TUFLOW 2011-09 and 2012-05 Release Notes

These notes for the TUFLOW 2011-09 and 2012-05 releases supplement the 2010-10 TUFLOW manual until the 2012 version of the manual is available later in the year. For clarification or more detail on any point below, please email support@tuflow.com.

(For those familiar with the 2011-09 release notes, these have been reproduced here for completeness, but in chronological order not reverse chronological order as previously presented. This is so that we can refer to numbered items in our support and on the forum without the numbers changing when this document is updated. The alternative was to have Microsoft add a reverse numbering feature into Word, but decided not to pursue this option!)

Table of Contents

2011-09 Release Summary

The 2011-09 release includes a major re-write of the GIS routines so to provide full support for .shp layers and be able to easily incorporate other formats into the future. It was times to coincide with:

- The ISIS update supporting the release of ISIS-TUFLOW-PIPE (ISIS-ESTRY link).

- The Tutorial Model Wiki with full support for GIS platforms that use .shp files (eg. ArcGIS and QGIS).

The default settings were left unchanged from the 2010-10 release, so the results from simulations carried out using 2010-10 builds should be identical or near identical with those using 2011-09.

2012-05 Release Summary

The 2012-05 release includes a range of new features, some nice enhancements to the 2D solver, and bug fixes. The release has been extensively tested internally and externally, and to the UK EA 2D Benchmark models. It has consistently shown performance improvements to all types of models, especially simulations that utilise direct rainfall, dambreak type inflows and simulations utilising the automatic manhole feature. The changes include:

- Enhanced wetting algorithm that provides significant improvement to inflows on steep areas (eg. direct rainfall models), whilst maintaining low mass error, greater stability and often larger timesteps, without needing to use TUFLOW's shallow depth stability and mass balance corrector features.

- Enhanced treatment of the 2D solution when transitioning between downstream and upstream controlled 2D flow regimes, resulting in improved stability and lower mass error.

- Enhancements and minor bug fixes in the automatic sizing and treatment of manholes.

- Full support for US Customary (English) Units for the USA market.

- Range of minor enhancements, new features such as infiltration, and bug fixes.

The default settings have been set to utilise the enhancements, therefore, some differences in results will occur compared with previous releases. A backward compatibility command is provided.

Note: Build 2012-05-AD fixes a bug in Builds 2012-05-AA to AC relating to manhole losses using the Engelhund approach – please see Item 120.

Note: Build 2012-05-AE fixes a bug relating to the ESTRY-ISIS link available via the ISIS-TUFLOW-PIPE software – please see Item 125.

General Note: It is always recommended that if using 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.

Build 2011-09-AA

Notes on the new features available in Build 2011-09-AA are:

1 **ISIS-ESTRY Link (ISIS-TUFLOW-PIPE):** ISIS and the 1D solution of TUFLOW (ESTRY) can now be linked in a similar manner to linking ISIS with TUFLOW 2D domains. The main driver for this feature is for ISIS-TUFLOW models to utilise the powerful pipe network and manhole modelling capabilities of TUFLOW and be able to link these networks into an ISIS river model.

Notes on the ISIS-ESTRY link are:

  (a) ISIS and ESTRY nodes will be considered as linked if:

    (i) An ESTRY node in a 1d_nwk layer, and an ISIS node in a "Read GIS ISIS Nodes ==" or "Read GIS ISIS Network ==" layer are snapped.

    (ii) The ESTRY node has a Conn_2D attribute (which has been renamed for 2011-09 empty layers as Conn_1D_2D) of either "X1DH" or "X1DQ".

    (iii) If Conn_1D_2D is blank then "X1DH" is assumed.

    (iv) A connector "X" channel type can be used to connect the end of the linked ESTRY channel to the ESTRY node snapped to the ISIS node if the end of the ESTRY channel and the snapped ISIS/ESTRY nodes are not in the same location.

    (v) Note that the upstream and downstream Inverts for the ESTRY node linked to ISIS should be set to -99999 unless the node is also being used to set the inverts of channels snapped to it.

  (b) An "X1DH" link means an ISIS 1D water level is being applied at the ESTRY node, ie. ISIS sends ESTRY a water level and ESTRY sends back a +/- flow to ISIS.

  (c) An "X1DQ" link means an ISIS in/outflow is being applied at the ESTRY node, ie. ISIS sends ESTRY a +/- flow and ESTRY sends back a water level.

  (d) For most ISIS-ESTRY links an ESTRY X1DH (the default) would be used. An X1DQ might be more appropriate where an ISIS model stops and flows into an ESTRY model.

  (e) Other Notes:

    (i) The mass balance _MB1D.csv file includes four new columns:

      • X1DH V In: The volume of water in via a X1DH link.

      • X1DH V Out: The volume of water out via a X1DH link.

      • X1DQ V Out: The volume of water out via a X1DQ link.

      • X1DQ V In: The volume of water in via a X1DQ link.

    (ii) The type or existence of a connection can be checked by viewing the Conn_1D_2D attribute in the 1d_nwk_N_check layer.

    (iii) The _messages.mif layer contains CHECK 1393 messages at each ESTRY node linked to an ISIS node.

TUFLOW

2   **Full .shp File Support:** All TUFLOW GIS outputs (eg. check layers) can now be in .shp file format making the use of ArcGIS much more streamlined, and products such as SAGA that don't recognise the .mif format a good option for using TUFLOW. There are number of new commands associated with this major feature:

(a)   SHP Projection == <.prj_file>. This .tcf command is similar to the MI Projection command that sets the .shp file projection for checking whether input layers are in the same projection, and for setting the projection of all output layers (eg. check layers). An example is:

SHP Projection == ..\model\shp\Projection.prj

If a model has a mixture of .mif and .shp files as input, then both MI Projection and SHP Projection should be specified.

(b)   Empty .shp files can now be created using the command below. If the second argument is MIF or is omitted, the empty files are in the .mif format. As with writing empty .mif files, ensure that you've specified the projection to be used using the MI Projection or SHP Projection command as described above prior to the command below.

Write Empty GIS Files == ..\model\shp\empty | SHP

(c)   The default output format for GIS check layers and GIS outputs such as the _TS layers is the .mif format. To produce check and output GIS layers as .shp files specify the following command in the .tcf file.

GIS Format == SHP

Note that the format of an input layer is solely controlled by the file extension (ie. .mif for the MIF format and .shp for the SHP format).

3   **Reworking of GIS Routines:** Please note that the 2011-09 release includes a complete reworking of all GIS TUFLOW routines so that irrespective of the GIS format the same routines are used to read and write GIS layers. This means that all the old .mif routines have been replaced (major undertaking!) by the new ones. Whilst we have carried out major testing of both .mif and .shp files for input and output, please don't hesitate to contact support@tuflow.com should any problems or issues arise, even if using the traditional .mif format.

The new routines, whilst a major exercise, now allow the easy inclusion of other GIS formats. We would be pleased to hear of any other open GIS formats that TUFLOW modellers would like to have incorporated. Please send suggestions to support@tuflow.com.

4   **Ignoring of Redundancy Around Model:** TUFLOW now automatically strips any redundant rows/columns around the active area of the model to reduce simulation times. Whilst simulation times are largely dependent on the number of wet cells, a model with a large amount of superfluous space around the outside can run significantly slower than if the grid was reduced to the minimum needed. In one extreme case where a small sub-part of a model was being run on its own, the simulation time was

reduced by a factor of 15 by this new feature.

Note, the amount of RAM consumed by the model is not reduced, only the computational effort in some of the TUFLOW routines that need to scan the whole model.

5   **Nested If Scenarios:** The powerful If Scenario command can now be nested up to 10 levels (if you need more than 10, or 2 for that matter, you may do the head in of your reviewer...). The extract below from a .tcf file of a super model is shown below. The model is actually a combination of models that can be simulated at different resolutions and in different configurations depending on the simulations objective. Out of interest, the same .tcf file is also used to simulate all historical and design flood events.

The excerpt below shows the Casino sub-model being able to be run at either a 20m or 60m resolution using the same .tcf file. The logic as to which other sub-models/boundaries the Casino sub-model needs is built into the .tbc file using nested If Scenarios.

```
####################
### CASINO DOMAIN ###
####################

IF SCENARIO == CAS

    ESTRY Control File == ..\model_CAS\CAS_001.ecf

    ! Cell size dependent commands
    IF SCENARIO == 20m  ### 20m grid Domain
       START 2D domain == CAS_20m
          Geometry Control File == ..\model_CAS\CAS_20m_001.tgc
          Timestep == 7.5
       END 2D Domain
    ELSE IF SCENARIO == 60m  ### 60m grid Domain
       START 2D domain == CAS_60m
          Geometry Control File == ..\model_CAS\CAS_60m_001.tgc
          Timestep == 20
       END 2D Domain
    ELSE
       Pause == Should not be here - invalid CAS cell size scenario specified.
    END IF

    ! Cell size independent commands
    BC Control File == ..\model_CAS\CAS_001.tbc
    Read GIS IWL == ..\model_CAS\2D_iwl\2D_iwl_CAS_001.mif

END IF
```

6 **Pause command:** The new Pause command causes TUFLOW to stop whenever it encounters it. In the example above, the Pause command is used to pause the simulation and display the message shown as a cross-check that an appropriate cell size scenario has been specified. The user has the option to continue or discontinue the simulation via a dialog window.

7 **New SA Commands:**

(a) New .tcf command "SA Minimum Depth == <depth_metres>" to set the minimum depth a wet cell must have to apply an SA inflow. This can resolve a problem that has occurred where large SA inflows onto very shallow, high roughness areas can appear to gradually flow up hill. This was being caused by the SA inflow being greater than the rate at which the flow was travelling overland and the water would slowly creep up the dry slope at the edge of the flooded area. Using a SA Minimum Depth of around 0.1m ensures that this does not occur. Note that the only cases this problem has been seen to occur was when modelling an extreme flood event (PMF) on gently sloping high roughness areas.

(b) New .tcf command "SA Proportion to Depth == [ON | {OFF}]" to proportion SA inflows according to the depth of water. This feature also enhances SA inflows by applying an inflow in proportion to the depths of water of the wet cells contained within the SA polygon. Where the SA hydrographs have been derived by a hydrologic model that has already included an routing effects, this feature will tend to place more inflows in the deeper areas (ie. the creeks, rivers and downstream areas of the SA region), and hence reduce any routing duplication effects.

(c) The default setting for 2011-09 is not to use the above commands, however, it is likely that a future release will set these as the default.

Build 2011-09-AB

Notes on new features and bug fixes in Build 2011-09-AB:

8 **(Note: for the 2012-05 release, it is recommended to initially not use this feature – see Point 47.)**

Mass Balance Corrector: A nice new feature that carries out an additional iteration of the mass balance equation every half timestep is being made available. By switching this feature on using the command below, significant reductions in mass error can result for problematic models, particularly those with steep and/or very shallow flow. Improvements in model stability and "smoothness" also occur, with larger timesteps being feasible in many cases. So far testing on a range of models indicates changes in peak water levels vary from a few mm to a few cm if this feature is used. This feature is being released for testing in the wider TUFLOW community, and all going well, will likely become the default in the next major release (any feedback, good or bad, would be appreciated by emailing support@tuflow.com). The downside to this feature is that it will add around 5 to 10% to your simulation time, although this may easily be recovered if a larger timestep can be used. To switch this feature on, use "Mass Balance Corrector == ON " in the .tcf file.

9 Build 2011-09-AA will produce the error message "ERROR 0251 - Cannot have more than 10 GIS layers open at once." for some models. This limit is a hangover from the 2010-10 release which only applied to input layers. With the reworking of GIS routines to write output and check .shp layers, this limit can be

exceeded in some cases. The limit has been increased to 100 which hopefully should be ample (let us know if it isn't!). TUFLOW also now checks for any unclosed layers at the end of the simulation and closes them off neatly. If an unclosed layer exists "WARNING 0255 - One or more GIS layers not closed during simulation - please notify support@tuflow.com." occurs. This does not affect the simulation or GIS layers, but if you see this message please email support@tuflow.com your .tlf file.

10   A bug in Build 2011-09-AA that causes TUFLOW to stop in some instances where a nested If Scenario occurs after an Else command has been fixed.

Build 2011-09-AC

Notes on bug fixes in Build 2011-09-AC:

11   x1d_channels_check layer now written in .mif format correctly. This bug does not occur in releases prior to 2011-09.

12   1d_to_2d_check layer now exports SX cells correctly in .mif format. This bug does not occur in releases prior to 2011-09.

13   Excessive whitespace in check layers for the .mif format after Style lines has been removed – check files are now back to a similar size as per releases prior to 2011-09..

Build 2011-09-AD

Notes on bug fixes in Build 2011-09-AD:

14   TUFLOW now checks the date of .xf files with their original .tab layers. For all prior builds, including prior to 2011-09, that include the XF file feature, the only date comparison was with the .mid file. This meant that if a 2d_zpt.tab layer was updated and saved but **not** exported to .mif/.mid, TUFLOW would not issue an ERROR or WARNING 0004 if an XF file existed from a previous run. The Ignore attribute in the 1d_nwk and 1d_mh layers is now treated as a Character of length 1 (previously set as a Logical) as some GIS software have problems with Logical attributes in the .shp format.

15   The Ignore attribute in the 1d_nwk and 1d_mh layers is now treated as a Character of length 1 (previously set as a Logical) as some GIS software have problems with Logical attributes in the .shp format. Note, there is no need to change the Ignore attribute in 1d_nwk and 1d_mh MapInfo layers from Logical to Character, as TUFLOW recognises either specification. For .shp files, it is recommended that a Character of length 1 is used. Regenerating empty layers will now set the Ignore attribute as a Character. For the Ignore attribute, if a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.

16   Import of _TS.mif layers into MapInfo if 2D PO lines are present now works. This bug does not occur in releases prior to 2011-09.

17   The _messages.shp layer and any unclosed .shp check layers are now closed off neatly if TUFLOW stops with an ERROR message during the input phase. For prior builds these layers could not be opened in a GIS.

TUFLOW

18 2d_zsh layers with aggregated polylines cause an ERROR 0308 if writing check files is switched on. This bug does not occur in releases prior to 2011-09.

19 Fixed bug that didn't correctly set the wet/dry flags for the maximum and /or minimum map output if using the SMS High Res output option in combination with a Map Cutoff Depth == greater than zero.

20 Fixed bug that did not copy a file when using –c or –ca if the path to the file included a forward slash. TUFLOW is happy to accept forward slashes in pathnames, but when copying a model these files were omitted.

21 Fixed bug that when copying a model using –c or –ca the existing .xmdf file was deleted (and therefore the results for that simulation were lost!). This bug does not occur for .dat files.

22 If one or more null cells are located outside the reduced computational domain used by the 2011-09 release for minimising run times, the .2dm file would contain nodes with zero ID in the .2dm mesh file – this is now fixed. Note that the outlying null cells are included in the mesh as would be the case with releases prior to 2011-09.

Build 2011-09-AE

Notes on new features, enhancements, and bug fixes in Build 2011-09-AE:

New Features

23 New Storage Reduction Feature (SRF).

(a) The storage of 2D cells may now be reduced (for example to model hypothetical filling, or reduced storage from buildings), or increased. For example, if a cell has a SRF value of 0.1, then its storage (surface area) is reduced by 10%. If the SRF value is less than zero, the storage is increased. The default SRF value is, of course, zero, ie. no change in storage.

(b) SRF values are assigned to cells in one or both of the following ways:

(i) Using the new .tgc "Set SRF ==" and "Read GIS SRF ==" commands. The 2d_srf layer has only one attribute being a float or real value nominally called SRF.

(ii) Assigned to materials as follows.

- For .tmf files, the optional SRF value is the 10th number. If no 10th number exists, no SRF is applied. For unrequired intermediate values (eg. if varying n with depth or IL and CL values are not needed), these may now be entered as blanks separated by commas. For example, a line in the .tmf file for Material ID 10, that only sets the Manning's n and SRF values would be:

  10, 0.050, , , , , , , , 0.1 ! future rural residential with 10% loss of storage for house pads

- For material .csv files, the SRF value goes into the 5th column as a single number. If no 5th column entry exists, no SRF is applied.

(iii) You can check the SRF value applied by viewing the SRF attribute in the 2d_grd_check layer.

(iv) Note that a user can use a combination of material SRF values and "Read GIS SRF" layers, but the latter prevails. The order is dependent on that in the .tgc file. For example, if a cell's storage is adjusted by both "Read GIS SRF ==" and "Read GIS Mat ==" layers, the latter will prevail.

24 Hazard (Velocity times Depth) or Velocity can now be used for the cutoff values for evacuation routes. To set up:

(a) For VxD: For the Cut_Off_Type attribute specify one of "VxD", "Z0" or "Hazard"

(b) For Velocity: Specify one of "V" or "Velocity"

(c) For Depth (the default) you can optionally specify "Depth"

(d) More than one 2d_zshr layer may be used if different cutoff types are required.

25 People Hazard categories based on the Australian Rainfall and Runoff (ARR) Project 10 Stage One report, published by Engineers Australia, are now available via the "Map Output Data Types ==" command. The acronyms are:

(a) ZPA: Hazard to Adults

(b) ZPC: Hazard to Children

(c) ZPI: Hazard to Infants and frail/elderly people.

The values in the ZPA, ZPC and ZPI output are (see table and figure further below):

0 = Safe (no hazard)
1 = Low Hazard
2 = Moderate Hazard
3 = Significant Hazard
4 = Extreme Hazard

There is also a new .tcf command "ZP Hazard Cutoff Depth ==". If one value is specified, the cutoff depth to define when the Safe category applies is the same for ZPA, ZPC and ZPI. If three values are specified, these are the cutoff depths for ZPA, ZPC and ZPI respectively. The default is "ZP Hazard Cutoff Depth == 0.01", ie. if the depth is below 0.01m (1cm), the hazard category is Safe for ZPA, ZPC and ZPI.

The Engineers Australia report may be downloaded from http://www.tuflow.com/Download/Publications/2010.04 - ARR_Project_10_Stage1_report_Final.pdf and relevant tables and figures are reproduced below.
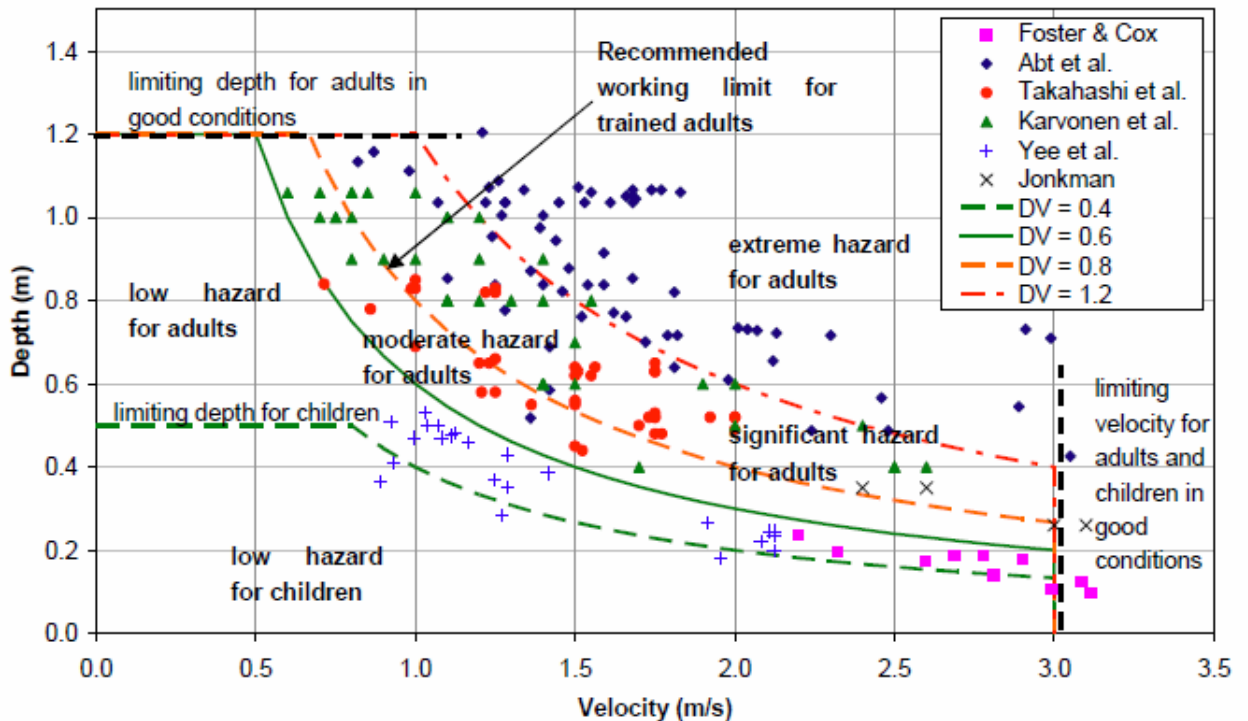
Table 5 Flow hazard regimes for infants, children and adults

DV (m^2s^{-1})	Infants, small children (H.M ≤ 25) and frail/older persons	Children (H.M = 25 to 50)	Adults (H.M > 50)
0	Safe	Safe	Safe
0 – 0.4	Extreme Hazard; Dangerous to all	Low Hazard[1]	Low Hazard[1]
0.4 – 0.6		Significant Hazard; Dangerous to most	
0.6 – 0.8		Extreme Hazard; Dangerous to all	Moderate Hazard; Dangerous to some[2]
0.8 – 1.2			Significant Hazard; Dangerous to most[3]
> 1.2			Extreme Hazard; Dangerous to all

[1] Stability uncompromised for persons within laboratory testing program at these flows (to maximum flow depth of 0.5 m for children and 1.2 m for adults and a maximum velocity of 3.0 ms^{-1} at shallow depths).

[2] Working limit for trained safety workers or experienced and well equipped persons (D.V < 0.8 m^2s^{-1})

[3] Upper limit of stability observed during most investigations (D.V > 1.2 m^2s^{-1})

Figure 8 Proposed hazard regimes as a function of depth and velocity and compared to available experimental data.

TUFLOW

26 Variable Z Shapes can now be restored once or repeatedly. Examples would be a breach of a flood defence wall or levee that is reinstated 6 hours later, or a sand bar of a creek entrance that repeatedly opens up and closes. To use this new feature:

(a) Add two more attributes to the 2d_vzsh layer as follows:

  (i) Restore_Interval, Float/Real value equal to the time in hours between when the variable Z shape as finished and when to start restoring the Zpts back to their original values.

  (ii) Restore_Period, Float/Real value equal to the time in hours needed to restore the Zpts back to their original values.

(b) If the above attributes are not added (ie. there is less than 10 attributes), or Restore_Period is zero or negative, there will be no restoration of the Z shape.

(c) If the keyword "REPEAT" is included in the "Shape_Options" attribute, the variable Z shape will repeatedly function indefinitely.

(d) Note: If two or more additional attributes have been added to a 2d_vzsh layer for the user's own reference, then the above attributes will need to be inserted otherwise the user's attributes will be interpreted as the ones above.

27 Tracking of maximums and minimums every timestep for PO and LP output is now available.

(a) To activate this feature, enter "Maximums and Minimums Time Series == ON" in the .tcf file.

(b) If activated, four more rows are added near the top of the _PO.csv file, and columns in the _LP.csv files, containing the Maximum, Time of Maximum, Minimum, and Time of Minimum values.

(c) The _TS.mif layer also contains the tracked values. Note, the max/min values in the _TS.mif layer if this feature is not activated are not tracekd every timestep, but are the max/min values based on the "Time Series Output Interval".

(d) This feature will most likely be made the default setting in the next major release as part of a general reworking of TUFLOW output for the new User Defined TUFLOW Output feature.

28 New SA flow feature to model seepage or infiltration based on a varying water level or flow rate elsewhere in the model. This feature was recently used to model the seepage of groundwater into a coastal lagoon that was dependent on the water level in the lagoon as observed from long-term historical measurements. Notes on this feature are:

(a) For example, to set the SA flow in/out of a model based on the water level somewhere in the model:

  (i) Create a 2d_po point object of Type "H_" at the location of the water level to be used to set the SA in/outflow.

  (ii) Add "Read GIS PO == …." to your .tcf file if not already there.

  (iii) Create a new 2d_sa layer (call it 2d_sapo) and add two attributes as follows:
    - PO_Type   Char of length 16
    - PO_Label Char of max length 40

  (iv) Digitise your SA polygon(s) covering the area of seepage or infiltration and for the attributes:
    - Set the Name attribute to the name of the water level vs flow curve in the BC database.

- Set PO_Type to "H_".

- Set PO_Label to the PO Label of the relevant 2d_po "H_" point to be used to determine the flow from the h vs Q curve.

(v) Add "Read GIS SA PO == …" to your .tbc file.

(b) To base the SA flow on the flow elsewhere in the model, use a Q_ PO type.

(c) To check the SA in/outflow:

(i) View the _MB.csv files. The SA in/outflow from the seepage or infiltration will be part/all of the SS columns.

(ii) Add "SS" to the "Map Output Data Types ==" command. This outputs over time the net in/outflow from all source flows (ST, SA, SX).
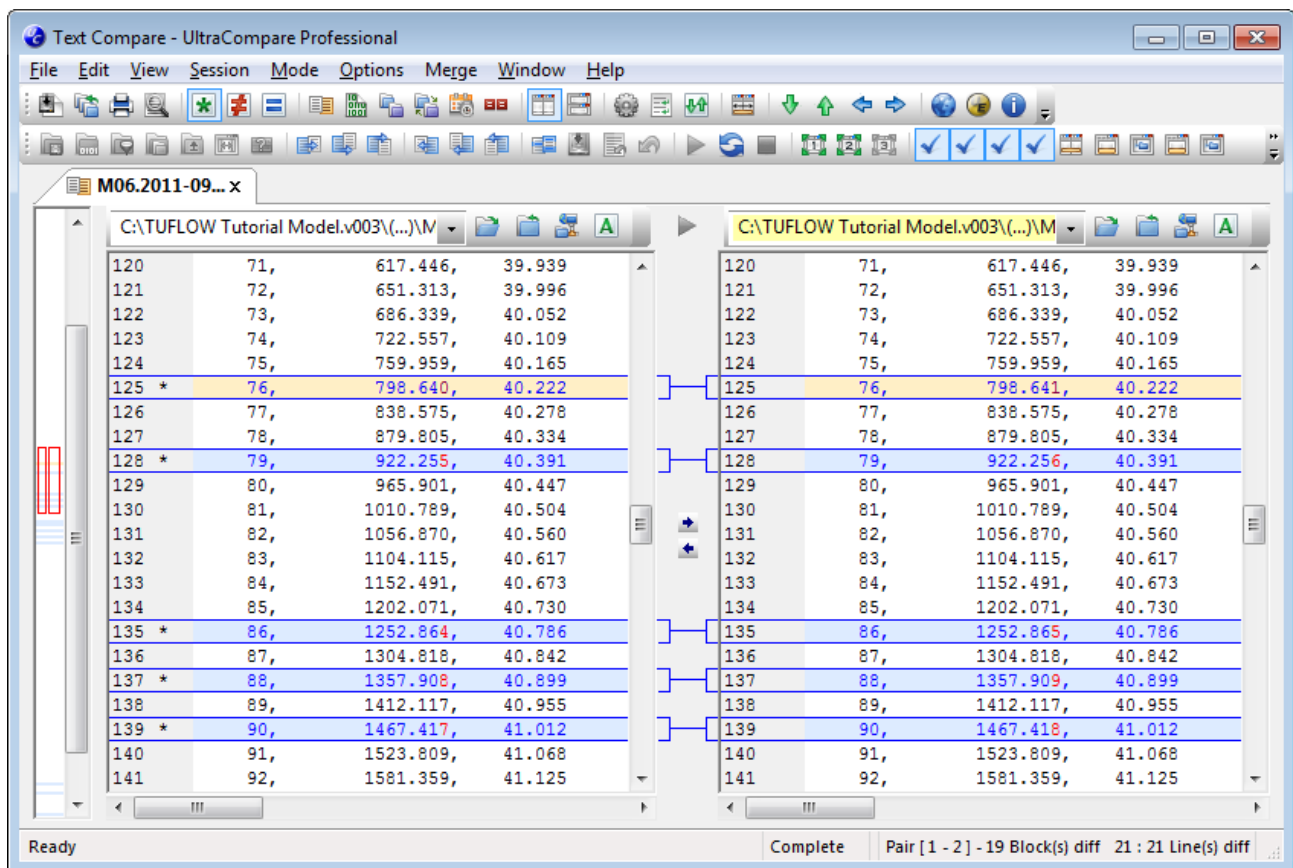
Enhancements

29 If opening or writing to an XMDF file fails, TUFLOW now repeatedly tries to open/write to the file rather than terminate. Reasons for an XMDF file not being able to be opened or written to are the usual suspects such as network connection is down, disk is full, file is protected by another application. Note, software such as SMS open XMDF files as read-only meaning that TUFLOW can continue to write to the XMDF file and that the results of the simulation can be viewed whilst the simulation is running.

30 TUFLOW AD module has been updated with the new GIS routines.

31 Increased the limit on the maximum number of vertices for a single region or aggregated region (including regions with holes) from 100,000 to 500,000.

32 Increased the limit on the maximum number of 2D cells that can be selected by a polyline from 10,000 to 50,000.

33 Increased the limit on the maximum number of XZ points in a 1D cross-section from 1,000 to 100,000. Also see http://www.tuflow.com/forum/index.php?showtopic=978. We are also planning to increase the limit in xsGenerator.exe utility accordingly.

Bug Fixes

34 Fixed a bug with restart files created by 2011-09 builds. Restart files generated by 2010-10 or earlier builds should now work with Build 2011-09-AE. Any restart files created by Builds 2011-09-AA to AD will need to be regenerated using Build 2011-09-AE or 2010-10 and earlier.

35 Fixed a bug in the 2011-09 release that fails to keep trying for a WIBU licence if all licences are taken. This applies to both Local and Network WIBU dongles. The bug does not exist in the 2010-10 or earlier releases. Also see http://www.tuflow.com/forum/index.php?showtopic=997.

36 The quotes placed around the message attribute in the _message.mif/.shp layers are now correctly applied and the attributes for these layers should now be read in correctly by GIS software.

37 Fixed a minor bug with nested If Scenarios. TUFLOW would previously stop if this bug occurred.

38 Fixed a bug where if there are more than 10,000 regions to be TINned in a Read GIS Shape command, if using XF files and/or specified the WRITE TIN option, TUFLOW would stop with an ERROR 2415. This message is now given as WARNING 2415 TUFLOW continues without writing anymore XF files or TINs for the remaining regions.

Important Note for Build 2011-09-AE

The results obtained from Build 2011-09-AE should be identical to those from earlier builds and the 2010-10 release. However, during quality control testing of the 2011-09-AE release it appears that for SP (single precision) builds the optimised compiler code is being treated slightly differently to previous builds. While most SP models tested gave identical results, a couple gave very slightly different results (averaging a fraction of mm difference). For example, for automatically generated 2D QH boundaries for SP builds, the flow values were different at the 7th significant figure (see image below comparing 2d_bc_tables_check.csv files), which causes very slightly different results. After much frustration and rebuilding of the code base we've identified that the compiler is optimising code for the 2011-09-AE single precision builds differently, thereby causing the slight inconsistencies (un-optimised code works fine, but is much slower!). Therefore, as always, it is recommended to cross-check new builds by re-running established models and comparing with the original results.
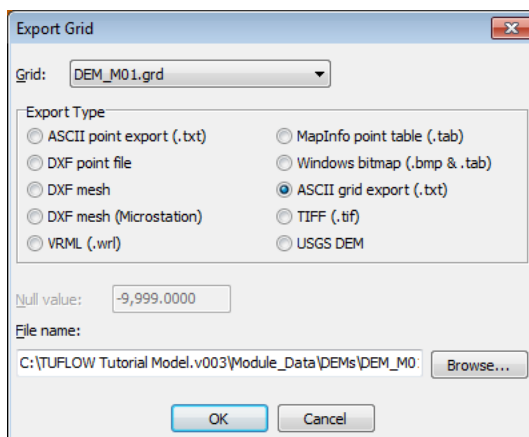
Build 2011-09-AF

Notes on new features, enhancements, and bug fixes in Build 2011-09-AF are below. Testing on a range of models has shown 2011-09-AF gives identical results to 2011-09-AE.

New Features

39 Direct reading of DEM Grids (*at last, no more creating, point inspections and export of Zpts!*).

(a) New .tgc command "Read Grid Zpts [{} | ADD | MIN | MAX]== <ESRI ASCII Grid>" that TUFLOW directly interrogates (point inspects) to set the Zpt elevations. This command is similar to "Read TIN Zpts ==" but works on a grid rather than a TIN.

(b) The only Grid format supported at present is the ESRI ASCII Grid format (same .asc format as used by TUFLOW_to_GIS to export grids).

(i) Nearly all 3D surface software (12D, ArcGIS, Discover, Vertical Mapper) offer this format for exporting grids. For example, in Vertical Mapper from Grid Manager, use Tools, Export… and chose ASCII grid export (.txt) as per below:

(ii) TUFLOW will accept any file extension at present as there doesn't seem to be a standard.

(c) By using this command, your TUFLOW model will very likely become cell size independent, so to change cell size only involves changing the .tgc "Cell Size ==" setting, and reducing your timestep. It's as simple as that.

Changing a 2D domain's orientation and dimensions will also be very very easy. If you need to change a 2D domain's orientation and/or overall dimensions, there is no need to regenerate, import into a GIS, point inspect and export your 2d_zpt layer(s) as these layers are no longer needed. All that needs to be done is change the 2d_loc layer or your "Orientation" and/or "Grid Size" commands.

(d) Testing thus far shows this new command to be very fast (much faster than doing a point inspection in Vertical Mapper, which is not particularly slow). And by default an XF file is written that carries out a binary dump of the Zpts so 2nd, 3rd, … time round it's even faster (seconds for tens of millions of points!). If you want to use or not use XF files for that layer, use the "XF ON" or "XF OFF" options (eg. to not use specify "Read Grid Zpts XF OFF ==").

(e) Like other .tgc commands, you can specify "Read Grid Zpts" as often as you like. You can also specify ADD, MIN or MAX in the same way as for other similar commands.

(f) It is planned to extend this capability to some other inputs such as direct reading of land-use grids (eg. satellite imagery) for defining materials, and grids of rainfall variation to set the 2d_rf f1 and f2 attributes.

40   Specify clip regions for "Read TIN Zpts ==" and the new "Read Grid Zpts ==" commands.

(a) These commands now accept a second argument to specify a GIS layer containing one or more polygons to clip the area of Zpts to be inspected. For example, the command below will only assign elevations to Zpts that lie inside polygons in the 2d_clip_DEM layer.

Read GRID Zpts == DEM\DEM_M01.txt | DEM\2d_clip_DEM.mif

(b) This is particularly useful for clipping out a TIN or DEM due to unwanted or irregular triangulation around the periphery, especially for secondary TINs/DEMs of proposed developments lying within the primary TIN/DEM.

Note: For your base Zpts from the primary DEM or TIN, do not clip this with your active 2d_code layer as this will cause problems with Zpts along any external 2d_bc boundaries. If no clip layer is specified, "Read GRID Zpts ==" or "Read TIN Zpts ==" assigns all Zpts falling within the TIN/Grid an elevation irrespective of whether a cell is active or inactive.

(c) The attributes of the clip layer are not used, and only polygons are processed. Polygons can have holes in them if required.

41   XF files now automatically generated for "Read TIN Zpts ==" command.

(a) By default, an XF file of the Zpts assigned an elevation from a "Read TIN Zpts" command is created so that loading up the Zpts is pretty well instantaneous for subsequent runs using that TIN. If the TIN is updated, TUFLOW will automatically resample the Zpts and create a new XF file.

(b) The XF file can be switched on or off from the global default setting using "XF ON" or "XF OFF" as mentioned above for the new Read Grid Zpts command.

Enhancements

42   ISIS-TUFLOW HX Links

(a) The HX cells in the 1d_to_2d_check layer are now colour coded to represent the ISIS unit/node that the flow in/out across the HX cells is associated with. By default HX cells are assigned to the upstream river unit as a lateral flow.

(b) The ISIS river units listed in the .csv files created if using "Write X1D Check Files == ON" were in some situations incorrect – this would be evidenced by the number of river units being different to the number of data columns. This has been corrected in this build.

(c) In collaboration with Halcrow we are looking into options to improve the ISIS-TUFLOW HX linking that in some situations can be problematic, especially where a V-shape profile forms in the ISIS longitudinal profile just upstream of junction. If you have a model that experiences this behaviour, please email support@tuflow.com or isis@halcrow.com as there is a new feature that we're testing available in this build.

(d) As a consequence of these investigations, significant improvements to the ISIS-TUFLOW linking in problematic models has resulted from:

(i) Setting the new "Mass Balance Corrector == ON" command – see description in Build 2011-09-AB below.

(ii) Assigning a FLC (typically 0.1 to 0.5 in value) to HX lines using the 2d_bc "a" attribute. For HX lines running along the river banks, especially those with high overtopping velocities, improved stability and representation of the energy lost as the water peels off from the river to floodplain or vica versa can be realised.

Please email any feedback from using the above to support@tuflow.com or isis@halcrow.com.

Bug Fixes

43 Fixed bug if Code -1 cells occurred in outer area of 2D domain –
see http://www.tuflow.com/forum//index.php?showtopic=1007

44 Fixed bug in 2011-09 release that caused TUFLOW to stop if an XY ERROR, WARNING or CHECK occurred before _messages layer is created – only known to occur for ERROR 2132.

45 Fixed bug that didn't correctly assign BC (box culvert) types in 2d_fcsh layers. The bug is evidenced by a CHECK 2097 message at U and V points indicating that the BC soffit is below the Zpt and the cell side is blocked preventing flow through the FC shape. Also the U and V Zpts in the 2d_zpt_check layer would be set very high at the instability detection level.
See http://www.tuflow.com/forum//index.php?showtopic=1008.

46 QGIS Null entries in the attribute data are now recognised and are treated as blank for character attributes and zero for real or integer attributes.

Build 2012-05-AA

Notes on new features, enhancements, and bug fixes in Build 2012-05-AA are described below.

New Features and Enhancements

47 **2D Solution Scheme Enhancements:** The 2D solution scheme has been improved as follows:

(a) Enhanced wetting algorithm that provides significant improvement to inflows on steep areas (eg. direct rainfall models), whilst maintaining low mass error, greater stability and often larger timesteps. The new approach does not require the shallow depth stability feature that was previously automatically invoked for direct rainfall models, or necessarily benefits from the mass balance corrector feature.

    (i) The new method makes an estimate of the likely velocity that will occur when a cell side first wets and feeds this information into the solution matrices. Previously, the velocity used was that from the previous timestep which was zero as the cell side was dry. The zero velocity essentially created a frictionless slope and would cause a surge of water, albeit very shallow, when the cell side first wets. This was not usually a major issue, however, with direct rainfall models all cell sides can become wet in one timestep and if the terrain is steep a significant surge and unacceptable mass errors would occur. This new feature largely overcomes this effect.

    (ii) The previous wetting and drying method can be activated by using "Wetting and Drying == ON", while the new approach, which is the default for the 2012-05 release, is set using "Wetting and Drying == ON METHOD B".

(b) In combination with the new wetting algorithm above, the 2D solution has been enhanced when transitioning between downstream and upstream controlled 2D flow regimes, resulting in improved stability and better-quality results in models where these flow regimes are prevalent. More detailed information on the water surface and energy slopes, and topography upstream and downstream, is used to assess whether flow is upstream or downstream controlled, and if upstream controlled whether it is super-critical or weir flow. Whilst requiring slightly more computational effort, there is a performance improvement, especially for highly complex flow problems, steep terrain and dambreak type analyses. For your typical river model where the majority of flow is sub-critical, there are marginal benefits and typically no noticeable effect on simulation times.

(c) Recommendations when migrating an existing model to the 2012-05 release are:

    (i) If "Mass Balance Corrector ==" is set to ON, comment out the command or set to OFF. Only in the occasional model does this feature now provide a benefit.

    (ii) For direct rainfall models if there are significant mass errors when the model first becomes wet, set "Cell Wet/Dry Depth == 0.0002". This is usually only required if the model has steep slopes. Note that setting to 0.0002 will cause longer simulation times as there will be more wet cells during the simulation.

    (iii) Most direct rainfall models will still require using double precision (DP) builds of TUFLOW.

    (iv) If you're 2D timestep was previously small, it's very likely that a larger timestep can be used.

(d) The above enhancements require that commands such as "Supercritical == " are set to ON. Therefore a number of these commands are hard-wired to ON for the 2012-05 release provided "Defaults == Pre 2012" is not set.

TUFLOW

48 **Automatic Manhole Enhancements and Bug Fixes:** The sizing of automatic manholes and application of losses has been enhanced. The new .ecf command "Manhole Approach == [Method A | {Method B}]" sets which approach is being used, with Method B being the default for the 2012-05 release. Method B includes several enhancements and bug fixes as documented below. Method A (the previous approach) tended to be too conservative (ie. higher losses and therefore higher flood levels) as indicated by some forum posts (for example, http://www.tuflow.com/forum//index.php?showtopic=941).

(a) Previously, a junction (ie. there is no manhole chamber) was assigned to a node where there was only one inlet and one outlet culvert and they were large in size (ie. negating the need for an access manhole). This caused a problem if a third culvert, such as a small side pipe draining a pit, entered the node. The side pipe caused a manhole to be created, thereby causing significant losses due to the expansion and contraction of flow in the manhole chamber. The new Method B approach is to make any node that has at least one large culvert (>1.801m dia for pipes and >1.201w by 1.801h for boxes) a junction. In this case only change in the velocity magnitude and direction, and invert level drop losses, are modelled. As there is no manhole chamber, no expansion/contraction losses are simulated. At a junction the change in velocity magnitude is based on the primary downstream/upstream culvert (these are listed in the .eof file and are based on the connected culvert of the most similar size and invert).

(b) If a pipe enters a manhole elevated above the main trunk culverts (eg. a small pipe draining a pit or gully trap into the manhole), this would previously cause higher losses than should be applied as the elevated pipe would enlarge the manhole and increase the losses. Method B tests for any inlet culverts entering the manhole that are elevated above the highest outlet culvert (note, whether a culvert is an inlet or outlet culvert is based on its digitised direction). If an inlet culvert's invert is above the highest outlet culvert soffit, the elevated culvert is not included in the determination of the manhole size or manhole loss calculations. The exit losses from the elevated culvert are still modelled but separately to the manhole losses and provided the culvert's flow is sub-critical, the full exit losses are applied irrespective of whether "Structure Losses ==" is set to ADJUST or FIX.

(c) If using the Engelhund approach (the default), the expansion/contraction losses of flow into and out of a manhole (ie. inlet culvert exit loss and outlet culvert entrance loss) was based on the change in velocities from the inlet culvert to the manhole, and from the manhole to the outlet culvert (as per the standard adjust structure losses equations in the TUFLOW manual). The velocity through the manhole is calculated using the manhole's vertical flow area. If the depth in the manhole was large (ie. well above the culvert soffits), excessive losses could result because of the small manhole velocity resulting from the large manhole depth. In reality, there would an inactive vertical flow area in the manhole in the zone above the culverts. This is now taken into account by limiting the vertical flow area in the manhole to a height of half the manhole flow length above the highest culvert soffit (excluding any elevated inlet culverts as discussed above).

(d) The 1d_mhc layer now shows the soffit of the highest contributing culvert at a manhole for cross-checking.

(e) Method B incorporates the following bug fixes. They have not been fixed for Method A to allow for backward compatibility if required for existing models:

(i) For the FIXED manhole loss approach, the losses were incorrectly applied if the fixed loss coefficient exceeded 0.5. The bug generated greater losses and typically slightly higher water levels in the manhole.

(ii) A bug that incorrectly reported the form (ie. bend) loss value for culverts in the _TSL layer has been fixed.

(f) Testing on a range of models with pipe networks has generally shown negligible to minor (5 to 20cm) changes in manhole and overland flood levels, with Method A being more conservative (producing higher levels) than Method B. Where the bulk of the flow is within the pipe network, greater changes may occur, especially for large culverts as discussed above.

49 **US Customary Units:** US Customary (or English or Imperial) Units are now fully supported.

(a) To activate US Customary Units use the command below anywhere in the .tcf file.
Units == US Customary
(alternatively, "Units == English" or "Units == Imperial" can be used).

(b) Input and output units are as follows:

(i) Length: feet

(ii) Rainfall, Initial Loss and Continuing Loss(/h): inches

(iii) Catchment Area: square miles

(iv) Constant eddy viscosity value: ft^2/s

(c) The default settings for all inputs are the same as for metric, but converted to their US Customary units' equivalent. Where the 2010-10 manual or these release notes refer to a default value in metric, TUFLOW will use the equivalent value in US Customary Units. To cross-check, search for the relevant parameter in the .tlf file.

50 **New Infiltration Feature:** TUFLOW now can infiltrate ponded water into the ground. Two methods are provided: Green-Ampt and Initial Loss/Continuing Loss. One or more soils need to be specified globally and/or via GIS layers/ASCII grids to activate the infiltration feature. Each soil type can have a different infiltration method including a no infiltration option, and have different infiltration parameters. Other parameters that can optionally be set are: the imperviousness of the surface (this is via the materials definitions); and groundwater or impervious level beneath the ground surface.

The **Green-Ampt** approach varies the rate of infiltration over time based on the soil's hydraulic conductivity, suction, porosity and initial moisture content. The **Initial Loss/Continuing Loss** method is a more simplistic approach and infiltrates water based on an initial amount then at a constant rate. Both methods monitor the amount of water infiltrated and if the ground becomes saturated, no more infiltration occurs. The amount of water that can be infiltrated depends: on the depth to groundwater or an impervious layer; and the soil's porosity and initial moisture. Only wet 2D cells can infiltrate water into the ground.

The amount of water that enters the soil is also dependent on the fraction impervious value of the overlying material layer. The default is that the overlying material is 100% pervious (ie. 0% impervious). However, if, for example, a concrete parking lot overlies a sandy soil, the imperviousness of the parking lot can be specified as 100% to totally, or, say 90% to partially restrict the rate of infiltration.

Note that the IL/CL infiltration is totally separate to the IL/CL materials values used to generate excess rainfall for direct rainfall simulations. It is possible to use both methods in the same simulation – for example, rainfall that doesn't reach the ground such as interception by trees would be modelled as a material IL and infiltration into the ground as IL/CL via soil types.

The various inputs available to setup infiltration are:

(a) **Fraction impervious** of the overlying material is specified via the materials definitions and should be a number from 0.0 to 1.0 where 0.0 is fully pervious and 1.0 is fully impervious. To specify:

   (i) Materials .tmf file: The 11th number is used for setting the Fraction Impervious. Numbers 1 to 8 are as documented in the 2010-10 manual. For the 9th number specify -1 as this field is reserved, and the 10th number is for the new Storage Reduction Feature (SRF) feature. For the SRF value use 99999. if not using the SRF feature, or specify the SRF value (0.0 is the default).

   (ii) Materials .csv file: Use the 6th column. Columns 1 to 3 are as documented in the manual. The 4th column is reserved and should be left blank. The 5th column is for the SRF number as discussed above except leave blank if no SRF number (ie. do not specify 99999.).

(b) .tcf commands:

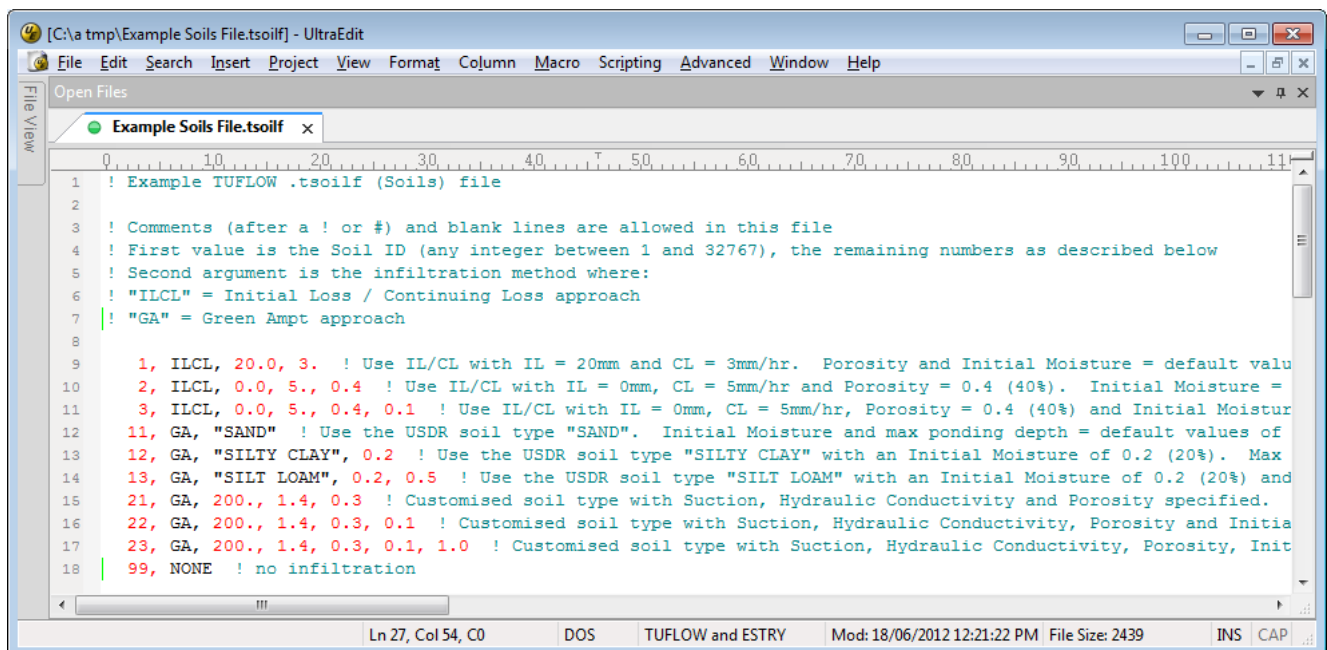   (i) "**Read Soils File == file.tsoilf**"

   An example soils (.tsoilf) file is shown further below and the parameters in the table below. It is similar to the materials file where you assign an integer ID to each soil, define the infiltration method (Options are "NONE", "ILCL" and "GA") followed by the parameters as the 3rd, 4th, 5th, … values.

   The Porosity, Initial Moisture and Max Ponding Depth are all optional with default values of 1.0, 0.0 and 0.0 respectively.

Method	.tsoilf Parameters						
	1st	2nd	3rd	4th	5th	6th	7th
No Infiltration	Soil ID						
Initial Loss / Continuing Loss	Soil ID	ILCL	Initial Loss (mm or inches)	Continuing Loss (mm/h or inches/h)	Porosity (Fraction)	Initial Moisture (Fraction)	
Green-Ampt	Soil ID	GA	USDR Soil Type (see table below)	Initial Moisture (Fraction)	Max Ponding Depth (m)		
			Suction (mm or inches)	Hydraulic Conductivity (mm/h or inches/h)	Porosity (Fraction)	Initial Moisture (Fraction)	Max Ponding Depth (m)

The hard-wired USDR values built into TUFLOW are:

USDR Soil Type	Suction (mm)	Hydraulic Conductivity (mm/h)	Porosity (fraction)
CLAY	316.3	0.3	0.385
SILTY CLAY	292.2	0.5	0.423
SANDY CLAY	239.0	0.6	0.321
CLAY LOAM	208.8	1.0	0.309
SILTY CLAY LOAM	273.0	1.0	0.432
SANDY CLAY LOAM	218.5	1.5	0.330
SILT LOAM	166.8	3.4	0.486
LOAM	88.9	7.6	0.434
SANDY LOAM	110.1	10.9	0.412
LOAMY SAND	61.3	29.9	0.401
SAND	49.5	117.8	0.417

(c) The .tgc commands available are:

(i) Set Soil == <iSoil>

where <iSoil> is the integer Soil ID in the .tsoilf file

(ii) Read GIS Soil == gis_soil_layer.mif/.shp

Only one attribute which is the iSoil integer value (same concept as the materials layer).

      (iii)   Read GRID Soil == soils_grid.asc

      (iv)   "Set GWL ==", "Read GIS GWL ==", "Read GRID GWL =="
            for setting the groundwater or impervious level. Default is for the GWL to be infinitely deep.

      (v)   "Set GWD ==", "Read GIS GWD ==", "Read GRID GWD =="
            for setting the groundwater or impervious depth. Default is for the GWD to be infinitely deep.

      (vi)   If a cell has a GWL and a GWD applied to it the higher one (elevation wise) prevails.

      (vii)   Note: 2D infiltration is activated by the occurrence of one of the above .tgc commands. If none of the above commands exists for a 2D domain, infiltration does not occur for that domain.

   (d)   If soils are specified, the parameters for each cell are written to the 2d_grd_check layer.

   (e)   Three new "Map Output Data Types ==" are provided:

      (i)   "CI" for Cumulative Infiltration over time in mm or inches.

      (ii)   "IR" for Infiltration Rate in mm/h or inches/h.

      (iii)   "dGW" for depth to groundwater (from the ground surface) in metres or feet.

51   New .tgc command "**Read GRID Location == <dem.asc>**" to set the size and location of a 2D domain based on a DEM. The dimensions of the ASCII grid is used to set the 2D domain's origin and X,Y dimensions (ie. replaces "Origin ==" and "Grid Size (X,Y) ==" or "Read GIS Location ==""). The orientation angle is set to zero (ie. the 2D domain will be orientated north-south). Useful where the model extent is the same as the DEM. "Cell Size ==" still needs to be specified and can be different to the DEM's cell size.

52   "**Read GRID <Type> ==**" has been extended from just Zpts to also accept grids of Code, Mat, Soil, GWL, GWD, IWL, CnM, WrF, FLC and SRF values. For real (float) inputs the value is interpolated from the grid. For integer inputs (eg. Material or Soil IDs), the value applied is the integer value of the ASCII grid cell that the TUFLOW point falls within. Examples of use are:

   (a)   Satellite imagery grids to easily assign material values.

   (b)   The quick conversion of 2D models in other software, by setting of Zpts and Manning's values using "Read GRID Zpt" and "Read GRID CnM" with exported ASCII grids of the bathymetry and the Manning's values (if the Manning's values are values of M, not n, then "Bed Resistance Values == MANNING M" needs to be specified).

XF files are by default automatically generated for all "Read GRID" datasets, making the startup time very fast for subsequent simulations.

A "Read GRID RF" feature is under development for the quick and easy input of rainfall distributions via radar imagery.

53   The .tgc "**Read TIN Zpt == <file.2dm>**" command now recognises .2dm files. The .2dm mesh file is used to generate a TIN to set elevations in the .tgc file. This option is useful for quickly setting up a TUFLOW model based on a flexible mesh model that uses a .2dm file.

54   New .tgc command "**Set Code and Clean Zpt == <Z_inactive>**". Assigns cells as active (Code 1) or inactive (Code 0) based on whether the cell has been assigned an elevation or not. Also extrapolates Z values to any unassigned Zpt values in cells assigned as active. The value of <Z_inactive> is used to assign an elevation to unassigned Zpts. This command negates the need to digitise active/inactive code polygons where:

(a) the DEM used to assign elevations has been trimmed to the catchment boundary or model extent (ie. all null areas of the DEM will be assigned an inactive code as these Zpts have not been assigned an elevation); or

(b) reading a .2dm file from another model to assign elevations – see point above.

This command must occur after the Zpts have been assigned their elevations from the DEM or 2dm file. Do not use the "Set Zpt ==" command as this assigns every Zpt a value, and therefore all Zpts have been assigned a value and "Set Code and Clean Zpt ==" will not work.

55 **Adaptive Time-stepping:** New .tcf command "**Maximum Courant Number == <Cr_max>**" that switches on adaptive time-stepping. This is a new **under-development** feature that at present is only available for 2D only models. Based on further successful testing, it may be extended to 1D/2D models. Cr_max would typically be in the range from 2 to 10, but higher values up to 60 or more have been successfully applied. Higher Cr-max values can be justified if consistent results are achieved using a smaller Cr_max or a fixed timestep. The "Timestep ==" command is only used to set the initial timestep if "Maximum Courant Number ==" is specified (Note: to avoid the simulation terminating early, specify an initial timestep that is less than the average adaptive timestep – this issue will be fixed for a future update). A Cr_max value of 0 or less switches adaptive timestepping off (which is the default). The maximum rate at which a timestep can increase is controlled by "Timestep Maximum Increase (%) == [{0.5} | <value_as_a_%>]". There is no limit to how quickly the timestep can decrease.

Note that adaptive time-stepping may slow the simulation down if there are a few locations in the model that significantly reduce the timestep to values below that normally used. Adaptive time-stepping can also be problematic for impact assessments if the timesteps in the before and after scenarios are sufficiently different to cause some unusual and unexpected impacts away from the area of interest!

56 New **"Map Output Format =="** GRID option to directly write ASCII grids of TUFLOW output. At present, the water levels, depths and/or velocities of the 2D domain are supported with other "Map Output Data Types ==" to be added in the future. There is no rotation of the results if the 2D domain is not north-south orientated. To activate, specify "GRID" as one of the outputs for "Map Output Format ==", and grids are written every map output interval and for the maximums/minimums at the end if maximums and/or minimums are being tracked. To only output the maximum/minimum grids at the end of the simulation use "Maximums and Minimums Only For Grids == ON". The output grids are placed in a sub-folder called "grids". Grid output can be specified with or without other output formats such as DAT and XMDF.

This feature is useful for large 2D only models (especially for the new GPU solver), that are orientated north-south. The TUFLOW_to_GIS –grid functionality is planned to be incorporated as an enhancement to this feature in the future, so that grid orientation and 1D WLL output are supported.

57 The default viscosity coefficients have been changed to 0.5 Smagorinsky coefficient and $0.05m^2/s$ constant. Note the 0.5 Smagorinsky coefficient is dimensionless (the viscosity coefficient in m^2/s using the Smagorinsky approach is recalculated every timestep using the Smagorinsky formula and varies spatially and temporally). The defaults were previously 0.2 and $0.1m^2/s$. Testing on the UK EA 2D benchmark Test 6 models showed that the lower constant component produces improved results. For nearly all models the influence of this change is negligible or minor, as the lower constant component and higher Smagorinsky coefficient tend to cancel each other out.

58 The .tcf "**SX ZC Check == <dZ_limit>**" command has been extended so the user can set a maximum change in ZC elevation caused by the use of any "Z" flags for 2D SX links. For example, if "SX ZC Check == 0.5" is specified, then if any 2D cell is lowered by more than 0.5m (or ft) an ERROR 2050 occurs. This

allows the modeller to automatically lower 2D cells within the specified limit, but flag an ERROR 2050 if there is a substantial change, which usually indicates there is something inconsistent between the 1D channel bed and the 2D topography. It is strongly recommended that if using the SX Z flag, that this new option is specified.

59 **New .tsf file (TUFLOW Summary File):** The new .tsf file (TUFLOW Summary File) that is output to the same location as the .tlf file provides a more concise summary of the simulation. This file can also be regularly updated during a simulation. Features are:

(a) Use "TSF Update Interval == [{0} | <interval_in_sec>]" to set the interval in seconds to update the .tsf file while a simulation is running. If set to 0 or less, the default, the .tsf file is only updated at the start and the end of the hydrodynamic calculations.

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

60 The .tbc command "**Global Rainfall BC == <BC_name>**" command has been changed so that it is applied to all active cells. Therefore if the rainfall being applied is the same for all cells, this command negates the need to use a 2d_rf layer.

This command predates direct rainfall modelling and was previously used to apply local rainfall inflows to only wet cells with automatic factoring up to include the contribution of dry cells. However, the command had a bug where no rainfall was applied if the model started completely dry, and was in conflict with the current practice of applying rainfall boundaries to all cells. At present there is no backward capability provided (please contact support@tuflow.com if you would like this).

61 The .tcf command "**SA Proportion to Depth == ON**" is now set as the default approach for SA inflows. Previously the approach was to proportion the SA flow equally over all wet cells. Proportioning to depth provides improved performance if using as a side inflow, especially for dambreak type analyses, and reduces the effect of any duplicated routing when applying local hydrographs from a hydrology model. If "Defaults == Pre 2012" is set, "SA Proportion to Depth == OFF" is applied as the default to maintain backward compatibility.

62 New .tcf command "**Force File IO Display == [ON | {OFF}]**" that, if set to ON, forces all file opening and closing information to be displayed to the screen and .tlf file. At present, nearly all file opening and closing is displayed, but some isn't and more maybe removed in future releases (eg. "Write PO Online == ON" displays a lot of file opening and closing information as the simulation proceeds, so this may be removed in the future). This command would mainly be used for debugging a file opening and closing issue.

63 If "GIS Format == [{MIF} | SHP]" command is not specified, the GIS format used for check layers and other GIS outputs is based on whether "MI Projection ==" or "SHP Projection ==" has been specified. If neither or both of these commands have been specified, and "GIS Format ==" has not been specified, the default of using .mif files is adopted. Previously, "GIS Format == SHP" had to be specified to output in .shp format.

64 The "CSV Time ==" command now recognises seconds and minutes (previously only hours and days). To output the time column in seconds or minutes, specify "**CSV Time == SECONDS**" or "**CSV Time == MINUTES**".

65 New .tcf command "**CSV Header Line == SINGLE**". Specifying SINGLE will only output a single header line in _PO.csv files, which makes it much easier to use the file for charting in Excel. The simulation name

is also included in the label so that it's easy to distinguish between simulations when graphing comparisons. This is not the default setting, so this command needs to be specified to activate the feature. Also note that the charting macros provided in TUFLOW Tools.xls have not yet been updated to support this format.

66 The maximum number of columns for 1D .csv output files was previously set to 250 to comply with the limitations of earlier versions of Excel. For the 2012-05 release, this limit has been removed; however, the number can be manually set using the new .tcf or .ecf command "**CSV Maximum Number Columns == <max_col>**".

67 Bed Shear Stress (BSS) and Stream Power (SP) map output can be misleading at very shallow depths as the BSS formula divides by the depth. The BSS and SP are for the 2012-05 release linearly reduced to zero once the depth is below a threshold (by default, 0.1m). To change this threshold, use "**BSS Cutoff Depth == <BSS_cutoff_depth>**" in the .tcf file.

68 The new command in Build 2011-09-AE, "**Maximums and Minimums Time Series == ON**", has been set as the default. However, for _PO.csv files the maximums and minimums are written not beneath the header lines, but to a separate _POMM.csv file.

69 The 1d_to_2d_check layer attribute "Z_Bed_2D" has been renamed "Lowest_ZC_2D" as the name was somewhat misleading for SX connections. In the .mif layer, all SX cells connected to the same 1D node are grouped together as one object, and the Lowest_ZC_2D (previously Z_Bed_2D) value is the lowest 2D cell ZC value of all the cells connected to the 1D node. For the .shp layer, it's not possible to group the cells as one object, so each cell is separate, and therefore, when clicking on the SX cells, note that the value is still the lowest ZC of all 2D cells, not of the individual cell. For 2D HX links, the value is the ZC value of the individual cell.

Bug Fixes

70 The flow exchange volumes across ISIS-ESTRY links were not included in the overall mass balance .csv files, and in the CE% displayed to the console and written to the .tlf file. This has been corrected for 2012-05, so hopefully those mass error percentages are back down to zero! This fix does not affect the hydrodynamic calculations, only the overall mass balance reporting.

71 Zero width and/or zero flow area for Q pits could cause TUFLOW to crash or NaNs to appear in the output. An ERROR 1092 or 1093 is now reported if these occur. If "Defaults == Pre 2012" is set, a WARNING 1092 or 1093 is issued and the model simulation will not stop.

72 If a zero width was specified in a HW (or CS) table for a channel section, this would potentially cause TUFLOW to crash or NaNs to appear in the output. The most common cause for this to occur was when specifying a HW table for an arch bridge with the top width set to zero. For the 2012-05 release, if a zero width is encountered it is set to 0.001 to avoid a divide by zero. If "Defaults == Pre 2012" is specified, the width is left at zero to maintain backward compatibility noting that there is a chance of a crash or NaNs occurring.

73 For bug fixes relating to automatic manholes, see point 48.

74 The node surface areas (NA tables) automatically calculated for some nodes with no manhole were incorrect when using the automatic manhole generation, with the nodal areas being based on an automatically generated manhole when no manhole was actually applied to that node. This occurred at nodes at the start or end of a culvert that was not connected to another 1D channel (ie. the node only had the culvert connected to it). In the majority of instances, this occurred at 2D SX connections and has no

bearing on the calculations, as the 1D/2D linking automatically applies a 1D HX link to the node, and being a water level (H) boundary link the nodal area is not used in the calculations. In other instances, such as the start of a pipe network with a 1D inflow boundary and no 2D link, the nodal area calculated would be slightly larger and would most likely have negligible influence on the results.

75 If a polyline for a wide Z line in a 2d_zsh layer contained coincident vertices, TUFLOW would generate NaNs and/or crash. All coincident vertices are now removed for Z lines prior to processing.

76 When using "Read GIS Z HX Line ==" in the .tgc file to set 2D HX cell elevations using HX lines, if any part of a HX line, except the last one, of a multi-part HX polyline has no points snapped to it, this causes TUFLOW to crash. This bug has been fixed and the crash only occurred for .shp layers. For .mif layers the simulation won't crash but there is a bug in the sense that the unprocessed polyline parts are not processed, for which no backward compatibility is provided.

77 If TUFLOW can't find or open a file, it displays a dialog box with options to Retry or Cancel. Previously if Cancel was selected the simulation was closed immediately without closing any unopened files (eg. check layers). For the 2012-05 release, all files are now closed off and the Premature Exit dialog is displayed as occurs if an ERROR message results. This fix is particularly important if using .shp layers as .shp layers, unlike .mif layers, need to have information in their header finalised before closing off, otherwise the layer cannot be opened.

78 Fixed a bug that failed to close all of the files when closing a .shp layer. If using "Write PO Online == ON" the problem was exacerbated each time the .shp layers were written and eventually a crash could occur. The workaround for 2011-09-AF and prior builds is to avoid using "Write PO Online", especially if there are frequent map outputs. The problem does not occur if using or outputting in .mif format.

79 Fixed incorrect paths in .wor and other files created by TUFLOW if TUFLOW was initiated using a relative path to the .tcf file.

80 Fixed bugs that incorrectly output the eddy viscosity (t) and flow regime (R) values for SMS High Resolution map output.

81 Fixed bug that did not display the correct eddy viscosity coefficient values for the first map output time.

82 Fixed bug that was not closing a clip layer used for "Read TIN" or "Read GRID" .tgc commands.

83 Fixed bug that incorrectly read SRF values as an integer for materials in the material .csv format.

84 .2dm files created by TUFLOW now handle models with 100,000,000 or more cells (previously, the .2dm file would be corrupted for models of this size). And yes, someone has created a model in excess of 100 million cells (120 million in fact!).

Backward Compatibility

85 For backward compatibility to the 2011-09 release use the following .tcf command:

Defaults == Pre 2012

Note that very similar results should occur, but identical results are unlikely. Even simply changing gravity from a hard-wired constant of 9.81 to a variable for supporting US Customary Units caused the results to change by fractions of a millimetre – the joys of software development!

Build 2012-05-AB

Notes on new features, enhancements and bug fixes in Build 2012-05-AB are provided below.

86 New 2D/2D linking method "**Link 2D2D Approach == Method C**":

   (a) "Link 2D2D Approach == Method B" introduced testing for dry 2D cells between the hidden 1D nodes. If dry cells were present, the water levels along the 2D link line were not interpolated between the two hidden 1D nodes (as Method A would do), but projected the water levels horizontally to the dry cells from the two hidden nodes. This solves a problem that can occur using Method A that creates spurious inflows along dry sections of the 2D link line.

   (b) However, Method B would allocate the storage to the hidden 1D nodes based on where the high point was located. Consequently, in some situations, especially if the 2D link line crosses a river or major flowpath, some hidden nodes are allocated a very small storage and/or a high bed elevation because it was only allocated the storage from one or two 2D link cells on the top of the river bank. The small storage and/or elevated bed would cause the 1D mass equation solution at the node to not converge, especially if transferring large volumes of water. This could result in a sudden, unacceptable, drop in water level across the 2D link line.

   (c) Method C is the same as Method B, but allocates the storage of the 2D link cells (these cells do not contribute to the model's storage so the storage is assigned to the hidden nodes) more evenly between the hidden nodes and ensures the nodes' bed elevation is at or below the lowest connected 2D link cell.

   (d) For models using 2D link lines that do **not** experience a sudden drop across some/all of the links, Method C should produce similar results. For problematic models, or if you've been frustrated with the 2D linking, please try Method C and if the model's performance does not improve, please contact support@tuflow.com.

   (e) For Build 2012-05-AB, the default setting is to use Method B, so "Link 2D2D Approach == Method C" will need to be specified.

   (f) Any feedback, good or bad, on Method C to support@tuflow.com would be much appreciated.

87 New .tcf file command "**Zpt Range Check == <zmin>, <zmax>**" that will stop with an ERROR 2444 (WARNING 2444 if Defaults == Pre 2012) if any final Zpt is less than zmin or greater than zmax. The defaults for zmin and zmax are -9998 and 99998 (-9998 is used for the minimum value as some 3D surface software use -9999 as the null value). Useful for checking there are no Zpts with inappropriate values. This check is only carried out once on the final Zpt values just before the simulation starts, and after writing the 2d_zpt_check layer. In the unlikely event of an invalid or NaN Zpt occurring an ERROR 2445 (or WARNING 2445) is issued.

88 **Enhancements and bug fixes for the new infiltration feature** in 2012-05-AA:

   (a) If the infiltration feature was switched on (eg. by specifying a soils layer), and there were no source in/outflows (eg. RF and SA boundaries), the model would rapidly dry and produce very large negative depths! This has been fixed.

   (b) Improved handling of 2D cells that are dry, but are receiving a source (eg. RF) inflow and subsequently wanting to infiltrate at the same time.

(c) New .tcf "GA Convergence Value ==" to set the Green Ampt iteration infiltration convergence test in m or ft. Also, the default value has been changed from 0.0001 to 0.001 metres (convert to feet for English Units option).

(d) New .tcf "GA Maximum Iterations ==" to set the limit on the number of iterations for the Green Ampt solution. The default value is 10. If the number of iterations exceeds this value a WARNING 2302 is issued.

(e) If specifying a fraction imperviousness for an overlying material, this is now applied correctly.

(f) The infinite depth option by specifying -99,999 is now correctly applied.

(g) Fixed a bug that misinterpreted "Map Output Data Types" "IR CI" (infiltration rate followed by cumulative infiltration) as "RC" (route categories). As a general note, when specifying output data types, it is now strongly recommended to separate each one with a space. For example, instead of "hVdIRCI", use "h V d IR CI" to ensure there is no misunderstanding.

89   .tgc "**Read Grid**…" commands:

(a) Grids that have origins with X,Y values less than -99,999 now work.

(b) ASCII grids that are located using "xllcenter" and "yllcenter" to set the origin in the ASCII file header are now recognised. Most ASCII grids use "xllcorner" and "yllcorner".

90   **2d_po bug fixes**:

(a) A bug in 2012-05-AA that incorrectly output a different number of header labels and data columns for multiple 2D domain models has been fixed.

(b) A bug in 2012-05-AA that incorrectly output water levels for polyline 2d_po objects has been fixed.

91   1d_nwk **M channels**:

(a) If two or more 1d_nwk M channels exist, there was a bug that could cause strange results and output NaNs if the size of the hQh matrices were different. This is now fixed.

(b) By default, TUFOW now uses an improved method for interpolating into the matrix. The previous method (which can be enabled using the new .ecf command "M Channel Approach == Method A") interpolates using a 4 value interpolation routine that, especially along the 0 flow diagonal, might not interpolate as accurately as using a triangular (3 point) technique. The main issue with the 4 point routine is that a slightly +ve or –ve flow of the wrong sign may result when interpolating close to the zero flow diagonal. The new "Method B" routine uses a triangular interpolation with the long side of the triangle parallel to the zero flow diagonal, and does not experience this problem. For backward compatibility to 2012-05-AA and prior releases, use "M Channel Approach == Method A" or "Defaults == Pre 2012".

92   Fixes a bug that would not recognise the "1D" if used in front of "Write Check Files" command, or if this command fell within a "Start 1D Domain" block in the .tcf file. For example, "1D Write Check Files == …" in the .tcf file was setting the location of the 2D check files when it should only set the location of the 1D check files.

93   Fixes a bug that incorrectly issued an "ERROR 0309 - Unexpected end of .mif file encountered." if a region object in a 2d_zsh.mif layer was the aggregation of two or more polygons (eg. two or more polygons had been aggregated or a region has a hole). TUFLOW now correctly reports "ERROR 2226 - Multiple region objects not supported for Read GIS Shape commands".

94 File naming of the new grid output now uses preceding zeros ("0") rather than underscores ("_") for the output time to make the file names more conducive to listing in order and for inclusion in GIS post-processing scripts.

95 The maximum number of materials allowed has been increased from 100 to 1,000 (yes, there is a model with over 100 different materials!).

96 If reading an .xf file directly (ie. the filename in the .tgc file is the .xf file, not the .mif or .shp file), the error reporting now makes sense if there is a problem accessing or reading the file.

97 Fixes a bug that sometimes shows a node as a manhole in the .eof file NA tables if the manhole feature is not active – this bug has no influence on the hydraulic calculations.

98 The "Read GIS Z Shape Route ==" command now works at 2D water level boundary and HX cells. Previously these cells were not considered in the route analysis.

Build 2012-05-AC

Notes on enhancements and bug fixes in Build 2012-05-AC are provided below.

Backward Compatibility

There are three backward compatibility issues to be aware of with Build 2012-05-AC:

99   Two of the bug fixes listed further below prevent the occurrence of "Not a Number" or NaNs (see description below). These changes may cause very slightly different results to occur in some models. Testing on a range of models has shown that some have zero change, while some have experienced changes of up to a few cm in peak levels at isolated locations. Unfortunately, it is not possible to provide backward compatibility for these changes as they are essential to ensure that NaNs do not occur. The changes apply to enhancements in the wetting and drying algorithm introduced to the 2012-05 release and does not affect TUFLOW releases prior to 2012-05.
What is a NaN?: In computing NaN stands for Not a Number and can be caused by, for example, a divide by zero that leads to a crash or nonsense results displayed or output by TUFLOW as "NaN" or a series of asterisks "********". This can occur, for example, when the implicit matrix solution overshoots and produces a zero or negative depth.

100 A slightly different approach is now used to track the maximum water level so that the same process is used for both cell cornered output (eg. .dat and .xmdf formats) and cell centred output (eg. the new ASCII grid option). As a consequence, models may show in isolated locations a very slight (usually less than a mm) difference in the maximum water level. A backward compatibility switch "Maximums Approach == Method A" is provided as discussed further below.

101 Consecutive flow values in automatic 2D HQ boundary tables are now automatically removed (TUFLOW would previously stop with ERROR 2194). For existing models that run with an automatic 2D HQ boundary, if the change in consecutive flow values is less than 0.00001 the results may be very slightly affected by this change. "HQ Boundary Approach == METHOD B" provides backward compatibility if this is an issue.

As always, to cross-check if your model is affected by the above points re-run a 2012-05-AA or AB simulation with 2012-05-AC and compare results by viewing the change in the maximum water level surface using the SMS data calculator or a GIS ASCII grid calculator.

Enhancements

102 New run options:

   (a)   **–acf** to automatically create folders. If –acf is specified, the dialog prompts that appear to create a non-existent folder (eg. results folder) do not appear and the folder is created automatically. Note that if the folder can't be created a dialog will appear.

   (b)   **–nq** to not display queries. At this stage the only query not displayed if this option is used is if you press Ctrl-C to terminate a simulation cleanly. If –nq is specified and Ctrl-C is pressed, the simulation terminates cleanly without a query dialog.

103 The "**If Scenario ==**" syntax commands could not previously be used around the "**Read GIS Cyclone/Hurricane ==**" and "**Read GIS ISIS/XP Node/Network/WLL/WLL Point ==**" commands. These commands can now be used within "If Scenario ==" blocks. All commands should now be covered by the "If Scenario" constructs.

TUFLOW ≈≈

104 The .tbc "**Read GIS BC ==**" command now accepts up to 10 GIS layers in the one line. This is needed for **.shp layers** where points and lines are both used (for example, if using HX lines and CN points), so multiple layers need to be read in for the one command. The format is similar to the "Read GIS Z Shape ==" command with the different GIS layers separated by the "|" character. For example:

```
Read GIS BC == gis\2d_bc_hx_L.shp | gis\2d_bc_hx_P.shp.
```

105 **M Channels**:

    (a) If the number of characters in a single line of an M channel stage-discharge or stage-area .csv file exceeded 1,000, TUFLOW would ignore the remaining values and issue an ERROR 1133. The limit has been increased to 10,000 characters for this build.

    (b) The 1d_nwk "Inlet_Type" (formerly "Branch") attribute is set to 12 characters in the empty layer and had a limit of a maximum of 50 characters if lengthened. This attribute is used for specifying M channel .csv filename(s), and has been increased to 256 characters (the limit of most GIS software string attributes) to allow for longer filenames to be specified. The empty layer length remains at 12 characters, so will have to be manually increased to make use of this feature.

106 Enhancement to automatic **2D HQ boundaries** that if the consecutive flow values are the same, the second entry is removed, so as to not cause an "**ERROR 2194** - Column 1 values not ascending" message. This problem would occur where the depth range for the HQ curve was very shallow. For backward compatibility use "**HQ Boundary Approach == METHOD B**" (the new approach is "**HQ Boundary Approach == METHOD C**").

107 If "**Mass Balance Output Interval ==**" is set to 0 (zero), the output interval used is the largest 2D computational timestep set by "Timestep ==". Previously if set to zero, the smallest of the "Time Series Output Interval ==" and "Map Output Interval ==" was used (which remains the default setting, or if "Mass Balance Output Interval ==" is set to -1).

108 For manually specified manholes, "**Manhole Default Type ==**" must be set to "C", "J" or "R"; the "CR" and "CJR" options cannot be used noting that "CJR" is the default. A new message "**ERROR 1207** - Default manhole type for digitised manholes with blank Type must be C, J or R" is now issued if "CR" or "CJR" is the setting and a blank type occurs for a manually specified manhole.

109 **Empty 1d_tab .shp** layers now are exported for point layers (previously only line layers).

110 "**ERROR 2009** - Grid cell is a boundary or inactive (null or land) cell, but has an internal boundary condition applied to it" was previously not written to the log or messages layer (only to the Console Window).

111 **New** "**WARNING 2239**" that occurs if the array limit of mass balance records retained throughout the simulation is exceeded. This can occur if the ISIS timestep is not equal to half the TUFLOW timestep, or if using the adaptive time stepping option. Previously TUFLOW would stop with a "Should not be here" message. A permanent fix to this issue is planned for a future release. Note that the mass balance summary presented at the end of the simulation may not be complete.

112 If specifying a .12da Super TIN file for the "**Read TIN Zpts ==**" command, TUFLOW would provide an unhelpful message regarding failed memory allocation – this has been rectified with a new message "**ERROR 2417** - Unable to process a 12da Super TIN file. Please use 12da single TIN files."

TUFLOW

Bug Fixes

113 Two bug fixes that would cause models to produce NaNs (Not a Number that results from, for example, a divide by zero) that relate to enhancements built into the 2012-05 release of the wetting and drying algorithm at very shallow depths. As discussed above, these fixes may cause very slight changes in results for some models.

114 Fixed a bug that would cause TUFLOW to crash if the model had **no TUFLOW 1D (ESTRY) links**, but had links to another 1D scheme such as **ISIS or XP-SWMM**. TUFLOW was always checking for any ISIS-ESTRY links even if there were no ESTRY elements, which would very occasionally cause a crash if ISIS or XP-SWMM were linked.

115 Bug fix that incorrectly included any sink/source flows (eg. rainfall) on water level boundary cells (including HX cells) in the **mass balance calculations**. Inflows to water level boundary cells are not used in the hydraulic calculations, and should not be part of the mass balance checks. The reported ME% for affected models should as a consequence stay the same or be reduced ☺.

116 Fixed a bug that would not correctly track the **maximums** if both cell cornered map output (.dat and .xmdf formats) and the new GRID (ASCII grid) output were both specified. As a consequence of this bug, the approach for tracking cell-centred output maximum water levels is now used by default for both cell-centred and cell corner output formats. This may cause a very slight lowering of maximum water levels (usually a mm or less) in isolated locations. To use the old approach, "**Maximums Approach == Method A**" can be specified in the .tcf file (the new approach is referred to as Method B).

117 Fixed a bug that would output data to _**PO.csv** files incorrectly if the first 2D domain contained no 2d_po locations.

118 Bug fix that incorrectly reported "Percentage Complete (%)" and ""Approximate Clock Time Remaining" in the **.tsf file** if the simulation start time was not equal to zero, and "**TSF Update Interval ==**" was specified.

119 If the value of a character attribute in a .dbf file contained one or more double quote characters, this would cause TUFLOW to not correctly read the attribute value. Any double quotes are now replaced by a space, and as-a-rule should not be used within attribute string fields.

Build 2012-05-AD

Bug Fix

120 Build 2012-05-AD fixes a bug in Builds 2012-05-AA to AC that affects manholes using the Engelhund approach for calculating losses. The Engelhund exit loss from the manhole, which is applied as an entrance loss to the out flowing culvert(s), was being incorrectly set to zero in Builds 2012-05-AA to AC. This is evidenced in these builds by the first loss value reported in the _TSL layer, which for outlet controlled flows will be shown as zero (see Section "4.5.2.5 Engelhund Manhole Loss Approach" in the 2010-10 TUFLOW Manual for a further description). For Build 2012-05-AD, the first value should now be non-zero, and the energy loss correctly applied.

Testing on a range of models has shown that increases in overland flood levels of 0 to 5 cm is likely for models with pipe networks using the Engelhund loss approach. Higher, very localised increases may occur due to local effects. The Engelhund exit loss is a relatively minor component of the overall losses in pipe networks, with friction (Manning's n) being the most dominant due to the length of the pipes (compared with, say, culverts under a road where entrance and exit losses typically are a larger proportion of the overall losses). Hence, the relatively minor effects on flood behaviour due to this bug.

For backward compatibility to Builds 2012-05-AA to AC, the .ecf command "Manhole Approach == METHOD B" can be specified. The bug fix is denoted by "Manhole Approach == METHOD C", the default setting for Build 2012-05-AD. "Manhole Approach == METHOD C" should be used in preference to "Manhole Approach == METHOD B", unless for legacy reasons backward compatibility is required. If using METHOD B, sensitivity testing the effects of METHOD C versus METHOD B should be carried out to check for any unacceptable differences.

Discussion on Approaches to Modelling Pipe Junction Losses

The approach taken to modelling junction and manhole losses affects the pipe network's energy losses and overland flood levels, especially if a large proportion of the flow is within the pipe network. As a general guide, the following approaches available in TUFLOW will produce the following outcomes:

(a) **Ideal Scenario:** If you are fortunate to have the details and budget to include manholes using 1d_mh layers, this will produce the most reliable results. Either the Engelhund approach or application of fixed losses based on guidance within the literature, or a combination of these, is recommended. However, in the absence of manhole details, one of the following approaches, or a combination, needs to be adopted.

(b) **Least Conservative (flatter hydraulic grade lines along pipes with lower flood levels overall):** No manholes ("Manholes at All Culvert Junctions == OFF" and no 1d_mh layers specified) and adjustment of structure losses (the default). In this case there will be minor losses at pipe junctions, because the "Structure Losses == ADJUST" default setting will adjust the entrance and exit losses at pipe connections down according to the approach/departure velocities (see Equations in Section "4.7.4.1 Adjustment of Contraction and Expansion Losses" of the 2010-10 manual). As the velocities are usually similar from one pipe to the next the entrance and exit losses are reduced downwards, often close to zero (as would be expected for a junction of two pipes with no manhole, no change in direction and no change in invert level).

(c) **Middle Ground:** Automatically create and size manholes and apply the Engelhund approach (the default setting in TUFLOW 2011-09 and 2012-05 releases). The automatic creation of manholes may be slightly conservative as a manhole (including junctions) is created at every closed node (a closed

node occurs where only culverts and pits are connected). If there are many short pipes in the network an excess of manholes may result thereby causing greater energy losses. However, if single very long culvert channels are used to represent lengths of same sized pipes, any manholes along the long pipes will not be modelled and losses will be underestimated. In this case it is generally good practice to split the long culvert channels into several channels (this will also produce an improved hydraulic grade line along the pipe).

(d) **Most Conservative (steeper hydraulic grade lines along pipes with higher flood levels overall):**
No manholes ("Manholes at All Culvert Junctions == OFF" and no 1d_mh layers specified); fixed structure losses on all culverts ("Structure Losses == FIX" is specified or 1d_nwk "F" flag is used); and the entrance and exit loss coefficients are not adjusted, for example if 0.5 and 1.0 are used. This scenario applies the full (eg. 0.5 and 1.0) entrance and exit loss coefficients at the pipe junctions, and typically produces the greatest energy losses along pipe networks, and therefore, produces the higher flood levels in upper areas.

Build 2012-05-AE

(Mandatory Update for ISIS-TUFLOW-PIPE models utilising the ESTRY-ISIS link – please see Item 125)

New Features and Enhancements

121 The popular "Read Grid Zpt ==" .tgc command has been enhanced as follows:

(a) Values are now assigned to Zpts that fall within the outer half of the DEM cells around the DEM perimeter. Previously, if processing a group of DEM tiles (ie. using multiple "Read Grid Zpt ==" commands, one for each tile), any Zpt lying within the outer half of the DEM perimeter cells were not assigned a value. The new approach assigns values to Zpts around the DEM perimeter based on an interpolation of the two DEM cells the Zpt lies nearest to. Provided tiled DEMs have a common edge (or overlapping edge), all Zpts near the DEM edges should now be assigned a value.

(b) For iSP (single precision) builds, the original method occasionally generates a NaN (not a number) at a Zpt due to precision issues. This has been fixed for this new approach.

(c) The new approach is NOT the default setting for this build so as to preserve backward compatibility. To apply specify "Grid Approach == METHOD B" prior to the "Read Grid" commands in the .tgc file. "Grid Approach == METHOD A" will activate the original approach. "Grid Approach ==" maybe used any number of times within the .tgc file with the latest setting prevailing when a "Read Grid ==" command is processed.

(d) If associated .xf files exist from previous simulations, and "Grid Approach == METHOD B" is applied, TUFLOW will automatically resample and replace the existing .xf files. The Zpt values will essentially be identical or very similar except for around the perimeter of the DEM as discussed above.

(e) The above also applies for all "Read Grid" options (eg. also applies to "Read Grid Mat ==").

(f) "Grid Approach == METHOD B" will be the default setting for the May 2013 TUFLOW release.

122 Determining when a simulation is past its peak is now easier via the following new features:
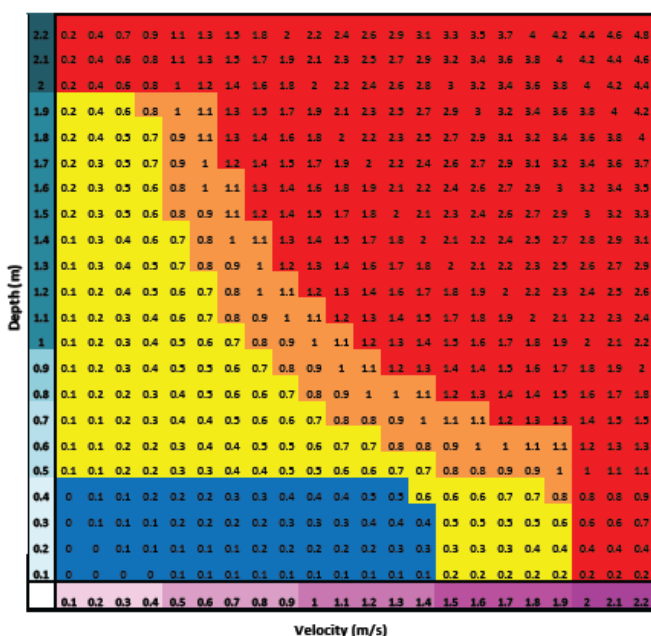
(a) New command "End After Maximum == <eam>" will terminate a simulation <eam> hours after the last time a new maximum was recorded anywhere in the model. "End Time ==" should also still be specified as an upper limit to finish the simulation. This feature is under trial and options such as specifying a tolerance or percentage of elements to determine when a simulation has reached its peak, along with automated finishing of steady-state simulations, are being investigated for the May 2013 release.

(b) If maximums are being tracked (ie. "Maximums and Minimums == ON or ON MAXIMIMS ONLY"), TUFLOW now displays additional information to the Console Window and .tlf file for each "Screen/Log Display Interval ==". At the end of each line three numbers are displayed after "Mx". The first two numbers are the percentage of 1D nodes and percentage of 2D cells that reached a new maximum in the last computational timestep. The third number is the time in decimal hours since no new maximum was recorded anywhere within the model. For example, "Mx 10 21 0.0" indicates that 10% of 1D nodes and 21% of 2D cells recorded a new maximum last timestep, and the time since the last recorded maximum is zero as would be expected. Once all 1D nodes and 2D cells have reached their maximums the third (time) value will increase above zero.

123 The Queensland Reconstruction Authority (QRA) hazard categories have been incorporated into TUFLOW's map output options.   To output and track the maximum value add "ZQRA" to the "Map Output Data Types ==" command. The hazard categories are based on the below (refer to http://qldreconstruction.org.au/u/lib/cms2/resilient-floodplains-part2-full.pdf, © *The State of Queensland (Queensland Reconstruction Authority) 2011-2012*).

Schedule 4 - Flood hazard criteria

Indicative flood hazard criteria

The following indicative flood hazard criteria have been prepared for use in preparing flood investigations (level 2), and planning evaluations based on latest available engineering guidance. In the absence of other more appropriate flood hazard definitions, the criteria below may be used.

Velocity (m/s)

- 🟥 Extreme Hazard
- 🟧 High Hazard
- 🟨 Significant Hazard
- 🟦 Low Hazard

Rules

	Low	Significant	High	Extreme
Depth	<0.5	<2	<2	2+
Velocity	<1.5	<2	<2	2+
DxV Ratio	<0.6	0.6 to <0.8	0.8 to <1.2	1.2 +

Rationale

1. **Low** – self evacuation possible for adults and children, vehicle stability within tolerance for large 4WD

2. **Significant** – working limit for trained safety workers, Vehicle evac unsuitable, Building Code limitation

3. **High** – limit of uncompromised stability for adults (dangerous to most)

4. **Extreme** – in excess of known stability limits

References

1. ARR Revision Project 10: Appropriate Safety Criteria for People
   a. Children – Significant Hazard DV ≤ 0.6 & D ≤ 0.5
   b. Adult – Moderate Hazard DV ≥ 0.6
   c. Working limit for trained safety workers or experienced and well equipped persons DV < 0.8

2. ARR Revision Project 10 State 2 Report: Appropriate Safety Criteria for Vehicles (Draft)
   a. Large 4WD DV ≤ 0.6 & D ≤ 0.5

3. Dale et al. (2004) Structural flood vulnerability and the Australianisation of Black's Curves
   a. Fibro/Tile construction D < 0.5 & V < 2
   b. Draft QDC for flood hazard areas for Deemed to Satisfy provisions – V < 1.5

4. BMT WBM (2012) Newcastle City-wide Floodplain Risk management Study and Plan P.81-82
   a. Hydraulically suitable for wading by able-bodied adults V < 2 & D < 0.8
   b. Hydraulically suitable for light construction (e.g. timber frame and brick veneer) V < 2 and D < 2

5. Jonkman et al. (2008) Methods for the estimation of loss of life due to floods: A literature review and proposal for a new method Natural Hazards P. 364
   a. Level of hazard to people can be categorized as low, moderate, significant or extreme.

TUFLOW

124 If "ZH" is specified in "Map Output Data Types ==" and the topography in the model does not change over time (ie. no variable Z shapes or morphological changes), the ZH Zpt values are output once, rather than every timestep thereby not consuming disk space unnecessarily. This feature is only available if using the XMDF format. The ZH map output will appear under a XMDF folder "Fixed". Very useful if you are comparing two or more runs that have different topography (eg. before and after scenarios), and you wish to easily view or compare the topography for each scenario within SMS or post-process using TUFLOW_to_GIS.

Bug Fixes

125 If using the ESTRY (TUFLOW 1D solver) to ISIS link via ISIS-TUFLOW-PIPE, when an ESTRY node was linked to a ISIS unit that is also linked to TUFLOW 2D HX cells, the HX cells were incorrectly linked to ESTRY and ISIS, rather than solely to ISIS. This could be observed in the 1d_to_2d_check layer as the primary and secondary 1D nodes linked to the HX cells are shown as a combination of ESTRY and ISIS nodes. Also in some instances TUFLOW would stop with unexpected messages such as "ERROR 2044 - 2D HX cell ZC level is below 1D bed level. 2D ZC Level = 40.02; 1D Bed Level = 104.6" with the elevations quoted being erroneous. This bug is now fixed and the correct linkages between ISIS and TUFLOW's 1D nodes and 2D HX cells should now occur (review by checking the Conn_1D_2D attribute in the 1d_nwk_N_check layer and the 2D HX cell connections in the 1d_to_2d_check layer).

126 For 1d_nwk M channels, the following now applies:

   (a) If the "Number_of" attribute is zero (0), one (1) will be used. Previously a value of zero would be applied and the flows in the hQh matrix were set to zero.

   (b) If the Manning's n attribute was set to zero (0), the simulation would stop with an "ERROR 1261 - Invalid Manning's n…" message. This has been corrected, and a zero n value is now accepted (M channels do not use this attribute in the hydraulic calculations).

   (c) The output of Conveyance values in the .eof and other files for M channels has been removed as it is not applicable.

127 Fixed a bug that sometimes reported a "WARNING 1317 - WLL does not cross (2 point WLL only) or snap to 1D channel." where a WLL with 4 or more vertices is correctly snapped to the vertice of a channel. If this WARNING was previously being issued, the resulting WLL triangulation may now be different. Note, if using WLLs with 4 or more vertices, one of the mid vertices must be snapped to a channel vertice; you can't snap either of the two end vertices of the WLL to the channel.

128 If specifying more than one layer for a "Read GIS BC ==" command, any CN or SC connections were not recognised in the 2nd, 3rd, …. layers. This is now fixed.

129 The volumes displayed at the end of the simulation would appear corrupted or grossly wrong if the value exceeded 2.1 trillion as can occur in high volume models, especially those using US Customary (English) units. The correct values are now reported.

130 If a 1d_nwk channel exists as a line of zero length (eg. two vertices digitised on top of each other – yes this is possible in some GIS software!), a divide by zero and NaNs occur when writing the _TS output. This bug has been fixed.

131 If the water level in a 1D node fluctuates widely, especially in a wide range (ie. a deep node) it is possible that a NaN occurs before the limit of 10 depth exceeded UNSTABLE 1998 messages are issued and the simulation is stopped. Once the NaN occurs the instability test fails and the NaNs will spread through the model. The NaNs are now trapped, an UNSTABLE 1996 or 1997 message is reported, and the simulation is stopped.