labels were reported using metric units (the values are correct and are in English Units, but the label was in metric). These labels have been corrected.

86 For earlier 2013-12 builds a VW (Variable Weir) 1D channel could cause TUFLOW to crash – this has now been fixed.

87 For all .csv tabular data referenced by the 1d\_tab layer format, except for XZ tables, a bug that did not correctly read tables in the same .csv file that are located in a vertical alignment has been fixed. For example, if a .csv file contains two tables with one above the other, for example one is for a bridge's height width relationship (HW table) and the other for the loss coefficients (LC table), the tables would not be read in correctly – this bug has been fixed.

88 Bug fixes that now automatically allocates the correct number of 2D SX cells to a 1d\_nwk node with the Conn\_1D\_2D attribute set to "SX" when:

- (a) the 1d\_nwk Conn\_No attribute is used to control the number of SX cells, and
- (b) if a culvert with more than one barrel is connected to the node, the number of barrels are now used to set the width, which controls the number of automatically selected SX cells.

Note that the fixes above only apply to nodes, not pits for which there are no known bugs.

89 Bug fix that would not report an "ERROR 0043 - Mismatch in number of If Scenario or If Event and End If commands" if this occurred in the .tgc file.

90 Fixed bug that would sometimes cause WLL output to not contain any values (because all WLL triangles would be set to dry) for an Output Zone if the Output Zone's map output formats were different to the whole of model map output formats.

91 In the 2012-05 and 2013-12 releases the .tgc file was not closed after processing and would remain locked for the duration of the simulation. This has been fixed.

**92 GPU Module:**

- (a) Hazard category map outputs produced from a GPU solver run are now written as integer values – previously Hazard categories were interpolated at cell corners and a non-integer value could result. When interpolating Hazard categories, the nearest integer is used based on the interpolated value.  
**Note that the maximum hazard from the GPU solver is not tracked every timestep as yet to minimise GPU memory usage, however, this feature is being built in as an option for the next major release. The maximum Hazard values are presently based on using the maximum depth and maximum velocity product.**
- (b) This build of the GPU solver is now compatible with Windows XP – the previous 2013-12 builds failed to integrate if using Windows XP.
- (c) Additional checks that the NVidia card's compute capability is compatible with the GPU solver have been built in (older NVidia cards can be incompatible).
- (d) As a result in a change in the compiler version due to the above changes a very slight (several millimetres) change in peak water levels may occur in sections of some models based on testing thus far. These changes have occurred for single precision runs with double precision simulations showing negligible (fractions of a mm) changes in peak levels.

## Build 2013-12-AE – Bug Fixes

Build 2013-12-AE introduces two bug fixes as outlined below:

- 93 Fixed licence issue if invalid licence handle was returned for a WIBU licence. This issue only occurred for Build 2013-12-AD. Previously the error below was returned:

```
Checking TUFLOW Threads Licence...Dongle Check Failed. Return Code C1/106.
```

This would cause the simulation to stop. TUFLOW will now attempt to re-take a licence and continue the simulation as per builds prior to 2013-12-AD

- 94 Fixed bug that would not recognise the 1D operational control variable types “dHUD” or “dH12” – for a description of these please see points 52(l) and 52(o) above. Previously the error below was returned:

```
ERROR 1522 - Unrecognised or unsupported operating control variable in line below, or invalid line.
```

```
Operating Control Line = <variable name> == dHUD
```

# TUFLOW GPU Module

This section contains information on the TUFLOW GPU Module including instructions on checking for compatible graphic cards and updating drivers.

**Note: The GPU Module does not utilise all of TUFLOW’s features, therefore, it is important to read this document to be aware of the features that can and cannot be used.**

**Note: Also see updates to the GPU Module functionality and bug fixes listed in:**

Item **67** for Build 2013-12-AB

Item **74** for Build 2013-12-AC

Item **92** for Build 2013-12-AD

## Table of Contents

<b>TUFLOW GPU Description.....</b>	<b>49</b>
Introduction .....	49
Conservativeness.....	49
Stability and Adaptive Timestepping.....	50
<b>Will the Results be the Same as TUFLOW “Classic”?.....</b>	<b>51</b>
<b>Compatible Graphics Cards .....</b>	<b>52</b>
Compute Capability.....	55
<b>Updating NVIDIA Drivers.....</b>	<b>55</b>
<b>Running TUFLOW GPU .....</b>	<b>56</b>
TUFLOW Control File Commands .....	56
Cell Codes.....	57
Topography .....	57
Bed Resistance .....	57
Initial Water Levels.....	57
Boundary Conditions.....	58
Infiltration.....	59
Output.....	59
Time-Series Output (2d_po) .....	59
Excluded TUFLOW Functionality.....	59
Single and Double Precision.....	59
Troubleshooting .....	60
FAQ.....	60



## TUFLOW GPU Description

### Introduction

TUFLOW's GPU Module, by BMT WBM, is a powerful new solver built into the existing TUFLOW software. As its name implies it utilises the substantial parallel computing ability of modern Graphics Processor Units, or GPUs.

TUFLOW GPU is an explicit solver for the full 2D Shallow Water Equations, including a sub-grid scale eddy viscosity model. The scheme is both volume and momentum conserving. The scheme is 1<sup>st</sup> order in space with various formulations available including 1<sup>st</sup>, 2<sup>nd</sup> and 4<sup>th</sup> order in time, with adaptive or fixed time-stepping.

Presently the solver runs 2D models only, but thanks to the power of modern GPUs very large models (>100 million cells) with fine grids can now be run within a sensible timeframe.

The front end is still TUFLOW, and very little modification of the model input is needed to utilise the GPU solver. Similarly the output data is still written by TUFLOW and the same output formats and data types are available. TUFLOW GPU was submitted for the 2012 UK Environment Agency 2D Benchmark Tests, where along with TUFLOW and TUFLOW FV, were top performers in terms of consistent results and run times. The scheme has also been benchmarked against theoretical solutions and through comparisons with TUFLOW and TUFLOW FV.

TUFLOW GPU in its present form is ideally suited to first-pass rapid and accurate hydraulic solutions, as an alternative to using hydrologic models (ie. whole of catchment modelling), real-time flood inundation forecasting and high resolution 2D solutions. TUFLOW's 1D pipe/pit/manhole functionality is planned to be incorporated in future versions.

### Conservativeness

The explicit solver computes the volume flow across cell boundaries and volume cannot leave one cell without being placed in its neighbour. As a result volume is conserved and mass error is 0%. The transfer of momentum across cell boundaries is computed in the same way and once external forces are considered (bed slope, bottom friction, and wet perimeters of non-uniform depth) momentum has been found to be conserved.

The scheme is unlike the TUFLOW "Classic" solution that solves the same equations implicitly using matrices, and therefore can successfully over step the solution using much larger timesteps (hence why it is important to monitor mass error in implicit schemes such as TUFLOW to check that the solution is converging).

## Stability and Adaptive Timestepping

There are three primary processes that determine the maximum time step that an explicit solution to the Shallow Water Equations can use:

- 1 The volume lost from a cell in any given time step cannot leave the cell with negative volume. If we express the Courant number as  $Nu = \frac{u dt}{dx}$ , then keeping  $Nu < 0.25$  guarantees that a cell cannot be depleted to zero volume in a single time step. In practice we have found  $Nu < 1.0$  to be a suitable upper limit for  $Nu$  since cells that have fluid leaving on all boundaries do not usually have the maximum velocity in the model.
- 2 The Shallow Water Equations admit harmonic solutions (i.e. shallow water waves with speed  $c = \sqrt{gh}$ ). This gives rise to a non-dimensional wave speed number  $Nc = \frac{c dt}{dx}$ . For the 4th order solver (the default and recommended solution), we have found the solution to remain stable for  $Nc \leq 2.0$ . For Build 2013-12-AB, the limiting default for  $Nc$  was reduced to 1.6 due to the occasional instability in very deep fast flowing water. The default target Courant criteria  $Nc$  is now set to 1.6 (previously was 2.0). For very deep, fast flowing water (eg. >20m and >4m/s), the GPU solution in one application experienced a brief spike causing the tracked maximums to show elevated mounds or bumps in the maximum water level surface. Reducing  $Nc$  to 1.6 resolved this issue. Note that as with any model / numerical solver, the results should always be scrutinised, especially for adaptive time-stepping solutions as these can experience a brief instability and then restabilise, but the tracked maximums may record the bounce.
- 3 Momentum diffusion. The sub-grid scale eddy viscosity term causes diffusion of momentum. This gives rise to non-dimensional diffusion number  $Nd = \frac{\nu dt}{dx^2}$ . Again for the 4th order solver we have found the solution to remain stable for  $Nd \leq 0.3$ .

After each time step the solver checks for model instability (NaNs or sudden increases in  $Nu, Nc, Nd$ ) and if necessary will reduce the time step and repeat the step. If adaptive time stepping is off the time step will not be increased again, and may be decreased again if further instabilities are encountered. If adaptive time-stepping is on the solver controls the time step to maintain  $Nu, Nc, Nd$  at or just under the limits quoted above of 1.0, 1.6, and 0.3 respectively. If desired the user may adjust these limiting values using "Maximum Courant Number ==". If NaNs are encountered this overall scalar will be reduced slightly and the time step repeated; the overall scalar will not be increased again, but may be decreased again if further instabilities are encountered. If a sudden increase in  $Nu, Nc, Nd$  is encountered the time step will be temporarily decreased and the time step repeated. The time step will then be allowed to gradually ramp up again until one of  $Nu, Nc, Nd$  reaches its control limit. The time step also can't exceed the plot output or map output time intervals.

Generally the solver has been found to be exceptionally stable, with rejected time steps rare. They may however occur if the user specifies an initial time step that is too high, or if there is sudden change in a boundary water level or hydrograph at some point in the simulation. If rejected time steps occur a warning message is printed in the .gpu.tlf log file.

## **Will the Results be the Same as TUFLOW “Classic”?**

No, the exact same results will not occur between a TUFLOW “Classic” and a TUFLOW GPU simulation. There are a range of reasons including the topographical representation, boundary treatment and representation of upstream controlled flow regimes. For example, TUFLOW “Classic” utilises ground elevations at the cell centres and mid-sides (therefore whole cells and/or cell sides can wet and dry, and thin obstructions such as fences can be represented), whilst GPU only utilises cell centre elevations (only whole cells wet and dry).

However, the primary reason is that the two solution schemes, whilst solving the same equations, are different. TUFLOW “Classic” is a second order semi-implicit (matrix) finite difference solution, whilst GPU is an explicit first order spatial finite volume solution. For tests such as those in the UK EA 2D Benchmarking, “Classic” and GPU produce similar results, especially when compared to the range in results produced by all the other schemes. The UK EA tests, however, are predominantly over shallow flow, where bed friction tends to dominate. For deep, fast flowing water the bed resistance term is less dominating and the representation of other terms such as eddy viscosity, the degree of numerical diffusion, whether 1<sup>st</sup> order or 2<sup>nd</sup> order, and other influences will have a more influential effect on the results. Observations to date between “Classic” and GPU are that similar levels result for shallow, friction dominated areas, while for deep, fast flowing models with river bends and other flow complexities TUFLOW GPU tends to produce higher water levels for the same Manning’s values (for example, 0.2m higher in 10m deep and 3m/s velocity).

As for any numerical modelling, models should ideally be calibrated. If calibration is not feasible and GPU (or any other scheme) is being cross-checked or validated to another solution, it is appropriate to adjust Manning’s n values and/or eddy viscosity coefficient/formulation within conventional bounds to achieve a reasonable match. The modeller should cross-check that the other solution is also using Manning’s n and eddy viscosity parameters that are within conventional bounds. If non-conventional parameters are required in either solution, further investigation should be carried out to ascertain the reason and the influence of the non-conventional parameters.

## Compatible Graphics Cards

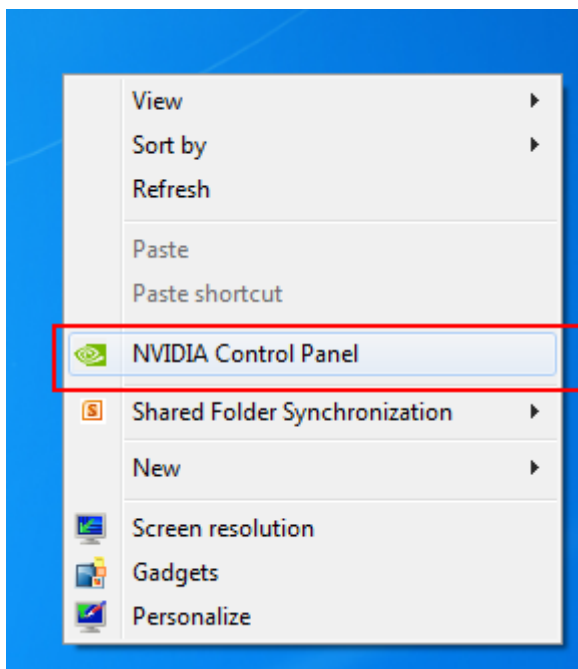
TUFLOW GPU requires an NVIDIA CUDA enabled GPU. A list of CUDA enabled GPUs can be found on the following website:

<http://developer.nvidia.com/cuda-gpus>

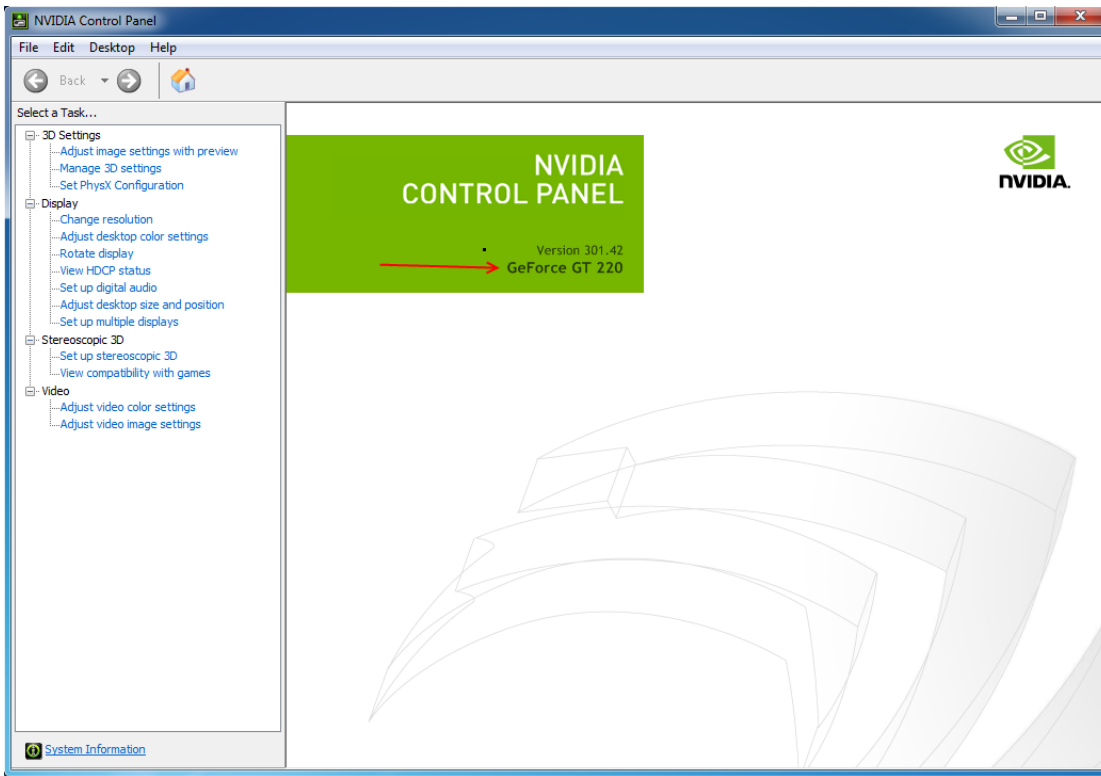
To check if your computer has an NVIDIA GPU and if it is CUDA enabled:

1. Right click on the Windows desktop
2. If you see “NVIDIA Control Panel” or “NVIDIA Display” in the pop up dialogue, the computer has an NVIDIA GPU
3. Click on “NVIDIA Control Panel” or “NVIDIA Display” in the pop up dialogue
4. The GPU model should be displayed in the graphics card information
5. Check to see if the graphics card is listed on the following website: <http://developer.nvidia.com/cuda-gpus>

The following screen images show the steps outlined above, this may vary slightly between NVIDIA card models.




**Screenshot: Accessing NVIDIA Control Panel from the desktop**



**Screenshot: NVIDIA GPU Model**

### CUDA-Enabled GeForce Products

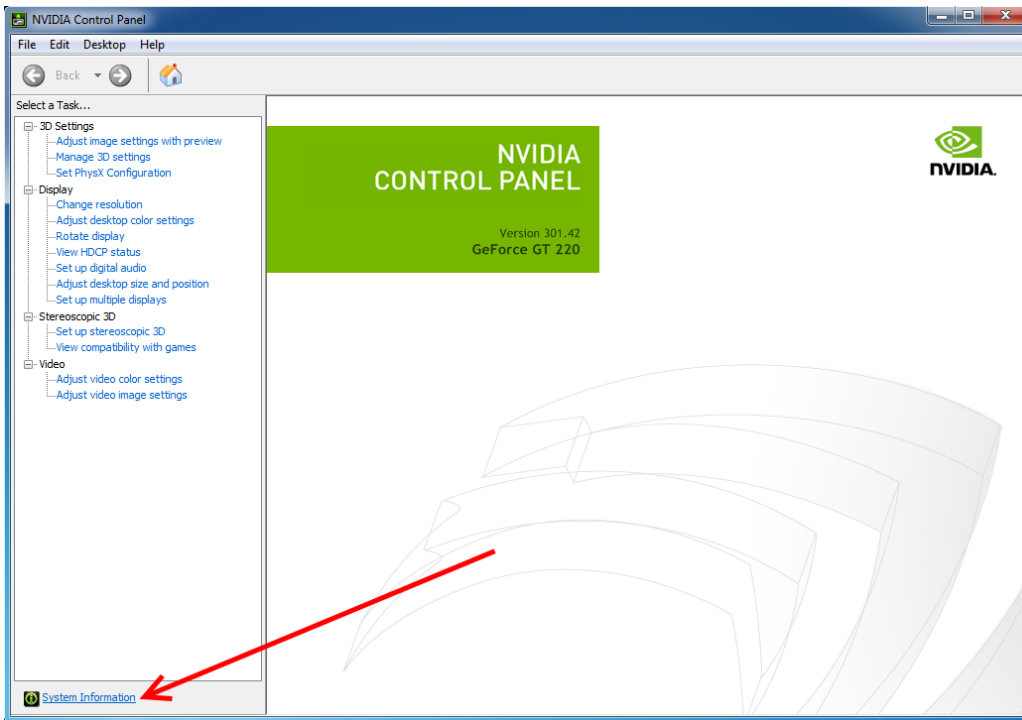
GeForce 8, 9, 100, 200, 400-series, 500-series, and 600-series GPUs with a minimum of 256MB of local graphics memory.

	GeForce Desktop Products		GeForce Notebook Products	
	GPU	Compute Capability	GPU	Compute Capability
	GeForce GTX 690	3.0	GeForce GTX 680M	3.0
	GeForce GTX 680	3.0	GeForce GTX 675M	2.1
	GeForce GTX 670	3.0	GeForce GTX 670M	2.1
	GeForce GTX 560 Ti	2.1	GeForce GTX 660M	3.0
	GeForce GTX 550 Ti	2.1	GeForce GT 650M	3.0
	GeForce GTX 460	2.1	GeForce GT 640M	3.0
	GeForce GTS 450	2.1	GeForce GT 640M LE	3.0
	GeForce GTS 450*	2.1	GeForce GT 635M	2.1
	GeForce GTX 590	2.0	GeForce GT 630M	2.1
	GeForce GTX 580	2.0	GeForce GT 620M	2.1
	GeForce GTX 570	2.0	GeForce 610M	2.1
	GeForce GTX 480	2.0	GeForce GTX 580M	2.1
	GeForce GTX 470	2.0	GeForce GTX 570M	2.1
	GeForce GTX 465	2.0	GeForce GTX 560M	2.1
	GeForce GTX 295	1.3	GeForce GT 555M	2.1
	GeForce GTX 285	1.3	GeForce GT 550M	2.1

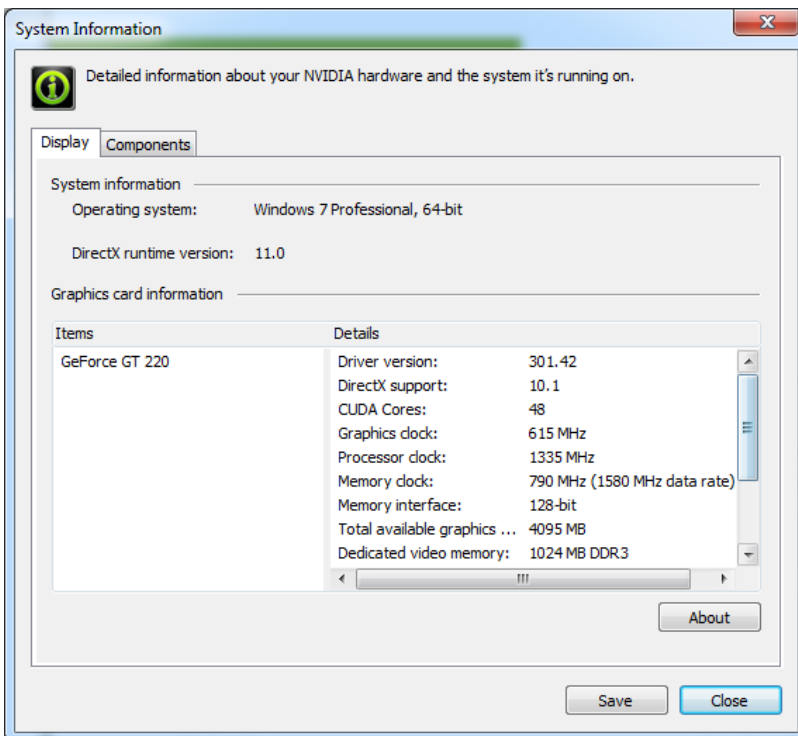
**Screenshot: Check the Website for your NVIDIA Card**

More information on the card can be found in the “System Information” section, which is accessed from the NVIDIA Control Panel. The system information contains more details on the following:

- The number of CUDA cores
- Frequency of the graphics, processors and memory
- Available memory including dedicated graphics and shared memory.



**Screenshot: Accessing System Information from NVIDIA Control Panel**



**Screenshot: NVIDIA System Information**

## Compute Capability

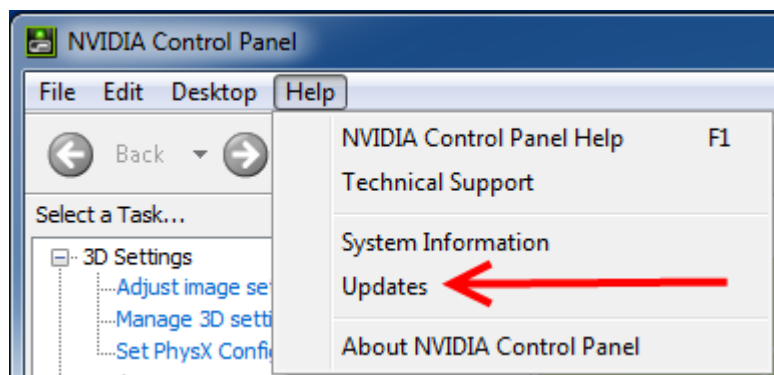
On the NVIDIA website each CUDA enabled graphics card has a “Compute Capability” listed. For cards with a compute capability of 1.2 or less, only the single precision version of the GPU Module can be utilised. **However, benchmarking has indicated that the double precision version is NOT required and that for all GPU simulations TUFLOW\_iSP exes should be used.**

## Updating NVIDIA Drivers

It is likely that the NVIDIA drivers will need to be updated to the latest version as the drivers shipped with the computers are usually outdated. To update, open the NVIDIA Control Panel (by right clicking on the desktop and selecting NVIDIA Control Panel or NVIDIA Display). Once the control panel has loaded, select Help >> Updates from the menu items.

If new drivers are available, please download and install these by following the prompts.

**NOTE:** Even if not prompted by the system, a restart is recommended to ensure the new drivers are correctly detected prior to running any simulations.



Screenshot: Accessing Driver Updates from NVIDIA Control Panel

## Running TUFLOW GPU

When running the GPU solver in TUFLOW, the pre (reading of data) and post (writing outputs) is managed by the standard TUFLOW engine. This allows the user to utilise the extensive range of GIS input functionality in TUFLOW.

To simulate the model via the GPU solver the command “GPU Solver == ON” is required in the .tcf. The TUFLOW GPU simulation is then started in the same manner as a standard TUFLOW simulation. This can be done via batch file, right click functionality in Windows or from a text editor.

The Ctrl-C option is also available if you wish to terminate the simulation early, but note that TUFLOW won't stop until the next map output as this is when it communicates with the GPU.

A list of available control file commands is given in the section below.

## TUFLOW Control File Commands

The following .tcf commands are specific to the TUFLOW GPU solver:

**GPU Solver == ON**

This is required to initialise the GPU solver in TUFLOW. If this command is not present the standard CPU solution is used.

**GPU Temporal Scheme == [ 1 | 2 | {4} ]**

Sets the order of the temporal solution. The default is a 4<sup>th</sup> order temporal solution and that recommended (therefore this command is usually not specified). Available options are:

- 1 – First order out of place
- 2 – Second Order
- 4 – Fourth Order

**We recommend the use of the 4<sup>th</sup> order temporal scheme as it is unconditionally stable with adaptive time-stepping turned on, and has found to give excellent results. Lower order schemes save a little on memory requirements, but are more prone to instability and in some cases unreliable results.**

**GPU Device IDs == <list\_of\_device\_ids>**

Controls the GPU device or devices to be used for the simulation if multiple CUDA enabled GPU cards are available in the computer or on the GPU itself. We're looking into the option of automatically assigning to the least used GPU to bypass this step. If you only have one GPU device, or you wish to use the primary device, this command is not needed. If there is more than one GPU device, and you wish to run the model across cards, enter a list of device IDs. For example, if you wanted to run a model using GPU devices 0 and 2, use “GPU Device IDs == 0, 2”. A GPU licence will be needed for each device ID.



**Maximum Courant Number == {1.0}**

This option when greater than zero switches on the adaptive time-stepping. For the 4<sup>th</sup> order time solver a value of 1.0 (the default) is recommended. If set to 0 adaptive time-stepping is turned off and the fixed timestep is used. The recommendation is not to specify this command and use the default settings. If the user wishes to use reduced time step control numbers (discussed in the section “Stability and Adaptive Timestepping” above, and is useful for checking if a model is time-step converged) then the scaling factor may be entered here.

**Timestep == <timestep\_sec>**

If adaptive time-stepping is off, sets the fixed timestep (see above). The timestep will always be much smaller than TUFLOW Classic’s timesteps as the scheme is explicit (TUFLOW uses an implicit scheme). As a general rule of thumb specify a timestep that is around one tenth of the TUFLOW timestep you would use.

If adaptive time-stepping is active (the default), the timestep value is only used for the very first step. To allow the same command to be used for either a TUFLOW Classic or a GPU simulation the timestep value is divided by 10 for the initial GPU timestep, therefore, enter a timestep value similar to that that you would use for TUFLOW.

## Cell Codes

Active (Code 1) and inactive cells (Code 0) are assigned cell codes in the same manner as for TUFLOW.

## Topography

The GPU solver has only one elevation per cell (ZC). Therefore, thin Z lines are automatically treated as thick Z lines if using the GPU solver. TUFLOW’s topographic manipulation commands are available for use on GPU models keeping in mind that only functionality that modifies the ZC values is applicable. As with all models, check that the final topography is as expected. Flow constrictions (FC) and layered flow constrictions (LFC) are not as yet utilised by the GPU solver and should not be used. As of Build 2013-12-AB, form loss coefficients (FLC), storage reduction factors (SRF) and cell flow width factors (CWF) are now supported.

## Bed Resistance

Manning’s n values are specified in the same manner as for TUFLOW. All materials and Manning’s n functionality is available for use with the GPU solver including depth-varying Manning’s n curves.

## Initial Water Levels

Initial water levels (IWL) can be applied in the same manner as for TUFLOW.

## Boundary Conditions

Currently supported boundary conditions are water levels (2D HT), and source based inflows with some restrictions (ie. 2D SA, ST and RF inflows). Until 1D links are incorporated, 2D QT boundaries (which utilise a hidden 1D node) are not supported.

A water level boundary is required for water to exit the model. If the water level boundary level lies below 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 1 Available Boundary Types**

Boundary Type	TUFLOW Layer/Command	Notes
Water Level	2d_bc (HT)	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.
Source Inflows	2d_sa 2d_rf 2d_bc (ST) Global Rainfall == (.tbc commands)	All SAs are treated as either SA ALL, or if the new "Read GIS Streams ==" command has been specified, SA's by default are applied as SA STREAM ONLY (see the description on the new streamlines feature at the start of this document).  TUFLOW Classics ability to distribute the flow over the wet cells and weight the flow according to depth are being investigated for implementation in the GPU solver.

To keep the memory footprint minimal, there is a limit on the number of different source inflows each cell can have. Any number of source inflows can be specified for the model, but a single cell can only have up to four (4) different source inflows. Each inflow can also be weighted, for example, to distribute rainfall hyetographs differently to each cell, through TUFLOW's 2d\_rf f1 and f2 attributes.

Note that the IL and CL values in the materials file are NOT yet used by the TUFLOW GPU solver for direct rainfall modelling, primarily to keep the memory footprint small (GPUs have small memory capacity compared with CPUs). An IL/CL loss can be modelled using the IL/CL infiltration option, which is the preferred approach, as mentioned below and documented in the TUFLOW release notes for 2011-09/2012-05 or the 2013 manual once available. If not applying infiltration the rainfall applied must therefore be excess rainfall.

## Infiltration

The GPU solver includes the IL/CL and Green-Ampt infiltration options in Build 2013-12-AA, with Horton added for Build 2013-12-AB. If using US Customary (English) Units, Build 2013-12-AB must be used due to bug fixes.

All TUFLOW Classic's infiltration inputs and outputs are valid and can be used with the GPU solver.

## Output

The GPU solver solves for water level and velocities at the cell centres. Therefore, like TUFLOW, to output to the SMS .dat or .xmdf formats, the water levels and velocities must be interpolated or extrapolated to the cell corners.

For large models (>1,000,000 cells), using SMS or other post-processors maybe slow, and GIS outputs are often the only way to manage the outputs from large GPU models. At present the GRID map output format is the most effective way of handling large models, especially the new .flt binary format introduced for Build 2013-12-AB. Other formats including the new ones should be trialled as to whether practical or not. For the latest options on map output and also the new Output Zones feature, which are all available to the GPU solver, see the discussion in this document.

## Time-Series Output (2d\_po)

2D plot output (PO) can be extracted by the GPU during a simulation for water levels (H\_) and flows (Q\_) in the same manner as for TUFLOW. Water levels should be point objects and flows polyline objects.

## Excluded TUFLOW Functionality

A range of TUFLOW's features are not utilised by the GPU solver. As of Build 2013-12-AB, form loss coefficients (FLC), storage reduction factors (SRF) and cell flow width factors (CWF) are supported. However, other TUFLOW features including flow constrictions (FC) and layered flow constrictions (LFC), weir factors (WrF), evacuation routes are not yet incorporated. If you are unsure, please email [support@tuflow.com](mailto:support@tuflow.com).

## Single and Double Precision

TUFLOW GPU is available in both single precision and double precision. If the simulation is started with the single precision version of TUFLOW, the GPU solver will utilise a single precision version. The calculation method in the GPU due to its explicit nature uses depth unlike TUFLOW that uses water level due to its implicit scheme. This means that precision issues associated with applying a very small rainfall to a high water level are not applicable.

Unless testing shows otherwise, use the single precision version of TUFLOW for all TUFLOW GPU runs. Also, double precision solutions on GPU cards can be four times slower than single precision!

NVIDIA Cards with a compute capability of 1.2 or less are only able to run single precision versions.

As of Build 2013-12-AC, to run GPU in DP "GPU DP Check == OFF" will need to be specified in the .tcf file (see Item7474(a)), otherwise ERROR 2420 occurs.

## Troubleshooting

If you receive the following error when trying to run the TUFLOW GPU model:

```
TUFLOW GPU: Interrogating CUDA enabled GPUs ...
```

```
TUFLOW GPU: Error: Non-CUDA Success Code returned
```

Please try the following steps:

1. Check the compatibility of your card and the latest drivers are installed (see instructions above).
2. Test with a user account that has administrator privileges as these may be required for running computations on the GPU.
3. If multiple monitors are running from the video card, try running with only a single monitor.

If the above steps fail to get the simulation to run, please email the NVIDIA system information (see instructions above) and TUFLOW log file (.tif) to [support@tuflow.com](mailto:support@tuflow.com).

## FAQ

As the GPU solver is relatively new, there is not a lot (yet) in this section, but the following may be of use:

**Question:** Can you supply some statistics for run times based on the trials you have done? Do you have a calculation spreadsheet for the GPU, similar to the one for the classic TUFLOW engine?

**Answer:** Based on a 4GB NVIDIA GTX680 (one of the better ones), the run times are in the order of 0.012 s per time step per million cells. The model time step is typically  $1.0 \cdot dx / U_{max}$ . For example, on a 20 m grid with maximum velocities of 10 m/s, the model time step is about 2.0s.

Run times for this model are therefore about 22 seconds per million cells for one hour model time.

**Question:** Can you confirm there is a 100M cell limit and what are the RAM requirements for such large models?

**Answer:** Without infiltration, you can model about 15 million cells per GB of GPU RAM, with infiltration it is about 12 million cells per GB. A card with 6 GB of RAM allows about 75 million cells. However, as the pre and post processing is handled by the TUFLOW engine, such a model would also require significant amounts of motherboard RAM as well. You can also run models across multiple GPU cards allowing for even larger models to be simulated. For example it is possible to run 180 million cells with infiltration losses over four GTX680 cards (4 GB each).