BMT WBM

TUFLOW
Flood and Tide Simulation Software

TUFLOW Build 2009-07-AA Release Notes

This document describes new features and commands for Build 2009-07-AA. This build is an initial release as some projects are in need of the new features. A final 2009 release will be made later this year upon completion and testing of other major new features such as recognition of ArcGIS .shp files and the XMDF output format.

The default settings have been left unchanged from the 2008 release, so the results from the 2008-08 and 2009-07 builds should be identical. There are a couple of known exceptions to this as discussed below for wide Z lines and possibly Z Shape TINs. 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 two data sets is an easy way to do this).

There are some enhancements that are likely to become the default settings in the final 2009 release. It is recommended that these be adopted using the commands discussed further below and listed as follows:

- HQ Boundary Approach == Method B

- Link 2D2D Approach == Method B

- Rainfall Boundaries == Stepped

- Maximum Velocity Cutoff Depth == 0.1

1   New .tcf command "HQ Boundary Approach == [{Method A} | Method B]". Although Method B is not yet the default, it is that currently recommended for new or under-development models using HQ boundaries (or old models if a slight change in results is manageable). Method A uses the method applied in prior builds. For Method B:

   (a)   the top level in the HQ curve is not the top value plus 10 (as for Method A), but the highest level in the HQ curve;

   (b)   a WARNING is issued if the top of the curve is exceeded whilst running;

   (c)   the 2d_bc d attribute can be used to specify the maximum depth to be used for generating the curve – this will stop very shallow curves being produced as these have been known to cause problems (Note: if d is less than 1m it is set to 1m).

2   Available as an option in the latter 2008-08 builds, "Link 2D2D Approach == Method B" is recommended for any models using the 2D/2D link feature (ie. 2d_bc Type "2D" lines). Method A is still set as the default, however, Method B is planned to be the default for the final 2009 release.

3   New .tcf command "Rainfall Boundaries == [STEPPED | {SMOOTHED} [{TIME CENTRED}]]".

   (a)   SMOOTHED TIME CENTRED is as per the approach used in the 2008 builds and prior releases. If TIME CENTRED is omitted the rainfall is smoothed (converted into a "hydrograph" shape curve), but the timing is such that the first rainfall occurs at the time of the first rainfall but because of the smoothing the volume of rainfall is delayed in time.

   (b)   The STEPPED option holds the rainfall constant for the time interval (ie. the rainfall has a histogram stair-step shape). This means, for example, the second rainfall value in the time-series is applied as a constant rainfall from the first time value to the second time value. As with all rainfall boundaries, the first and last rainfall entries should be set to zero (otherwise these rainfall values are applied as a constant rainfall if the simulation starts before or extends beyond the first and last time values in the rainfall time-series).

   (c)   The STEPPED option is likely to be set as the default in future releases.

4   In rare instances TINs have not been correctly created for Z Shape or Create TIN polygons. This, after much anguish, has been identified as occurring as a result of the compiler code optimisation. As a consequence, this part of the code is now compiled without optimisation (is a little slower) and may produce very slightly different Zpt values. If so, there maybe on average very slight changes to model results (fractions of a mm).

5   The buffering of Wide Z Shape Lines has been much improved to handle more complex line shapes. This may result in a different shaped buffer polygon, Zpt values and therefore results. Use the 2d_zsh_obj_check layer to view the buffer polygons.

6   In addition to the bug fixes and enhancements up to Build 2008-08-AI as listed on http://www.tuflow.com/Release_Notes_2008_08.htm, the following bug fixes have been made:

   (a)   Fixed bug that didn't apply the ADD option correctly for Read MI Shape commands.

   (b)   Fixed bug that didn't apply dz attribute if NO MERGE specified and no points snapped to region perimeter.

   (c)   ERROR/WARNING 2049 (zero Z value) not issued if using the ADD option for a Read MI Shape.

7   New .tcf command "Maximum Velocity Cutoff Depth == [<y> | {99999.}] for setting the depth above which the maximum velocity is tracked as the maximum velocity, rather than the velocity at the peak water level. If set to the default 99999., the maximum velocity will be as in previous releases, ie. based on the velocity at the peak water level. If set to 0. (zero), the maximum velocity is tracked as the maximum velocity irrespective of the velocity at the peak water level. If <y> is set to a value greater than zero, the maximum velocity is based on the velocity at the peak water level for depths below <y>, while for depths above <y>, the maximum velocity is based on the maximum velocity. A small value of <y> (eg. 0.1m) is recommended as high velocities can occur at very shallow depths during the wetting and drying process that might be misleading.

8   The new _Times.dat file contains the time the maximum velocity value was recorded (this will appear under the time of 900,002 in decimal hours in the _Times.dat file, while the time of peak water level is under time 900,001). The time of peak water level is no longer written to the _h.dat file as time 99999.1.

9   New .tcf command "Time Output Cutoff Depths == [{OFF} | <y1, y2….]". If one or more comma or space delimited depth values are specified, the _Times.dat file will contain the following time data for each depth:

   (a)   The time that a cell first experiences a depth greater than the depth(s) specified. These times are stored under the time of 100,000 + y in the .dat file, where y is y1, y2… This output is useful for mapping flood warning times for different depths of inundation.

   (b)   The duration of time that a cell is inundated above the depth(s) specified. These times are stored under the time of 200,000 + y in the .dat file, where y is y1, y2… This output is useful for mapping duration of inundation above specified depths.

   (c)   Note: For 1D WLL output, the above new _Times.dat output is not yet implemented. All 1D WLL output to the _Times.dat file is the time of maximum water level for all times in _Times.dat.

10   Hazard output extended to include up to 10 different hazard outputs per simulation, and 5 more hazard outputs have been added (Z5 to Z9) and are available for users to customise. Each hazard is tracked for its maximum if maximums output is on. The code used for calculating the hazard outputs is available from support@tuflow.com.

11   New .tcf command, "Rainfall Gauges == [One per Cell | {Unlimited per Cell}]". If each cell is only to be assigned one rainfall time-series use the "One per Cell" option. If One per Cell is set, an ERROR is issued if a 2D cell is assigned rainfall from more than one RF input (eg. from overlapping RF polygons) as it would cause a duplication of rainfall on that cell, and so is a good quality control check. Note that if the common boundary of adjoining polygons intersects exactly at a 2D cell's centre, this ERROR can be produced (slightly re-shape the polygons around the cell's centre to fix this). This option can also reduce the amount of RAM needed for direct rainfall models, as a separate grid is not needed for each RF input.

12   Two new options for "Map Output Data Types ==", BSS (Bed Shear Stress) and SP (Stream Power), are available for both 2D and 1D WLL map output. As a consequence Manning's n values are now assigned to B and W channels so that BSS and SP output can be produced (this does not affect the 1D hydraulic calculations). The equations for BSS and SP are given below where ρ is density of water (kg/m^3 – use Density of Water ==).

$$\tau_{bed} = \frac{\rho g V^2 n^2}{y^{1/3}}$$

$$Stream\,Power = |V|\,\tau_{bed}$$

13   Commas now allowed within character strings when reading .csv formats. The character string must be enclosed with quotes.

14   Build 2009-07-AA incorporates licensing of TUFLOW via Halcrow Software dongles (interim builds issued for Halcrow dongles should be replaced by this build).

15   A full 64-bit build is available for testing – please contact support@tuflow.com if interested. Presently it is only available for standalone dongles as we are yet to resolve 64-bit networking issues with the dongle suppliers.

16   Integer pointers for memory allocation have been increased from 4-bytes to 8-bytes, which was necessary for correct RAM allocation for HUGE models (these models would typically request a negative memory amount in previous builds). The 8-byte pointers allow models to have up to 9,223,372,036,854,775,807 real values rather than the 2,147,483,647 values previously using 4-byte integers. (When the pointers were first coded in 1989, it was never imagined that they would be exceeded!!!! The question now is will we ever exceed 9,223,372,036,854,775,807 values?).

17   Time varying wind stresses are now available. The undocumented .tcf "Apply Wind Stresses ==" has been disabled and should not be used. The 2d_bc "WT" (Wind vs Time) type is now available. If a 2d_bc WT type is found, wind stresses are turned on automatically. Only one WT field (digitise as a point anywhere in or out of the model) is currently recognised and is applied over the entire model. Spatially varying two or more WT time-series over a model is planned for a future release.

18   Fixed bug that did not correctly apply the background wind for a cyclone if the model was not orientated north-south/east-west (ie. Orientation Angle == 0.).

Cascade Reservoir Models (UK RIM Project):

A new option has been added to the .tbc "Read MI SA ==" command to allow the initiation of inflow hydrographs based on a flow or water level trigger so that, for example, reservoir failures can be initiated based on when the flood wave from an upstream failure reaches the reservoir so as to produce a worst case scenario. Details on the new feature are as follows:

1   An example of the new option is:

Read MI SA Trigger == mi\2d_satr_dambreaks.MIF

The 2d_satr layer is a 2d_sa layer (with one attribute, Name) to which three new attributes are added:

(a)   Trigger_Type Char(40)

(b)   Trigger_Location Char(40)

(c)   Trigger_Value Float

2   The Trigger_Type attribute must be set to "Q_" or "Flow" for a trigger based on a flow rate, or "H_" or "Level" for a trigger based on a water level.

3   The Trigger_Location attribute is the PO Label in a 2d_po layer (see Section 4.8 in the 2008 manual). The 2d_po Type attribute must also be compatible with the Trigger_Type (ie. it must include Q_ or H_).

4   The Trigger_Value attribute is the flow or water level that triggers the start of the SA hydrograph (ie. a dambreak hydrograph).

5   Every timestep, the flow or water level of the 2d_po object is monitored and once Trigger_Value is exceeded the SA hydrograph commences.

6   When digitising the SA trigger polygon, this should probably be on the downstream side of the dam wall and covering the width of the dam wall and possibly several cells thick (in the direction of flow). If there are stability issues, enlarging the SA polygon would help, but on testing thus far it's performed well with the SA polygon a few cells thick.

7   The 2d_po line (flow) or point (water level) referred to by Trigger_Location can be located anywhere in the model, but for the cascade reservoir failure modelling would usually be located across the valley at the site of the dam for a 2d_po flow line, or for a 2d_po water level point located just upstream of the dam.

8   2d_po flow lines MUST be digitised from LEFT to RIGHT looking downstream (if not, the flow across the 2d_po line will be negative and the trigger value will never be reached).

New Evacuation Route Feature

New Z Shape .tgc file command for Evacuation Route modelling, to provide output on: the level of inundation of routes; and for helping define evacuation routes; warning times; risks; and duration that routes are cut off. Details are as follows:

1   The new .tgc command is:

(a)   Read MI Z Shape Route == 2d_zshr…mif.

2   2d_zshr is the same as a 2d_zsh layer, but with the following three attributes added:

(a)   Route_Name Char(40)

(b)   Cut_Off_Type Char(40)

(c)   Cut_Off_Values Char(80)

3   Route_Name is used to label the route. A route can be split into several polylines if required provided all the lines have the same Route_Name attribute. This can be useful where the route is more easily sourced or digitised as several polylines, or if using the BRIDGE option – see further below.

4   Cut_Off_Type at present can only be "Depth", but other options such as "VxD" or "Velocity" are planned to be added. If Cut_Off_Type is blank, the value from the latest "Set Route Cut Off Type ==" command (see below) is used.

5   Cut_Off_Values is a comma delimited list of one or more values (eg. depths) used to categorise the severity of the inundation along the routes. For example, if "0.1, 0.3, 0.7" is specified, then where the water depth exceeds 0.1m, these sections of the route are assigned a Category 1; above 0.3m deep Category 2; and above 0.7m Category 3. The values should be in ascending order. Elsewhere the route is assigned Category 0 (ie. no or minimal inundation). The Route Categories are output over time in the _ER.dat file, and summarised in the _RCP.mif layer. The reason that several cut off values are specified would be for different levels of risk (eg. shallow depths would be acceptable for most vehicles and people to safely negotiate, while deeper depths would only be acceptable for higher set vehicles).

6   Unless the BRIDGE Shape_Option is specified (see further below), the Z lines carry out the usual adjustment of the Zpts based on whether they are thin, thick or wide.

7   The following two commands can be used in the .tcf and .tgc files. If used in the .tcf file, this sets the default values for the 2d_zshr Cut_Off_Type and Cut_Off_Values attributes if these attributes are left blank. The default values can be changed between different "Read MI Z Shape Route" commands by repeat usage of the same commands in the .tgc file if the user so desires.

(a)   "Set Route Cut Off Values" == 0.1, 0.3, 0.7

(b)   "Set Route Cut Off Type" == [{Depth}]

8 The _RCP.mif output layer is a layer of points showing where a route's cut off value(s) were first exceeded (eg. first point of closure). The layer contains the following attributes:

    (a)   Route_Name Char(40)
         (Name of the Route)

    (b)   Cut_Off_Value Float
         (Cut Off Value)

    (c)   First_Cut_Off_Time Float
         (Simulation time in hours when the cut off value was first exceeded)

    (d)   Last_Cut_Off_Time Float
         (Simulation time in hours when the cut off value last first exceeded)

    (e)   Duration_Cut_Off Float
         (Duration in hours that the cut off value was exceeded – not necessarily the difference between the first and last cut off times if the route reopened during this time)

9 The _RC.dat output file contains the Route Categories over time. This file can be used to view and animate the route category values.

10 If a Z Shape Route line is given a Shape_Option of "BRIDGE", the elevations along the Z Shape line are NOT used to adjust the Zpts, but are only used for assigning the evacuation route categories, therefore, the Z Shape Line can cross a river without blocking it! Because a route can be comprised of as many lines as you like as long as each line is given the same Route_Name they will all be regarded as being part of the same route. Therefore for a bridge, split this line from the lines either side and give all three lines the same Route Name, with the line crossing the river given a Shape_Option of "BRIDGE".


Manning's n vs Depth Curves

1 Curves of Manning's n versus depth are specified via the new Materials File .csv format.

2 Reading of materials files has been reworked, as follows.

    (a)   More than one materials file may now be specified using any number of "Read Materials File ==" commands.

    (b)   Both .tmf and the new .csv formats are supported.

    (c)   Any combination of .tmf and .csv files can be used.

    (d)   Up to 100 materials in total are allowed.

3 To use the new .csv format, manage the database of materials and curves in a .xls file in a similar manner to BC databases, and use the TUFLOW Tools.xls macros to export to .csv. If a material has an n vs depth curve, it must be in the .csv format.

4 By default, when the n vs depth values are read they are assumed to be in the first two columns of the .csv file containing the data. However, you can specify the columns to be used by placing the column label after the .csv filename. If only one column label is given, this is assumed to be the depth column and the next column along must be the n column. As for boundary data, TUFLOW searches down the rows until it finds the first numeric values and will start reading from this row until a row is found with no more numeric values.

5 Integer numbers must still be used for Material IDs, although work has commenced to handle non-numeric Material IDs

6 Use the "n" option for "Map Output Data Type ==" to see how the n values vary over time.

7 The third column of the .csv format is reserved for the infiltration parameters. At present, only the initial loss/continuing loss option is available and is entered as two comma delimited numbers.

8 The fourth column is reserved for a new feature being developed that allows the user to allocate a Landuse Hazard ID that can be used in deriving the Z1, Z2, Z3… Hazard output. This will allow the user to generate hazard categories not only on velocity and depth, but also land-use. Different hazard formulae will be able to be used for different materials, for example urban areas may have a different formula to rural areas. Note, this feature is not yet fully implemented.

9 To give a description of the material, this must be done after all inputs for that material, and must be preceded by a "!" or "#". Whilst Build 2009-07-AA only reads the first four columns, future releases may read more columns, hence the necessity to use a "!" or "#" to preserve backward compatibility.

In the example below:

- Material 1 would have a constant n value of 0.03 and no infiltration parameters.

- Material 5 would vary n with depth using the four y1, n1, y2, n2 values (as per .tmf format approach) and no infiltration parameters.

- Materials 11 and 12 will source n vs y curves from Grass.csv. As only one column label has been specified, the y values must occur under that label and the n values must occur in the next adjoining column (see image of sheet further below). An IL of 10mm and CL of 2mm/h for both materials will be used for any direct rainfall.

- Materials 21 and 22 will source n vs y curves from Trees.csv. As two column labels have been specified, the y and n values must occur under the specified labels (see image of sheet further below). An IL of 20mm and CL of 2mm/h for Material 21, and 25 and 2 for Material 22, will be used.

- Material 31 will extract n vs y values using Column A of Pavement.csv for y values and Column B for n values. IL = 0 and CL = 0.
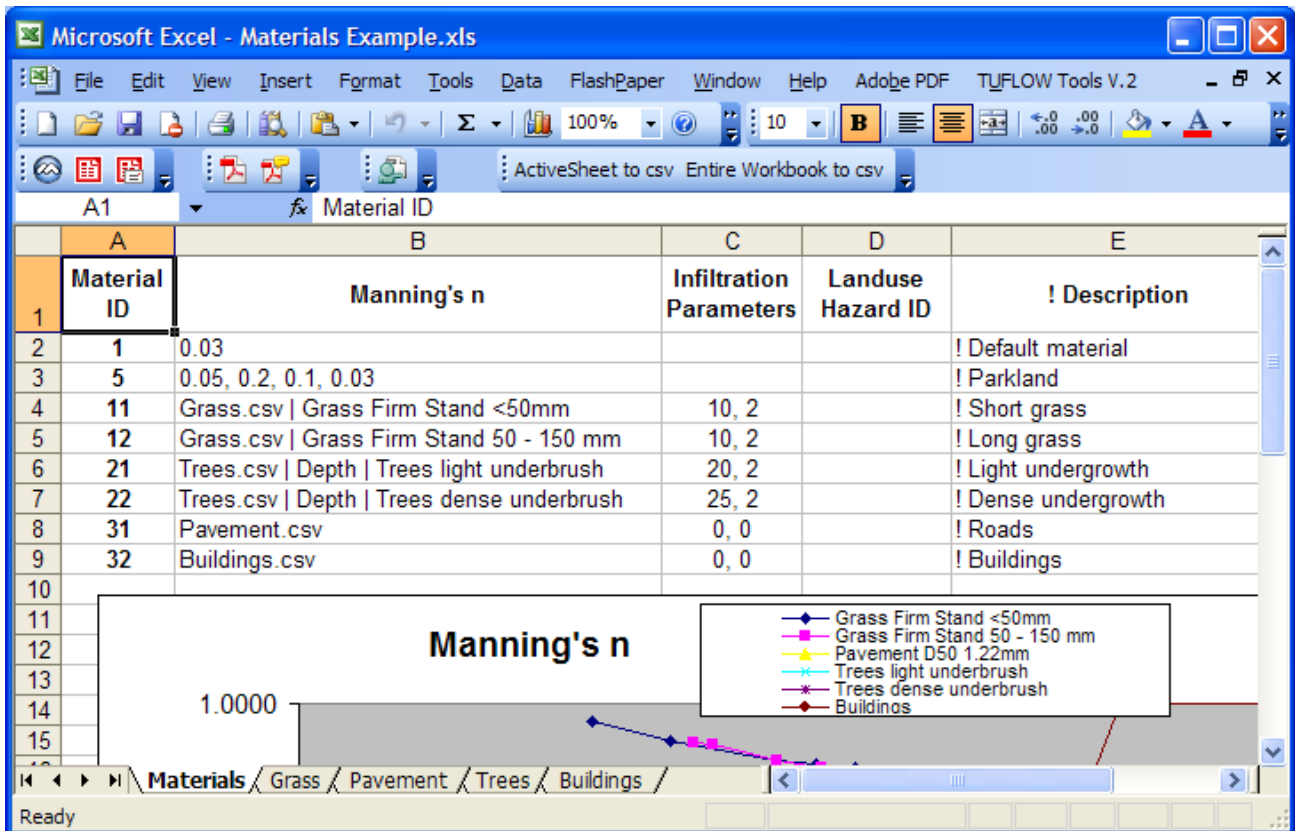
Microsoft Excel - Materials Example.xls

A29

	A	B	C	D	E	F	G
1	Grass Firm Stand <50mm			Grass Firm Stand 50 - 150 mm			
2	y (m)	n		y (m)	n		
3	0.0077	0.8800		0.0155	0.7530		
4	0.0132	0.7650		0.0177	0.7400		1.0000
5	0.0277	0.6660		0.0277	0.6660		
6	0.0360	0.6460		0.0377	0.6310		
7	0.0477	0.6260		0.0527	0.5947		

Materials \ Grass \ Pavement \ Trees \ Buildings

Ready

Microsoft Excel - Materials Example.xls

A1   Depth

	A	B	C	D	E	F	G
1	Depth	Trees light underbrush	Trees dense underbrush				
2	y (m)	n	n				
3	0.030	0.480	0.800	US FHA Urban Drainage Manual			
4	0.060	0.310	0.600				
5	0.090	0.194	0.360				

Materials / Grass / Pavement \ Trees / Buildings

Ready

Microsoft Excel - Materials Example.xls

A1   Pavement D50 1.22mm

	A	B	C	D	E	F	G	H	I
1	Pavement D50 1.22mm								
2	y (m)	n							
3	0.002	0.400786834							
4	0.00309	0.074916123							
5	0.00331	0.059524983							

Materials \ Grass / Pavement / Trees / Buildings

Ready

M Channels (User Defined Flow Matrix)

This new feature allows the modeller to define the flow through a channel based on a flow matrix specified by the user. To set up user-defined flow matrix 1D channels in TUFLOW follow the steps below.

1   In the 1d_nwk layer, the following attributes are required:

(a)   ID = ID of the channel

(b)   Type = "M" (M stands for Matrix)

(c)   US_Invert = Invert level of channel

(d)   DS_Invert (optional) - Invert level of channel is taken as the maximum of US_Invert and DS_Invert

(e)   Inlet_Type (formerly Branch) = relative path to flow matrix file (must be a .csv file), and optionally flow area matrix file

(f)   pBlockage = Percentage blockage of channel (only used if no flow area matrix is provided)

(g)   Width = Width of channel (only used if no flow area matrix is provided)

(h)   Number_of = Number of channels (flow and area matrices are multiplied by this value). If zero is set to one.

2   Create the flow matrix along the lines of the image below and export the file to .csv using Save csv tool. The .csv file is that referenced in the 1d_nwk Inlet_Type attribute above. Notes using the example in the image further below are:

(a)   TUFLOW searches through the sheet until more than 3 numbers are found at the beginning of a row (Row 3 in the example).

(b)   This first row contains a multiplication factor (in A3) then upstream depth values (in the direction the channel is digitised). The depth values are added to the channel invert to set the water level.

(c)   The next rows have the downstream depth in Column A and then the flows.

(d)   Note that at present the matrix must be square and that the u/s and d/s depths must be the same values. The flows along the diagonal must be zero, and to the left of the diagonal negative (or zero) and to the right positive (or zero).

3   Optionally create a flow area matrix of the same dimensions and depth values as for the flow matrix. Note:

(a)   The path to the area.csv file is specified after the flow .csv file in the Inlet_Type attribute (separate the two filenames using a "|"; eg. "..\UD_Q.csv | ..\UD_A.csv").

(b)   The factor value in the A3 cell is not used (but the one from the flow matrix is used to factor the areas).

(c)   The area values are only used for outputting the channel velocity (they are not used for the hydraulic computations other than when the channel velocity is used for other channels, eg. adjusting structure losses).

(d)   If an area matrix is not provided, the area used is the Width times the pBlockage times the average of the upstream and downstream depths.

WV Channels (Variable Geometry Weir)

The new V option for W (weir) channels allows the modeller to vary the cross-section geometry of a weir using a trapezoidal shape. To set up a WV channel in TUFLOW follow the steps below.

1 In the 1d_nwk layer, the following attributes are required:

 (a) ID = ID of the channel

 (b) Type = "WV"

 (c) Len_or_ANA = Nominal length in m (only used for calculating nodal storage if UCS is on)

 (d) US_Invert = -99999 (the invert level is specified in the .csv file discussed below)

 (e) DS_Invert = -99999 (the invert level is specified in the .csv file discussed below)

 (f) Inlet_Type (formerly Branch) = relative path to a .csv file containing information on how the weir geometry varies.

 (g) Height_Cont = Trigger Value (the upstream water level to trigger the start of the failure; upstream water level is determined as the higher water level of the upstream and downstream nodes).

2 The .csv file must be structured as follows (also see example below):

 (a) TUFLOW searches through the sheet until more than 4 numbers are found at the beginning of a row (Row 2 in the example below).

 (b) Each row of values is read until the end of the file or a row with no or less than four numbers is found. There is no limit on the number of rows of data.

 (c) The four columns must be as follows and in this order. The labels for the columns are optional.

  (i) Time from start of breach in hours.

  (ii) Weir bed level in metres.

  (iii) Weir bed width in metres.

  (iv) Side slope (enter as the vertical distance in metres for one metre horizontal). For example, a value of 0.5 means a slope of two horizontal to one vertical.

3   In the example below, the weir once triggered will erode from a bed level of 270m to 254m, widen from a bed width of 0 to 20m and the side slope will remain constant at 0.5. The period of time for the erosion is 0.5hours.

4   Although in most cases the weir is eroded, the weir can also be raised/accreted as well or a combination of the two. Simply enter the change over time using as many rows as needed.

	Time from Breach (h)	Bed Level (m)	Bed Width (m)	Side Slope (1 horizontal to X vertical)	
1					
2	0	270	0	0.5	
3	0.5	254	20	0.5	
4					
5					