

TUFLOW

User Manual

GIS Based 2D/1D
Hydrodynamic Modelling

2010 (Build 2010-10-AB)

www.TUFLOW.com

www.TUFLOW.com/forum

support@tuflow.com

[New Features/Changes](#)

[How to Use This Manual](#)

[Chapters](#)

[Table of Contents](#)

[List of Figures](#)

[List of Tables](#)

[Appendices](#)

[.tcf File Commands](#)

[.tgc File Commands](#)

[.tbc File Commands](#)

[.tef File Commands](#)

[.ecf File Commands](#)

[Command Hyperlinks](#)

[Glossary & Notation](#)

TUFLOW 



This manual has been superseded



This manual has been **superseded** and is no longer the most up-to-date version. For the latest information, guidance, and updates, please refer to the current [**TUFLOW Classic/HPC**](#) manual.

We recommend using the latest version to ensure access to the most accurate and supported documentation.

<https://www.tuflow.com/Downloads>

Contents

Contents	i
How to Use This Manual	ii
About This Manual	ii
Chapters	iii
Table of Contents	iv
Appendices	xi
List of Figures	xiii
List of Tables	xiv
Glossary & Notation	xvi

How to Use This Manual

This manual is designed for both hardcopy and digital usage. It was created using Microsoft Word 2007, and has not been tested in its digital mode in other platforms.

Section, table and figure references are hyperlinked (click on the Section, Table or Figure number in the text to move to the relevant page). In later versions of Microsoft Word, you may have to hold the Ctrl key down to hyperlink.

Similarly, and most importantly, text file commands are hyperlinked and are easily accessed through the lists at the end of the manual (see [.tcf File Commands](#); [.ecf File Commands](#); [.tef File Commands](#); [.tgc File Commands](#) and [.tbc File Commands](#)). To quickly go to the end of the manual press Ctrl End. There are also command hyperlinks in the text (normally blue and underlined). Command text can be copied and pasted into the text files to ensure correct spelling.

Some useful keys to navigate backwards and forwards are Alt Left / Right arrow to go backwards / forwards to the last locations. Ctrl Home returns to the front page, which contains useful hyperlinks. Also, Ctrl End provides quick access to the end pages, which contain all the hyperlinks to the text file commands.

Any constructive suggestions are very welcome (<mailto:support@tuflow.com>).

About This Manual

This manual is a User Manual for the TUFLOW.exe (and ESTRY.exe) hydrodynamic computational engines. These engines are driven through a Console (DOS) Window and rely on third party software to provide the interface to the user and the engines. These software are typically a text editor (eg. UltraEdit), GIS platform (eg. MapInfo), 3D surface modelling software (eg. Vertical Mapper) and result viewing (eg. SMS). Please refer to the user documentation or help for the third party software you have chosen to use in addition to this manual.

Of particular note for existing users is that documentation shaded with a pale blue represents the new or modified text and sections of the manual for the 2010-10 and 2009-07 releases and in pale green for the 2008-08 release. We can't guarantee that all of the key changes and additions are shaded, however, hopefully we managed to identify most of them!

NOTE: All of the Read MI and Read MID commands in the manual have been replaced by Read GIS and Read RowCol. Read MI and Read MID remain valid commands and will do so indefinitely, so they can continue to be used. You can even use Read MI to read a .shp file if you wish! Read GIS is identical to Read MI, and Read RowCol is identical to Read MID. The Read GIS and Read RowCol syntax was adopted with the recognition of both .mif and .shp formats, and to make the intent of these commands clearer.

Chapters

1	INTRODUCTION	1-1
2	OVERVIEW	2-1
3	THE MODELLING PROCESS	3-1
4	DATA INPUT	4-1
5	RUNNING TUFLOW	5-1
6	2D/1D MODEL DEVELOPMENT	6-1
7	DATA OUTPUT	7-1
8	QUALITY CONTROL	8-1
9	TROUBLESHOOTING	9-1
10	NEW FEATURES AND CHANGES IN 2010-10 AND 2009-07	10-1
11	UTILITIES	11-1
12	TIPS AND TRICKS	12-1
13	REFERENCES	13-1
Appendix A	.tcf File Commands	A-1
Appendix B	.ecf File Commands	B-1
Appendix C	.tgc File Commands	C-1
Appendix D	.tbc File Commands	D-1
Appendix E	.tef File Commands	E-1
Appendix F	New Features and Changes for Past Releases	F-1
Appendix G	Command Hyperlinks	G-1

Table of Contents

1	INTRODUCTION	1-1
1.1	TUFLOW	1-2
1.2	ESTRY	1-4
2	OVERVIEW	2-1
2.1	Software Structure	2-2
2.2	Data Input	2-3
2.2.1	Structure	2-3
2.2.2	Suggested Folder Structure	2-5
2.2.3	File Types and Naming Conventions	2-6
2.2.4	GIS Input File Types and Naming Conventions	2-10
2.3	Performing Simulations	2-14
2.4	Data Output	2-14
2.5	Limitations and Recommendations	2-15
3	THE MODELLING PROCESS	3-1
3.1	Is a 2D or 2D/1D Model Feasible?	3-2
3.2	Linking 1D and 2D Domains	3-3
3.3	Data Requirements	3-6
3.4	Calibration and Sensitivity	3-6
3.5	Model Resolution	3-7
3.5.1	2D Cell Size	3-7
3.5.2	1D Network Definition	3-7
3.6	Computational Timestep	3-8
3.6.1	2D Domains (Courant Number)	3-8
3.6.2	1D Domains (Courant Number)	3-8
3.6.3	2D/1D Models	3-9
3.7	Eddy Viscosity	3-10
4	DATA INPUT	4-1
4.1	Control Files – Rules and Notation	4-4
4.2	Simulation Control Files	4-6
4.2.1	TUFLOW Control File (.tcf File)	4-6
4.2.2	1D Domains or ESTRY.exe Control File (.ecf File)	4-8

4.2.3	Run Time And Output Controls	4-9
4.3	GIS Layers	4-10
4.3.1	"GIS" or "MI" Commands	4-10
4.3.2	"RowCol" or "MID" Commands	4-11
4.4	2D Domains (.tgc File)	4-13
4.4.1	The .tgc (Geometry Control) File	4-13
4.4.2	2D Grid Orientation and Dimensions	4-14
4.4.3	2D Cell Codes	4-14
4.4.4	ZC, ZU, ZV and ZH Points (Zpts)	4-15
4.4.5	Building the Topography Layer by Layer	4-17
4.4.6	Zpt (2d_zpt) Layers	4-20
4.4.7	3D Breakline (2d_zln) Layers	4-21
4.4.8	3D TIN (2d_ztin) Layers	4-21
4.4.8.1	<i>Haul Road Example of Using Create TIN Zpts</i>	4-23
4.4.8.2	<i>Highway Embankment Removal Example</i>	4-26
4.4.9	Z Shape (2d_zsh) Layers	4-28
4.4.10	Variable Z Shape (2d_vzsh) Layers	4-33
4.4.10.1	<i>Variable Z Shape Example: Breaching of an Embankment</i>	4-35
4.4.11	Using Multiple Layers and Points Layers	4-39
4.4.11.1	<i>Points Only Layers</i>	4-39
4.4.12	Bed Resistance (Materials)	4-41
4.4.12.1	<i>Materials Files</i>	4-42
4.4.12.2	<i>.tmf Format</i>	4-42
4.4.12.3	<i>.csv Format (Manning's n vs Depth Curves)</i>	4-43
4.4.13	Multiple 2D Domains	4-46
4.5	1D Domains (Networks)	4-48
4.5.1	Nodes and Pits (1d_nwk Layers)	4-48
4.5.1.1	<i>Pit Channels</i>	4-49
4.5.1.2	<i>Pit Inlet Database</i>	4-50
4.5.1.3	<i>Connecting Pits and Nodes to 2D Domains</i>	4-52
4.5.2	Manholes (Automatic and 1d_mh Layer)	4-54
4.5.2.1	<i>Automatically Assigned Manholes</i>	4-54
4.5.2.2	<i>Manually Assigned Manholes (1d_mh Layer)</i>	4-55
4.5.2.3	<i>Digitising Culverts Connected to Manholes</i>	4-57
4.5.2.4	<i>Fixed Manhole Loss Approach</i>	4-57
4.5.2.5	<i>Engelund Manhole Loss Approach</i>	4-57
4.5.3	Channels	4-60

4.5.4	1d_nwk Attributes	4-64
4.5.5	How are Nodes and Channels Processed?	4-74
4.6	1D Topography	4-75
4.6.1	Channel Hydraulic Properties (CS) Tables	4-75
4.6.2	Node Storage (NA) Tables	4-77
4.6.2.1	<i>Storage (NA) Tables</i>	4-77
4.6.2.2	<i>Using Channel Widths</i>	4-77
4.6.2.3	<i>Storage Above Structure Obverts</i>	4-78
4.6.2.4	<i>Procedure for Assigning NA Tables</i>	4-78
4.6.3	Free-form Tabular Input (1d_tab, 1d_xs, 1d_na, 1d_bg Layers)	4-79
4.6.4	End Cross-Sections	4-82
4.6.5	Interpolated Cross-Sections	4-82
4.6.6	XZ Relative Resistances	4-84
4.6.6.1	<i>Relative Resistance Factor (R)</i>	4-84
4.6.6.2	<i>Material Values (M)</i>	4-84
4.6.6.3	<i>Manning's n Values (N)</i>	4-85
4.6.6.4	<i>Position Flag (P)</i>	4-85
4.6.7	Reducing Conveyance with Height	4-85
4.6.8	Effective Area versus Total Area	4-85
4.7	Hydraulic Structures and Supercritical Flow	4-87
4.7.1	How to Model Bridges and Box Culverts	4-87
4.7.2	2D Flow Constrictions (FC)	4-91
4.7.2.1	<i>2d_fcsh and 2d_fc Layers</i>	4-91
4.7.2.2	<i>Applying FC Attributes</i>	4-96
4.7.2.3	<i>Layered Flow Constrictions (2d_lfcsh Layers)</i>	4-99
4.7.3	2D Upstream Controlled Flow (Weirs and Supercritical Flow)	4-102
4.7.4	1D Hydraulic Structures	4-103
4.7.4.1	<i>Adjustment of Contraction and Expansion Losses</i>	4-103
4.7.4.2	<i>Bridges</i>	4-105
4.7.4.3	<i>Culverts and Pipes</i>	4-106
4.7.4.4	<i>Weirs</i>	4-111
4.7.4.5	<i>Variable Geometry Channels</i>	4-112
4.7.4.6	<i>Non-Inertial Channels</i>	4-112
4.7.4.7	<i>M Channels (User Defined Flow Matrix)</i>	4-113
4.7.4.8	<i>WV Channels (Variable Geometry Weir)</i>	4-114
4.8	Customised Output	4-116
4.8.1	Time-Series Output (2d_po, 2d_lp) from 2D Domains	4-116

4.8.2	Evacuation Routes (2d_zshr)	4-119
4.9	Initial Water Levels (IWL) and Restart Files	4-127
4.9.1	2D Domains	4-127
4.9.2	1D Domains	4-128
4.10	Boundary Conditions and Linking 2D/1D Models	4-129
4.10.1	Boundary Condition (BC) Database	4-129
4.10.2	BC Database Example	4-132
4.10.3	Using the BC Event Name Command	4-134
4.10.4	Recommended BC Arrangements	4-136
4.10.5	Linking 1D and 2D Domains	4-138
4.10.5.1	Linking ESTRY (TUFLOW) 1D Domains	4-138
4.10.5.2	Linking ISIS 1D Domains	4-138
4.10.6	1d_bc Layers	4-141
4.10.7	2d_bc, 2d_sa and 2d_rf Layers	4-144
4.10.7.1	2d_sa Trigger Option	4-145
4.11	Presenting 1D Domains in 2D Output (1d_wll)	4-158
4.11.1	WLL Method A	4-158
4.11.2	WLL Method B	4-159
4.11.2.1	Water Level Lines (WLL)	4-159
4.11.2.2	Water Level Line Points (WLLp)	4-160
4.12	Managing Multiple Events and Scenarios	4-162
4.12.1	Modelling Multiple Events Using an Event File	4-162
4.12.2	Modelling Different Scenarios Using If Scenario	4-166
4.13	Data Processing	4-169
4.13.1	Hierarchy	4-169
4.13.2	XF Files	4-170
4.14	UltraEdit	4-172
5	RUNNING TUFLOW	5-1
5.1	Installing a Dongle	5-2
5.1.1	BMT WBM Dongles	5-2
5.1.2	Protocols for Accessing Dongles	5-3
5.1.3	"C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File	5-3
5.1.4	WIBU Codemeter (Metal) Dongles (Local and Network)	5-4
5.1.4.1	Installing Codemeter Software	5-4
5.1.4.2	Codemeter Features	5-7
5.1.4.3	Requesting a Licence Change	5-8

5.1.5	SoftLok (Blue) Standalone (Local) Dongle	5-11
5.1.6	SoftLok (Blue) Network Dongle	5-11
5.1.6.1	SoftLok Dongle Security Server	5-12
5.1.6.2	Client Computers	5-14
5.1.7	Dongle Failure During a Simulation	5-15
5.2	TUFLOW.exe and .dll Files	5-16
5.2.1	Using TUFLOW with ISIS or XP-SWMM, or from SMS	5-19
5.3	TUFLOW.exe Options (Switches)	5-20
5.4	via Right Mouse Button in Microsoft Explorer	5-23
5.5	From UltraEdit	5-25
5.6	Batching Simulations using a Batch File	5-28
5.6.1	Simple Example and Switches	5-28
5.6.2	Windows NT/2000/XP/7 Priority Levels	5-29
5.6.3	Advanced .bat Files	5-30
5.7	From a Console (DOS) Window	5-31
5.8	The Console (DOS) Window Does Not Appear!	5-31
5.9	Customising TUFLOW using TUFLOW_USER_DEFINED.dll	5-31
5.10	ESTRY.exe	5-32
6	2D/1D MODEL DEVELOPMENT	6-1
6.1	Tutorial Model	6-2
6.2	Setting up a New Model	6-2
7	DATA OUTPUT	7-1
7.1	General	7-2
7.1.1	Console (DOS) Window Display	7-2
7.1.1.1	Console Window Shortcut Keys (Ctrl-C and Ctrl-S)	7-4
7.1.1.2	Customisation of Console Window	7-4
7.1.2	Message Boxes	7-7
7.1.3	_ TUFLOW Simulations.log Files	7-8
7.1.3.1	Local .log File	7-8
7.1.3.2	Global .log File	7-10
7.2	Check Files	7-11
7.2.1	Simulation Log File (.tlf or .elf file)	7-11
7.2.2	ERROR, WARNING and CHECK Messages (in .tlf and _messages.mif Files)	7-11
7.2.3	.wor File	7-12

7.2.4	.eof File	7-12
7.2.5	Using the Write Check Files Command	7-13
7.3	2D Domains	7-18
7.3.1	SMS (Map) Output (.dat or .xmdf Files)	7-18
7.3.2	Time-Series Output	7-27
7.3.3	Conversion to GIS Formats	7-27
7.4	1D Domains	7-28
7.4.1	Output File (.eof file)	7-28
7.4.2	SMS Output	7-29
7.4.3	GIS and Text 1D Domain Check Files	7-30
7.4.4	Time-Series Output	7-30
7.4.4.1	_TS.mif Layers	7-30
7.4.4.2	_ccA.mif Layer	7-32
7.4.5	Maximum/Minimum Output	7-33
7.5	Mass Balance Output	7-34
7.6	Simulation Summary – Healthy Model Indicators	7-42
8	QUALITY CONTROL	8-1
8.1	Unhealthy Models	8-2
8.1.1	Timestep	8-3
8.1.2	Tips for an Unhealthy 2D Domain	8-4
8.1.3	Tips for an Unhealthy 1D Domain	8-6
8.1.4	Tips for Unhealthy 1D/2D Links	8-9
8.2	Check List	8-11
9	TROUBLESHOOTING	9-1
9.1	General Comments	9-2
9.2	Suggestions and Recommendations	9-2
9.3	Large Models (Exceeding RAM)	9-3
9.4	Identifying the Start of an Instability	9-4
9.5	Why Do I Get Different Results?	9-5
10	NEW FEATURES AND CHANGES IN 2010-10 AND 2009-07	10-1
11	UTILITIES	11-1
11.1	Running DOS .exe Utilities	11-2
11.1.1	Using the Right Mouse Button	11-2

11.1.2	Using Batch (.bat) Files	11-4
11.2	TUFLOW_to_GIS.exe	11-5
11.3	dat_to_dat.exe	11-9
11.4	tin_to_tin.exe	11-12
11.5	12da_to_from_mif.exe	11-14
11.6	asc_to_asc.exe	11-16
11.7	convert_to_ts1.exe	11-17
11.8	xsGenerator.exe	11-19
11.9	TUFLOW_Tools.xls	11-21
11.10	MapInfo Tools	11-21
 12	 TIPS AND TRICKS	 12-1
12.1	MapInfo Tips	12-2
12.1.1	Move Snapped Nodes Switch in MapInfo	12-2
12.1.2	Setting Default Style from an Existing Layer	12-3
12.1.3	Digitising Common Polygon Boundaries	12-3
12.1.4	Setting Number of Undo Objects	12-3
12.1.5	Display Nodes (Vertices) on Polygons and Lines	12-4
12.1.6	Labelling Numbers using Fixed Number Decimal Places	12-5
12.1.7	Selecting Subsets and Concatenating Labels	12-7
12.1.8	Using MapInfo String Functions	12-9
12.1.8.1	<i>Example of Using String Functions</i>	<i>12-10</i>
12.2	UltraEdit Tips	12-16
12.3	Vertical Mapper Tips	12-17
12.3.1	Creating High Quality Flood Extent Maps	12-17
12.4	Microsoft Excel Tips	12-20
12.4.1	Save csv and Using .xls File to Manage Databases	12-20
12.5	DOS Window Tips	12-22
12.5.1	Copying and Pasting in a Console (DOS) Window	12-22
 13	 REFERENCES	 13-1

Appendices

Appendix A	.tcf File Commands	A-1
A.1	Geographic Reference Commands (.tcf)	A-1
A.2	File Management Commands (.tcf)	A-3
A.3	Simulation Time Control Commands (.tcf)	A-9
A.4	Output Control and Format Commands (.tcf)	A-11
A.5	Bed Resistance Commands (.tcf)	A-21
A.6	Flow Constriction (FC) Commands (.tcf)	A-24
A.7	Time-Series Output (PO & LP) Commands (.tcf)	A-25
A.8	Initial Water Level (IWL) Commands (.tcf)	A-27
A.9	Restart File Commands (.tcf)	A-28
A.10	Wetting and Drying Commands (.tcf)	A-29
A.11	Supercritical and Weir Flow Commands (.tcf)	A-32
A.12	Eddy Viscosity Commands (.tcf)	A-35
A.13	Miscellaneous Commands (.tcf)	A-36
A.14	Water Level Instability Detection Commands (.tcf)	A-44
A.15	Boundary Condition Commands (.tcf)	A-46
A.16	Boundary Treatment Commands (.tcf)	A-49
A.17	External 1D Schemes Commands (.tcf)	A-54
A.18	Cyclone/Hurricane and Wind Commands (.tcf)	A-56
A.19	Wave Radiation Stress Commands (.tcf)	A-59
Appendix B	.ecf File Commands	B-1
B.1	Geographic Reference Commands (.ecf)	B-2
B.2	File Management Commands (.ecf)	B-2
B.3	Simulation Control Commands (.ecf)	B-5
B.4	Output Control and Format Commands (.ecf)	B-11
B.5	Model Network and Topography Commands (.ecf)	B-15
B.6	Accessing Fixed Field Data Commands (.ecf)	B-24
B.7	Initial Water Level (IWL) Commands (.ecf)	B-26
B.8	Boundary Condition Commands (.ecf)	B-27
B.9	Manhole Commands (.ecf)	B-29
Appendix C	.tgc File Commands	C-1
C.1	Grid Size, Location and Orientation Commands (.tgc)	C-2
C.2	Reading External Formats (.tgc)	C-4
C.3	Model Grid Commands (.tgc)	C-5
C.4	Model Bathymetry / Elevation Commands (.tgc)	C-9
C.5	Other Commands (.tgc)	C-20
Appendix D	.tbc File Commands	D-1
D.1	Boundary Condition Commands (.tbc)	D-2
Appendix E	.tef File Commands	E-1
E.1	Event File Commands (.tef)	E-2

Appendix F	New Features and Changes for Past Releases	F-1
F.1	2008-08 Release	F-2
F.2	2007-07 Release	F-13
F.3	2006-06 Release	F-24
F.4	2005-05 Release	F-33
F.5	Builds 2004-06-AC to 2001-03-AA	F-37
Appendix G	Command Hyperlinks	G-1

List of Figures

Figure 2-1	TUFLOW Data Input and Output Structure	2-4
Figure 3-1	Example of a Poor Representation of a Narrow Channel in a 2D Model	3-3
Figure 3-2	1D/2D Linking Mechanisms	3-4
Figure 3-3	Modelling a Pipe System in 1D underneath a 2D Domain	3-5
Figure 3-4	Modelling a Channel in 1D and the Floodplain in 2D	3-5
Figure 3-5	Effect of Enhanced Dry Boundary Viscosity Term Treatment	3-11
Figure 4-1	Location of Zpts and Computation Points	4-16
Figure 4-2	Example of Using a 2d_ztin or 2d_zsh Layer to Remove a Highway Embankment	4-27
Figure 4-3	Example of Materials .csv File Format	4-46
Figure 4-4	Different Flow Patterns from 2D FCs and 1D/2D Links when Modelling a Submerged Culvert	4-90
Figure 4-5	Setting FC Parameters for a Bridge Structure	4-98
Figure 4-6	1D Inlet Control Culvert Flow Regimes	4-109
Figure 4-7	1D Outlet Control Culvert Flow Regimes	4-110
Figure 4-8	Interpretation of PO Objects and SMS Output	4-126
Figure 4-9	Examples of 2D HX Links to 1D Nodes	4-140
Figure 7-1	Viewing Time-Series Data in MapInfo – Checking Flow Balance in a 2D/1D Model	7-31
Figure 7-2	Example of the _ccA.mif Layer Showing Culvert Performance	7-32
Figure 8-1	Thematic Map Example of _TSMB.mif Layer of a 1D Domain	8-7
Figure 8-2	Thematic Map Example of _TSMB1d2d.mif Layer of 1D/2D HX Links	8-9

List of Tables

Table 2.1	Recommended Sub-Folder Structure	2-5
Table 2.2	List of Most Commonly Used File Types	2-7
Table 2.3	GIS Input Data Layers and Recommended Prefixes	2-11
Table 4.1	Reserved Characters – Text Files	4-4
Table 4.2	Notation Used in Command Documentation – Text Files	4-5
Table 4.3	TUFLOW Interpretation of GIS Objects	4-12
Table 4.4	Cell Codes	4-15
Table 4.5	2D Zpt Commands	4-18
Table 4.6	2d_zpt Attribute Descriptions	4-20
Table 4.7	2d_ztin Attribute Descriptions	4-22
Table 4.8	Z Shape (2d_zsh) Attribute Descriptions	4-31
Table 4.9	Variable Z Shape (2d_vzsh) Attribute Descriptions	4-34
Table 4.10	..._pts Attribute Descriptions	4-40
Table 4.11	Q Pit Flow Regimes	4-52
Table 4.12	1D Manhole (1d_mh) Attribute Descriptions	4-55
Table 4.13	1D Channel and Node Types	4-61
Table 4.14	1D Model Network (1d_nwk) Attribute Descriptions	4-64
Table 4.15	1D Model Network (1d_nwke) OPTIONAL Attribute Descriptions	4-73
Table 4.16	Channel Cross-Section Hydraulic Properties	4-76
Table 4.17	1D Table Links (1d_tab) Attributes	4-80
Table 4.18	Hydraulic Structure Modelling Approaches	4-88
Table 4.19	Flow Constriction Shape (2d_fcsh) Attribute Descriptions	4-92
Table 4.20	Flow Constriction (2d_fc) Attribute Descriptions	4-94
Table 4.21	Layered Flow Constriction Shape (2d_lfcsh) Attribute Descriptions	4-100
Table 4.22	Layered Flow Constriction Shape (2d_lfcsh..._pts) Attribute Descriptions	4-101
Table 4.23	1D Culvert Flow Regimes	4-108
Table 4.24	Time-Series (PO) Data Types	4-121
Table 4.25	Plot Output (2d_po) Attribute Descriptions	4-125
Table 4.26	2d_iwl Attributes	4-128
Table 4.27	1D Initial Water Level (1d_iwl) Attributes	4-128
Table 4.28	BC Database Keyword Descriptions	4-130
Table 4.29	Recommended BC Arrangements	4-137
Table 4.30	1D Boundary Condition and Link Types	4-141
Table 4.31	1D Boundary Conditions (1d_bc) Attribute Descriptions	4-143
Table 4.32	2D Boundary Condition and Link Types	4-146

Table 4.33	2D Boundary Conditions (2d_bc) Attribute Descriptions (Read GIS BC)	4-152
Table 4.34	2D Source over Area (2d_sa) Attribute Descriptions (Read GIS SA)	4-156
Table 4.35	2D Direct Rainfall ¹ over Area (2d_rf) Attribute Descriptions (Read GIS RF)	4-157
Table 4.36	1D WLL (1d_wll) Attributes	4-160
Table 4.37	1D WLL Point (1d_wllp) Attributes	4-161
Table 5.1	2010-10 TUFLOW Versions (iSP, iDP, w32, w64)	5-18
Table 5.2	TUFLOW.exe Options (Switches)	5-20
Table 7.1	Types of Check Files	7-13
Table 7.2	SMS (Map) Output Files and XMDf Types	7-20
Table 7.3	Channel and Node Regime Flags (.eof File)	7-28
Table 7.4	_MB.csv File Columns	7-36
Table 7.5	_MB1D.csv File Columns	7-37
Table 7.6	_MB2D.csv File Columns	7-39
Table 7.7	_TSMB.mif Attributes	7-41
Table 7.8	_TSMB1d2d.mif Attributes	7-41
Table 7.9	Simulation Summary Healthy Model Indicators	7-42
Table 8.1	Quality Control Check List	8-11
Table 9.1	Possible Reasons for Different Results between TUFLOW Builds	9-6
Table 10.1	New Features and Changes for the 2010-10 and 2009-07 Releases	10-2
Table 11.1	TUFLOW_to_GIS.exe Options (Switches)	11-6
Table 11.2	dat_to_dat.exe Options (Switches)	11-10
Table 11.3	12da_to_from_mif.exe Options (Switches)	11-15
Table 11.4	asc_to_asc.exe Options (Switches)	11-16
Table 11.5	convert_to_ts1.exe Options (Switches)	11-18
Table 11.6	xsGenerator .mif Layer Attributes	11-19
Table 11.7	xsGenerator.exe Options (Switches)	11-20

Glossary & Notation

attribute	Data attached to a GIS object. For example, an elevation is attached to a point using a column of data named “Height”. The “Height” of the point is an attribute of the point.
Build	The TUFLOW Build number is in the format of year-month-xx where xx is two letters starting at AA then AB, AC, etc for each new build for that month. The Build number is written to the first line in the .elf and .tlf log files so that it is clear what version of the software was used to simulate the model. The first Build was 2001-03-AA. Prior to that, no unique version numbering was used.
cell	Square shaped computational element in a 2D domain.
centroid	The centre of a region or polygon.
channel	Flow/velocity computational point in a 1D model.
CnM	CnM is a Chezy C, Manning’s n or Manning’s M bed resistance value.
code	Code refers to the code assigned to cells to indicate a cell’s status. It must have a value of one of the following. • -1 for a null cell • 0 for a permanently dry cell • 1 for a possibly wet cell • 2 for an external boundary cell
command	Instruction in a control file.
control file	Text file containing a series of commands (instructions) that control how a simulation proceeds or a 1D or 2D domain is built.
.dbf	Industry standard database file format used by ESRI GIS .shp layers to store attribute data.
DTM	Digital Terrain or Elevation Model
element	An element in a finite element mesh as written by TUFLOW for viewing the 2D grid and results in SMS.
fixed field	Lines of text in a text file that are formatted to strict rules regarding which columns values are entered in to. In previous versions of ESTRY and TUFLOW all text input was in fixed

field format. These formats are still supported, and are still used in a few instances as documented in this manual. Refer to previous manuals for the full documentation on fixed field input.

The following notation is used to define the type and width of the columns. “A” indicates a text (character string), “I” an integer, “F” a decimal or real number and “X” an unused single character space. For example, (A2, 3A1, 2I5, 5X, F10.0), indicates that input along the line starting at column 1 is a two character text (A2) followed by three single characters, then two integers over five columns each, five unused spaces and finally a decimal number over the next ten columns.

The location of the text, integer or decimal number can occur anywhere within the columns designated.

Fric The field used to store bed friction information. This may be the material type or ripple height.

GIS Geographic Information System that can import/export files in .mif or .shp formats.

**GIS
(as alternative to MI)** Same as MI (see MI below)

Grid The mesh of square cells that make up a TUFLOW model.

h-point Computational point located in the centre of a 2D cell.

Invert The elevation of the base (bottom) of a culvert or other structure.

IWL Initial Water Level

land cell A land cell is one that will never wet, ie. an inactive cell.

Layer A GIS data layer (referred to as a “table” in MapInfo).

line A GIS object defining a straight line defined by two points. See also, polyline (Pline).

MAT Material type.

Material Term used to describe a bed resistance category or land-use. Examples of different materials are: river, river bank, mangroves, roads, grazing land, sugar cane, parks, etc.

MI “MI” indicates input or output of a GIS layer. Prior to the 2010-10 release the only format TUFLOW recognised was the .mif format, hence the notation “MI”. The 2010-10 release also recognises the .shp format and the notation “GIS” may be used as an alternative to “MI”.

MID	“MID” indicates input or output of the attribute data of a GIS layer where the first two attributes are the row and column of the 2D cells. The row and column (often labelled “n” and “m”) are used to locate the cell or cell side so that the other attribute data can be assigned to the cell. Prior to the 2010-10 release the only format TUFLOW recognised was the .mid format, hence the notation “MID”. The 2010-10 release also recognises the .dbf format and the notation “RowCol” may be used as an alternative to “MID”. Other formats such as comma delimited (.csv format) may also be used.
.mid	MapInfo Industry standard GIS import/export file that contains the attribute data of geographic objects in a .mif file.
.mif	MapInfo Industry standard GIS import/export file that contains the attribute data types and the geographic coordinates of objects. The attribute data of the objects is stored in the .mid file by the same filename.
node	Water level computation point in a 1D domain. Node in a finite element mesh used for viewing 2D results in SMS. The nodes are located at the cell corners. Node is also used by MapInfo to refer to vertices along a polyline or a region (polygon).
null cell	A null cell is an inactive 2D cell used for defining the inactive side of an external boundary.
obvert	The elevation of the underside (soffit) of a culvert or other structure.
pit	A node with attributes that are used to define a pit channel.
pit channel	A small channel inserted at a pit typically used to convey water from overland 2D domains to 1D pipe networks.
pit inlet	The entrance to a pit channel (eg. a gully trap)
point	GIS object representing a point on the earth’s surface. A point has no length or area.
polygon	See region.
polyline (or Pline)	A GIS object representing one or more lines connected together. A polyline has a length but no area.
polyline segment	One of the lines that make up a polyline.
region	A GIS object representing an enclosed area, ie. a polygon. A region has a centroid, perimeter and area.

RowCol	Identical to MID (see above).
.shp	ESRI GIS layer file containing the geographic coordinates of objects.
SMS	Surface Water Modelling Software distributed by Aquaveo (formerly EMS-I) (www.aquaveo.com) for viewing TUFLOW results.
snap	When geographic objects are connected exactly at a point or along a side. For example, use the “snap” feature in MapInfo. The snap tolerance can be changed using Snap Tolerance .
soffit	The elevation of the underside of a bridge deck or the inner top of a culvert. Same as obvert. Note this manual uses the term obvert.
u-point	Computational point, midway along the right hand side of a 2D cell, where the velocity in the X-direction is calculated. The cell’s left hand side also has a u-point belonging to the neighbouring cell to the left.
v-point	Computational point, midway along the top side of a 2D cell, where the velocity in the Y-direction is calculated. The cell’s bottom side also has a v-point belonging to the neighbouring cell to the bottom.
vertice or vertex	Digitised point on a line, polyline or region (polygon).
WrF	Weir calibration factor for upstream controlled weir flow.
ZC	A “C” Zpt located at the cell centre.
ZH	A “H” Zpt located at the cell corners.
Zpt or Zpts	Points where ground/bathymetry elevations are defined. These are located at the cell centres, mid-sides and corners.
ZU	A “U” Zpt located at the right and left cell mid-sides.
ZV	A “V” Zpt located at the top and bottom cell mid-sides.

1 Introduction

Section Contents

1	INTRODUCTION	1-1
	1.1 TUFLOW	1-2
	1.2 ESTRY	1-4

1.1 TUFLOW

TUFLOW is a computer program for simulating depth-averaged, two and one-dimensional free-surface flows such as occurs from floods and tides. TUFLOW was originally developed for modelling two-dimensional (2D) flows, and stands for Two-dimensional Unsteady FLOW. However, it incorporates the full functionality of the ESTRY 1D network or quasi-2D modelling system based on the full one-dimensional (1D) free-surface St Venant flow equations (see below). TUFLOW has also been dynamically linked (fully integrated) with the ISIS and XP-SWMM 1D solution engines.

The 2D solution algorithm is based on Stelling 1984, and is documented in Syme 1991. It solves the full two-dimensional, depth averaged, momentum and continuity equations for free-surface flow. The scheme includes the viscosity or sub-grid-scale turbulence term that other mainstream software omit. The initial development was carried out as a joint research and development project between WBM Oceanics Australia and The University of Queensland in 1990. The project successfully developed a 2D/1D dynamically linked modelling system (Syme 1991). Latter improvements from 1998 to today focus on hydraulic structures, flood modelling, advanced 2D/1D linking and using GIS for data management (Syme 2001a, Syme 2001b). TUFLOW has also been the subject of extensive testing and validation by WBM Pty Ltd and others (Barton 2001, Huxley, 2004).

TUFLOW is specifically orientated towards establishing flow and inundation patterns in coastal waters, estuaries, rivers, floodplains and urban areas where the flow behaviour is essentially 2D in nature and cannot or would be awkward to represent using a 1D model.

A powerful feature of TUFLOW is its ability to dynamically link to 1D networks using the hydrodynamic solutions of ESTRY, ISIS and XP-SWMM. The user sets up a model as a combination of 1D network domains linked to 2D domains, ie. the 2D and 1D domains are linked to form one overall model.

TUFLOW solves the depth averaged 2D shallow water equations (SWE). The SWE are the equations of fluid motion used for modelling long waves such as floods, ocean tides and storm surges. They are derived using the hypotheses of vertically uniform horizontal velocity and negligible vertical acceleration (ie. a hydrostatic pressure distribution). These assumptions are valid where the wave length is much greater than the depth of water. In the case of the ocean tide the SWE are applicable everywhere.

The 2D SWE in the horizontal plane are described by the following partial differential equations of mass continuity and momentum conservation in the X and Y directions for an in-plan cartesian coordinate frame of reference. The equations are:

$$\frac{\partial \zeta}{\partial t} + \frac{\partial(Hu)}{\partial x} + \frac{\partial(Hv)}{\partial y} = 0 \quad (2D \text{ Continuity})$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} - c_f v + g \frac{\partial \zeta}{\partial x} + g u \left(\frac{n^2}{H^{4/3}} + \frac{f_l}{2g\Delta x} \right) \sqrt{u^2 + v^2} - \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) + \frac{1}{\rho} \frac{\partial p}{\partial x} = F_x$$

(X Momentum)

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + c_f u + g \frac{\partial \zeta}{\partial y} + g v \left(\frac{n^2}{H^{4/3}} + \frac{f_l}{2g\Delta y} \right) \sqrt{u^2 + v^2} - \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \frac{1}{\rho} \frac{\partial p}{\partial y} = F_y$$

(Y Momentum)

where

ζ = Water surface elevation

u and v = Depth averaged velocity components in X and Y directions

H = Depth of water

t = Time

x and y = Distance in X and Y directions

Δx and Δy = Cell Dimensions in X and Y directions

c_f = Coriolis force coefficient

n = Manning's n

f_l = Form (Energy) Loss coefficient

μ = Horizontal diffusion of momentum coefficient

p = Atmospheric pressure

ρ = Density of water

F_x and F_y = Sum of components of external forces (eg. wind) in X and Y directions

The terms of the SWE can be attributed to different physical phenomena. These are:

- Propagation of the wave due to gravitational forces.
- Transport of momentum by advection.
- Horizontal diffusion of momentum or sub-grid scale turbulence (see Section 3.7).
- External forces such as bed friction, rotation of the earth, wind, wave radiation stresses, and barometric pressure.

The computational procedure used is an alternating direction implicit (ADI) finite difference method based on the work of Stelling, 1984. The method involves two stages per timestep each having two steps, giving four steps overall. Each step involves solving a tri-diagonal matrix.

Stage 1, Step 1 solves the momentum equation in the Y -direction for the Y -velocities. The equation is solved using a predictor/corrector method, which involves two sweeps. For the first sweep, the calculation proceeds column by column in the Y -direction. If the signs of all velocities in the X -

direction are the same the second sweep is not necessary, otherwise the calculation is repeated sweeping in the opposite direction.

The second step of Stage 1 solves for the water levels and X-direction velocities by solving the equations of mass continuity and of momentum in the X-direction. A tri-diagonal equation is obtained by substituting the momentum equation into the mass equation and eliminating the X-velocity. The water levels are calculated and back substituted into the momentum equation to calculate the X-velocities. This process is repeated for a recommended two iterations. Testing on a number of models showed there to be little benefit in using more than two iterations unless there are rapid changes in the hydraulic conditions per timestep as may occur with modelling inundation from a dambreak.

Stage 2 proceeds in a similar manner to Stage 1 with the first step using the X-direction momentum equation and the second step using the mass equation and the Y-direction momentum equation.

The solution as formulated by Stelling has been enhanced and improved to provide much more robust wetting and drying of elements, upstream controlled flow regimes (eg. supercritical flow and upstream controlled weir flow), modifications to cells to model structure obverts (eg. bridge decks) and additional energy losses due to fine-scale features such as bridge piers.

1.2 ESTRY

ESTRY is a powerful 1D network dynamic flow software suitable for mathematically modelling floods and tides (and/or surges) in a virtually unlimited number of combinations.

The program was developed by WBM Pty Ltd over a period of nearly fifty years and has been successfully applied to a wide range of investigations from simple single channel applications to complex quasi-2D flood models to large pipe network systems. The network schematisation technique used allows realistic simulation of a wide variety of 1D and quasi-2D situations including: complex river geometries; associated floodplains and estuaries; and urban channel and pipe network systems. By including non-linear geometry it is possible to provide an accurate representation of the way in which channel conveyance and available storage volumes vary with changing water depth, and of floodplains and tidal flats that become operable only above certain water levels.

There is a considerable amount of flexibility in the way the network elements can be interconnected, allowing the representation of a river by many parallel channels with different resistance characteristics and the simulation of braided streams and rivers with complex branching. This flexibility also allows a variable resolution within the network so that areas of particular interest can be modelled in fine detail with a coarser network representation being used elsewhere.

The scheme is based on a numerical solution of the 1D unsteady St Venant fluid flow equations (momentum and continuity) including the inertia terms. This capability of modelling tidal flows has the added advantage of enabling the tidal portion of a flood model to be calibrated separately using readily obtainable measurements of the tide levels and flows. Extension of the calibrated tidal model into the floodplain then results in a more accurate flood model that the flood channels can be calibrated separately against available records.

The 1D solution uses an explicit finite difference, second-order, Runge-Kutta solution technique (Morrison and Smith, 1978) for the 1D SWE of continuity and momentum as given by the equations below. The equations contain the essential terms for modelling periodic long waves in estuaries and rivers, that is: wave propagation; advection of momentum (inertia terms) and bed friction (Manning's equation).

$$\frac{\partial(uA)}{\partial x} + B \frac{\partial \zeta}{\partial t} = 0 \quad (1D \text{ Continuity})$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + g \frac{\partial \zeta}{\partial x} + k |u| u = 0 \quad (1D \text{ Momentum})$$

where

u = depth and width averaged velocity

ζ = water level

t = time

x = distance

A = cross sectional area

B = width of flow

$$k = \text{energy loss coefficient} = \frac{gn^2}{R^{4/3}} + \frac{f_l}{2g\Delta x}$$

n = Manning's n

f_l = Form (Energy) Loss coefficient

R = Hydraulic Radius

g = acceleration due to gravity

In addition to the normal open channel flow situations, a number of special types of channel are available including:

- uniform open channel, with or without specified bed gradient;
- subcritical and supercritical flow regimes;
- non-inertial channels;
- circular, rectangular (box) and irregular culverts;
- pit or manhole inlets;
- bridges;
- weir channels for flow across roadways, levees etc;
- user defined structures; and

- uni-directional channels of any type capable of being specified, to allow flow in only one direction.

The type of information provided as output by the model for a flood or tide simulation includes the water levels, flows, and velocities throughout the area being modelled for the simulation period. Other information available includes maximum and minimum values of these variables as well as total integral flows (integrated with time) through each network channel.

2 Overview

Section Contents

2	OVERVIEW	2-1
2.1	Software Structure	2-2
2.2	Data Input	2-3
2.2.1	Structure	2-3
2.2.2	Suggested Folder Structure	2-5
2.2.3	File Types and Naming Conventions	2-6
2.2.4	GIS Input File Types and Naming Conventions	2-10
2.3	Performing Simulations	2-14
2.4	Data Output	2-14
2.5	Limitations and Recommendations	2-15

2.1 Software Structure

TUFLOW and ESTRY are the computational engines for carrying out 2D/1D hydrodynamic calculations. They do not have their own graphical user interface, but utilise GIS and other software for the creation, manipulation and viewing of data. These software are:

- A GIS that can import/export .mif files (MapInfo Interchange Format) or .shp files (ESRI standard).
- 3D surface modelling software (eg. Vertical Mapper) for the creation and interrogation of a DTM, and for importing 3D surfaces of water levels, depths, hazard, etc.
- SMS (Surfacewater Modelling System – www.ems-i.com) or WaterRIDE (www.waterride.net) for the viewing of results and creation of flow animations.
- A good text editor such as UltraEdit. UltraEdit has been colour customised especially for TUFLOW formatted files.
- Spreadsheet software such as Microsoft Excel.

The above combination of software offers a very powerful and economical system for 2D/1D hydraulic modelling.

Should a complete Graphical User Interface (GUI) that allows the user to create, manage and view models and model output within the one interface, two options are available:

- SMS TUFLOW Interface (available as of SMS Version 9.2 or later – see <http://www.aquaveo.com/sms>)
- XP-SWMM2D is a module of the new XP-SWMM interface that offers a complete 1D/2D modelling environment. The 1D XP-SWMM solution has been dynamically linked to the TUFLOW 2D. See www.xpsoftware.com.au/products/xp2d.htm.

ISIS 1D models can also be dynamically linked to TUFLOW 2D domains using ISIS-TUFLOW, a module of ISIS (contact software@halcrow.com for more information).

2.2 Data Input

2.2.1 Structure

Figure 2-1 illustrates the data input and output structure. Text files are used for controlling simulations and simulation parameters, whilst the bulk of data input is in GIS formats. The GIS approach offers several benefits including:

- the unparalleled power of GIS as a “work environment”;
- the many GIS data management, manipulation and presentation tools;
- input data is geographically referenced, not 2D grid referenced, allowing the 2D cell size to be readily changed;
- substantial cost savings in not having to develop a specialised graphical interface;
- efficiency in producing high quality GIS based mapping for reports, brochures, plans and displays;
- handover to clients requiring data in GIS format; and
- better quality control.

A GIS system is used to set up, modify, thematically map and manage the data. At the time of writing the most used GIS for TUFLOW modelling is MapInfo, however, with the support of .shp files as part of the 2010-10 release, ArcGIS will be significantly easier to use as a TUFLOW modelling environment. Applications of TUFLOW using other CAD/GIS platforms have also been successfully adopted by users.

For time-series data and other non-geographically located data, spreadsheet software and/or the SMS TUFLOW Interface are used.

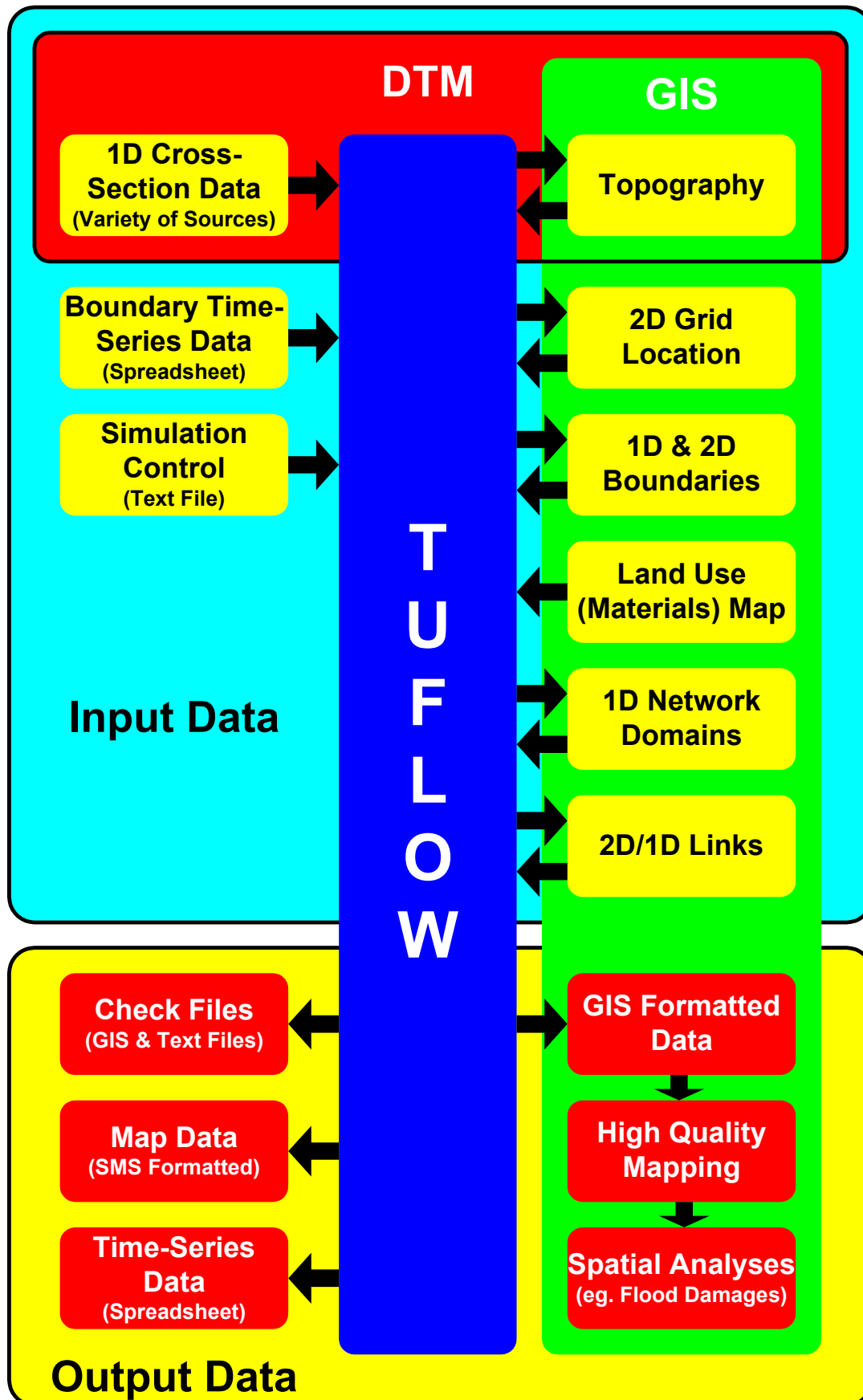


Figure 2-1 TUFLOW Data Input and Output Structure

2.2.2 Suggested Folder Structure

Table 2.1 presents the recommended set of sub-folders to be set up for a 2D/1D model or a 1D only model. Any folder structure may be used, however, it is strongly recommended that a system similar to that below be adopted. For large modelling jobs with many scenarios and simulations, a more complex folder structure may be warranted, but should be based on that below.

Note:

- **Files are located relative to the file they are referred from.** For example, the path and filename of a file referred to in a .tgc file is sourced **relative to the .tgc file (not the .tcf file)**.
- Whilst TUFLOW readily accepts spaces in filenames and paths, other software may have issues with spaces. It is therefore recommended that spaces are not used in the simulation path and filename without prior testing.
- Filenames and extensions are not case sensitive.

Table 2.1 Recommended Sub-Folder Structure

Sub-Folder	Description
Locate folders below on the system network under a folder named “tufLOW” or “estry” in the project folder (eg. J:\Project12345\tufLOW) These folders should be backed up regularly	
bc dbase	Boundary condition database(s) and time-series data for 1D and 2D domains.
model	.tgc, .tbc and other model data files, except for the GIS layers which are located in the model\mi folder (see below).
model\mi model\shp	GIS layers that are inputs to the 2D and 1D model domains. Also GIS workspaces.
runs	.tcf and .ecf simulation control files.
runs\log	.tlf or .elf log files and _messages.mif files (use Log Folder)
For large models the folders below can be located on a local hard drive under a folder “tufLOW” or “estry” under the project folder (eg. C:\Project12345\tufLOW) These folders do not need to be backed up regularly	
results	The result files (use Output Folder).
check	GIS and other check files to carry out quality control checks (use Write Check Files).

2.2.3 File Types and Naming Conventions

Files are generally classified as:

- Control Files
- Data Input Files
- Data Output Files
- Check Files

Control files are used for directing inputs to the simulation and setting parameters. The style of input is very simple, free form commands, similar to writing down a series of instructions. This offers the most flexible and efficient system for experienced modellers. It is also easy for inexperienced users to learn.

Data input files are primarily GIS layers and comma-delimited files generated using spreadsheet software. Models may still use the original fixed field data input formats if desired.

Data output files are primarily map output in SMS formats, GIS layers, text files and comma-delimited files (see Section 7).

In addition to the above, an extensive range of check files are produced in GIS, text and comma-delimited formats to carry out quality control checks (see Section 7.2).

The most common file types and their extensions are listed in Table 2.2.

Table 2.2 List of Most Commonly Used File Types

File	Extension	Description	Format
Control Files			
TUFLOW Simulation Control File	.tcf	Controls the data input and output for a 2D or a 2D/1D simulation. The filename (without extension) is used for naming all 2D domain files. Mandatory.	Text
TUFLOW Boundary Conditions Control File	.tbc	Controls the 2D boundary condition data input. Is mandatory for a 2D or 2D/1D simulation.	Text
TUFLOW Event File	.tef	Database of .tcf and .ecf file commands for different events.	Text
TUFLOW Geometry Control File	.tgc	Controls the 2D geometric or topographic data input. Is mandatory for a 2D or 2D/1D simulation.	Text
ESTRY Simulation Control File	.ecf	Controls the data input and output for 1D domains. The filename (without extension) is used for naming all 1D output files. Mandatory.	Text
Read Files	.trd .erd .rdf	A file that is included inside another file using the Read File command in .tcf, .tgc and .ecf files. Minimises repetitive specification of commands common to a group of files.	
Data Input			
ArcGIS SHP Layers	.shp .dbf	ArcGIS's industry standard for GIS layers. The .shp file contains information on the GIS objects coordinates, while the .dbf file contains the attribute data information associated with the objects.	Binary
Comma Delimited Files	.csv	These files are used for boundary condition databases, boundary condition tables, 1D cross-sections, 1D storage tables, etc. They are opened and saved using spreadsheet software such as Microsoft Excel.	Text
MapInfo MIF/MID Files	.mif .mid	MapInfo's industry standard GIS data exchange format. The .mif file contains the attribute data definitions and the geographic data of the objects. The .mid file contains the attribute data. Used for the majority of data inputs. The .mid files are of similar format to .csv files, so they can be opened by Excel or other spreadsheet software.	Text
TUFLOW Materials File	.tmf	Sets the Manning's n values for different bed material categories in 1D and 2D domains.	Text

File	Extension	Description	Format
Fixed Field Files	variety of extensions	Most new models do not require any fixed field input. However, for those hard-core modellers who like the fixed field input style, these formats are still supported. Note, the fixed field format documentation has been removed from this manual – see manuals prior to 2007 downloadable from www.tuflow.com .	Text
Data Output (see Section 7)			
SMS Super File	.sup	SMS super file containing the various files and other commands that make up the output from a single simulation. Opening this file in SMS opens the .2dm file and the primary .dat files.	Text
SMS Mesh File	.2dm	SMS 2D mesh file containing the 2D/1D model mesh and elevations. It also contains information on materials and 2D grid codes.	Text
SMS Data File	.dat	SMS generic formatted simulation results file. TUFLOW output is written using the .dat format. See Table 7.2 and Map Output Data Types for the different .dat file outputs.	Binary
SMS XMDF File	.xmdf	An alternative to using the .dat files described above. .xmdf files are much faster to access and can contain all TUFLOW map output within the one file (rather than a file per output type). See Table 7.2 and Map Output Data Types for the different .xmdf file data sets available.	Binary
Comma Delimited Files	.csv	These files are used for 2D and 1D time-series data output. They are opened and saved using spreadsheet software such as Microsoft Excel.	Text
MIF/MID Files	.mif .mid	Used for GIS based output including graphing of 1D and 2D time-series output within a GIS.	Text
TUFLOW Restart File	.trf	2D domain computational results at an instant in time for restarting simulations.	Binary
ESTRY Restart File	.erf	1D domain computational results at an instant in time for restarting simulations.	Text

File	Extension	Description	Format
Check Files (see Section 7.2)			
TUFLOW Log File	.tlf	A log file containing information about the 2D/1D data input process and a log of the 2D simulation.	Text
ESTRY Log File	.elf	A log file containing information about the 1D data input process and a log of 1D only simulation.	Text
ESTRY Output File	.eof	Original ESTRY output file containing all 1D input data and results. Very useful for checking 1D input data and reviewing flow regimes in 1D channels.	Text
Comma Delimited Files	.csv	These files are used for outputting processed 1D and 2D domain time-series boundaries and other data for checking. They are opened and saved using spreadsheet software such as Microsoft Excel.	Text
MIF/MID Files	.mif .mid	A range of 1D and 2D domain check files are produced for checking processed input data within a GIS.	Text

2.2.4 GIS Input File Types and Naming Conventions

As the bulk of the data input is via GIS data layers, efficient management of these data is essential. For detailed modelling investigations, the number of TUFLOW GIS data layers has been known to reach over a hundred for large complex models, although the majority of models would utilise five to twenty layers. Good data management also caters for the many other GIS layers (aerial photos, cadastre, etc) being used.

It is strongly recommended that the prefixes described in Table 2.3 be adhered to for all 1D and 2D GIS layers. This greatly enhances the data management efficiency and, importantly, makes it much easier for another modeller or reviewer to quickly interpret the model.

Data input is structured so that there is no limit on the number of data sources. Commands are repeated indefinitely in the text files to build a model from a variety of sources. For example, a model's topography may be built from more than one source. A DTM may be used to define the general topography, while several 3D elevation lines (breaklines) define the crests of levees. The "build-a-model" approach offers unlimited flexibility and efficiency.

Table 2.3 GIS Input Data Layers and Recommended Prefixes

GIS Data Type	Suggested File Prefix	Description	Refer to Section
2D Domain GIS Layers			
2D Boundaries and 2D/1D Links	2d_bc_ 2d_hx_ 2d_sx_	Mandatory layer(s) defining the locations of 2D boundaries and 2D/1D dynamic links. For large models it may be wise to separate the boundary conditions from the 1D/2D links, in which case use the 2d_hx_ and 2d_sx_ prefixes. Cell code values may also be defined in this layer.	4.10
2D Cell Codes	2d_code_	Optional GIS layers containing objects, typically polygons, that define the cell codes. Note: The preferred approach is to define cell codes using the 2d_bc layer (see Read GIS Code with the BC option).	4.3
2D Flow Constrictions	2d_fc_	Optional layers defining the adjustment of 2D cells to model bridges, box culverts, etc.	4.7
Gauge Level Output Location	2d_glo_	Optional layer defining the location of the gauge for output based on water level rather than time intervals. See Read GIS GLO .	
2D Grid	2d_grd_	Optional layers used to define the 2D grid or mesh. Now primarily used as a quality control check file (in earlier versions was a mandatory input). Contains information on the 2D cell: reference, code, material and other information.	4.3
2D Initial Water Levels	2d_iwl_	Optional layer(s) defining the spatial variation in 2D domain initial water levels at the start of the model simulation.	4.9
2D Grid Location	2d_loc_	GIS layer defining the origin and orientation of the 2D grid. This layer is optional, however, is the preferred method for geographically locating 2D domains.	4.3
2D Longitudinal Profile Output Locations	2d_lp_	Optional layer(s) defining the locations longitudinal profile output from the 2D model domain	4.8
2D Land-Use (Materials) Categories	2d_mat_	Layers to define or change the land-use (material) types on a cell-by-cell basis.	4.3
2D Plot (Time-Series) Output Locations	2d_po_	Optional layer(s) defining the locations and types of time-series output from the 2D domains.	4.8

GIS Data Type	Suggested File Prefix	Description	Refer to Section
Rainfall over Area	2d_rf_	Optional layer(s) defining the polygons of sub-catchment areas for applying rainfall directly onto 2D domains.	4.10
2D Source over Area	2d_sa_	Optional layer(s) defining the polygons of sub-catchment areas for applying a source (flow) directly onto 2D domains.	4.10
2D Elevations over an area	2d_z_	Optional layer(s) that define areas (polygons) of elevations at a constant height.	4.3
Elevation Lines (Breaklines) (Ridges and Gullies)	2d_zln_ 2d_zlr_ 2d_zlg_	Optional 2D or 3D breaklines defining the crest of ridges (eg. levees, embankments) or thalweg of gullies (eg. drains, creeks). Ridges and gullies cannot occur in the same layer so 2d_zlr_ is often used for ridges and 2d_zlg_ for gullies.	4.3
2D Elevations as points	2d_zpt_	Layer(s) that define the elevations at the 2D cells mid-sides, corners and centres.	4.3
2D and 3D Shapes for (Points, Lines, Polygons)	2d_zsh_ 2d_ztin_	Points, lines and polygons defining 2D and 3D shapes for changing the elevations. Lines can be specified with a width (thickness), and polygons are used as the boundaries for creating TINs (triangulations).	4.4.8 4.4.9
Variable (over time) 2D and 3D Shapes	2d_vzsh_	Points, lines and polygons defining the final geometry of a breach or other variation in model topography over time. Also includes points defining trigger locations.	4.4.10
Flow Constriction 2D and 3D Shapes	2d_fcsh_	Points, lines and polygons that modify the 2D cell sides flow width, place a lid (obvert or soffit) on a 2D cell, additional energy losses, and other modifications.	4.7.2.1
Layered Flow Constriction 2D and 3D Shapes	2d_lfcsh_	Points, lines and polygons that allow the user to modify the 2D cell sides flow width, percentage blockage, and additional energy losses, for up to three layers in the vertical. Designed for modelling 2D flow under and over bridges, pipes and other obstructions across the waterway.	4.7.2.3

GIS Data Type	Suggested File Prefix	Description	Refer to Section
1D Domain GIS Layers			
1D Boundaries	1d_bc_	Layer(s) defining the locations of 1D domain boundaries. Note: Any links to the 2D domain are automatically determined via the 2d_bc layer(s).	4.10
1D Initial Water Levels	1d_iwl_	Optional layer(s) defining the spatial variation in initial water levels at 1D nodes at the start of the model simulation.	4.9
1D Manholes	1d_mh_	Optional layer(s) defining the location and parameters of manholes.	4.5.2
1D Domain Network	1d_nwk_	Layer(s) that define the 1D or quasi-2D domain network of flowpaths (channels), storage areas (nodes) and pit inlets (pits).	4.5
1D Tabular Input	1d_tab_ 1d_xs_ 1d_na_ 1d_bg_	Optional layer(s) that provide links to tabular data (eg. a cross-section's X-Z values). Tabular data includes cross-sections (XZ and processed forms); storage surface area versus height at nodes (NA tables); and loss versus height coefficients at a structure (BG tables). Although, different table links can occur within the same layer, some modellers prefer to separate them and use prefixes such as those suggested to the left. For medium to large models, create a folders for each of xs, na and bg under the model folder, and place the 1d_xs_, 1d_na_ and 1d_bg_ layers in these folders respectively, along with all of the linked .csv files.	4.6.3
1D Water Level Lines for SMS Output	1d_wll_	Lines of horizontal water level (as judged by the modeller). These lines are used to generate 3D surfaces or water level, velocity and other output of 1D domains. This allows the combined viewing and animation of 2D and 1D domains together.	4.11
1D WLL Points	1d_wllp_	Points that define the elevations (usually from a DTM) and material values across the WLLs. This offers high quality viewing and mapping of the 1D domains.	4.11

2.3 Performing Simulations

TUFLOW or ESTRY simulations are started by:

- using Microsoft Explorer (a file association between .tcf files and TUFLOW.exe or a .ecf file and ESTRY.exe is required – see Section 5.4);
- directly from UltraEdit (see Section 5.5);
- running a batch file (see Section 5.6);
- from a Console Command Window (see Section 5.7); or
- using a GUI such as the SMS TUFLOW Interface.

2.4 Data Output

TUFLOW produces a range of output as presented below (see Section 7). In addition, several post-processing utilities are used for transferring data to GIS and other software (see Section 11).

Output is structured into two categories:

- Check Files for checking and quality control of models.
- Result Files containing the 1D and 2D results.

Result Files (Sections 7.1, 7.3 and 7.4)

Result files contain the hydraulic results of the simulation in the 1D and 2D domains:

- SMS formatted mesh and results files for viewing the 2D and 1D domains and their results. Animations of results are created using SMS.
- .csv (comma delimited) text output of time series data for direct input into spreadsheet software such as Microsoft Excel.
- .mif/.mid files for viewing 2D and 1D domain results in GIS.
- text files that log the simulation.

Check Files (Section 7.2)

Check files are produced so that modellers and reviewers can readily check that the constructed model is as intended. Advanced models draw upon a wide variety of data sources. The check files represent the final data set after all data inputs, allowing the model construction to be viewed in its final form. The check files take the following forms:

- .mif/.mid GIS formats for viewing graphically any errors, warnings and checks, the 1D network, 2D grid, 2D topography, 2D/1D boundaries and connections, and other formats;
- text files for checking parameter and tabular inputs.

2.5 Limitations and Recommendations

TUFLOW is designed to model free-surface flow in coastal waters, estuaries, rivers, creeks, floodplains and urban drainage systems. Flow regimes through structures are handled by adaptation of the 1D St Venant Equations and the 2D Shallow Water Equations using standard structure equations. Supercritical flow areas can be represented (see note below).

Limitations and recommendations to note are:

- 1 In areas of super-critical flow through the 2D and 1D domains, the results should be treated with caution, particularly if they are in key areas of interest. Hydraulic jumps and surcharging against obstructions may occur in reality – these highly 3D localised effects are not modelled by software such as TUFLOW.
- 2 The Smagorinsky viscosity formulation is preferred over the constant viscosity formulation to model sub-cell turbulence (Barton 2001). It is always good practice to carry out sensitivity tests to ascertain the importance of the viscosity coefficient(s) and formulation.
- 3 Caution may be needed when using very small 2D cell sizes, particularly when the flow depth is significantly larger than the cell width (Barton 2001). Modelling on a very fine grid (<1m), especially in deep water, may start to violate the assumptions of 2D solutions based on the Shallow Water Equations. CFD (Computational Fluid Dynamics) codes that model the turbulence and other terms much more accurately may be needed in these situations. The influence of the viscosity (sub-grid scale turbulence) term can be particularly relevant in these situations. However, testing of models with very fine cell sizes (down to 0.1m) has indicated that reliable results can be obtained. There seems to be a trend indicating that slightly higher [Viscosity Coefficient](#) maybe justified of up to 0.5 using the Smagorinsky [Viscosity Formulation](#), which is the preferred (and default) approach.
- 4 Modelling of hydraulic structures should always be cross-checked with desktop calculations or other software, especially if calibration data is unavailable. All 1D and 2D schemes are only an approximation to the complex flows that can occur through a structure, and regardless of the software used should be checked for their performance (Syme 1998, Syme 2001).
- 5 There is no momentum transfer between 1D and 2D connections when using the sink/source connection approach (SX link). The HX link does preserve momentum in the sense that the velocity field is assumed to be undisturbed across the link, but the velocity direction is not influenced by the direction of the linked 1D channel. In most situations these assumptions are not of significant concern, however they may influence results where a large structure (relative to the 2D cell size) is modelled as a 1D element.

3 The Modelling Process

Section Contents

3	THE MODELLING PROCESS	3-1
3.1	Is a 2D or 2D/1D Model Feasible?	3-2
3.2	Linking 1D and 2D Domains	3-3
3.3	Data Requirements	3-6
3.4	Calibration and Sensitivity	3-6
3.5	Model Resolution	3-7
3.5.1	2D Cell Size	3-7
3.5.2	1D Network Definition	3-7
3.6	Computational Timestep	3-8
3.6.1	2D Domains (Courant Number)	3-8
3.6.2	1D Domains (Courant Number)	3-8
3.6.3	2D/1D Models	3-9
3.7	Eddy Viscosity	3-10

3.1 Is a 2D or 2D/1D Model Feasible?

With present day computers, there are few computer hardware constraints in setting up 1D models. However, for 2D models the first step is to decide whether it is feasible and practical to set up a model, given the limitations of your computer hardware. Experienced modellers can usually quickly determine an answer by considering the following:

- 1 Clearly understanding/defining the model's objectives, and if known, the modelling budget.
- 2 Determining the minimum cell size required to model the hydraulics accurately enough to meet the study's objectives. Preferably at least three to four cells across the major flowpaths (depending on the topography). Minor flowpaths may be more coarsely or not represented if they play no significant role hydraulically in regard to meeting the modelling objectives. For example, residual water drains over a floodplain may not affect peak flood levels; in which case, it may not be necessary to model them.
- 3 If it is not possible to model a major flowpath with a sufficient cell resolution (see Figure 3-1), the flowpath can be modelled as a 1D branch cut through the 2D domain (see Section 3.2 and Sketch 1c in Figure 3-2). This may allow a larger cell size to be used, and a greater area modelled in 2D, or a faster simulation time. For example, the river may be modelled in 1D and the floodplain in 2D.
- 4 Establish possible boundary locations for the model. These are influenced by locations that are well defined hydraulically, and any constraints on the extent of the topographic data (DTM). Dynamically linking with a 1D domain offers significant flexibility in locating the 2D domain.
- 5 Determine the number of rows and columns of the grid based on the overall dimensions of the 2D domain and the minimum cell size. Calculate the number of cells (rows by columns), and estimate the average number of cells that would be wet.
- 6 In 2001, using a P3 1GHz computer, overnight simulations of models varying in cell size from 5m to 60m for durations of 12 to 120 hours were achieved with several hundred thousand wet cells. In 2007, these models are running around 10 times faster, and models in excess of a million wet cells have been created! For large models, it may be beneficial to start with a coarser cell size to facilitate quick turnover of simulations before proceeding to a finer cell size. This is a relatively easy process as most input data is not cell size dependent. **Note that halving the 2D cell size typically corresponds to increasing the simulation time by a factor of eight (four times as many cells and half the timestep).**

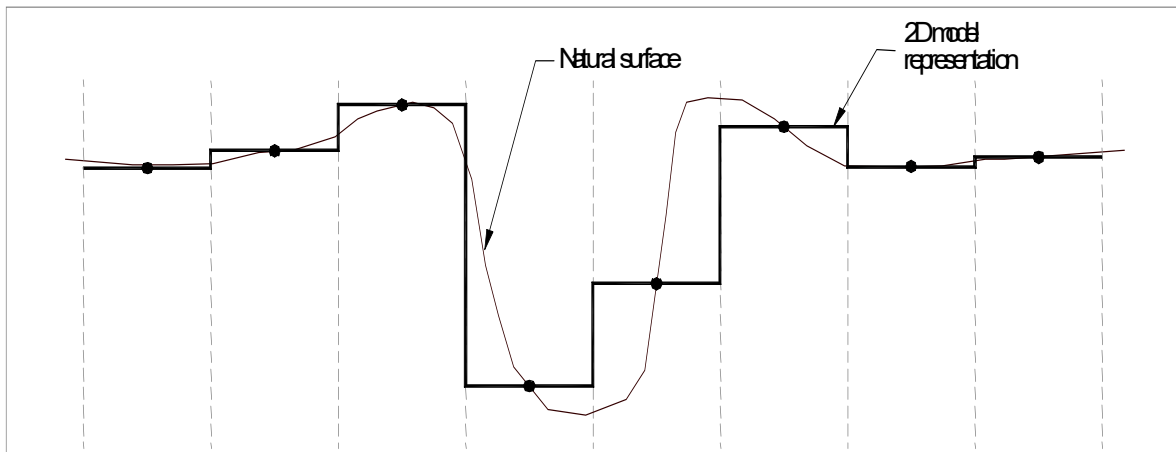


Figure 3-1 Example of a Poor Representation of a Narrow Channel in a 2D Model

3.2 Linking 1D and 2D Domains

TUFLOW 1D and 2D domains can be linked in a variety of ways as illustrated in Figure 3-2 (Benham, et al, 2003). The simplest approach is to replace part of a 1D model by nesting a 2D domain inside the broader scale 1D model as shown in Sketch 1a in Figure 3-2. This approach was developed by Syme (1991) and has been widely applied through various versions of the TUFLOW software since 1990.

Further refinements to TUFLOW were incorporated during the late 1990s to be able to:

- Insert 1D networks “underneath” a 2D domain or through, for example, an embankment (see Figure 3-3 and Sketch 1b in Figure 3-2).
- Replace or “carve” a 1D channel through a 2D domain (see Figure 3-4 and Sketch 1c in Figure 3-2).

The linking or stitching together of 2D domains was incorporated in 2006, while current research is focusing on 2D nesting (ie. reducing the 2D cell size in defined areas).

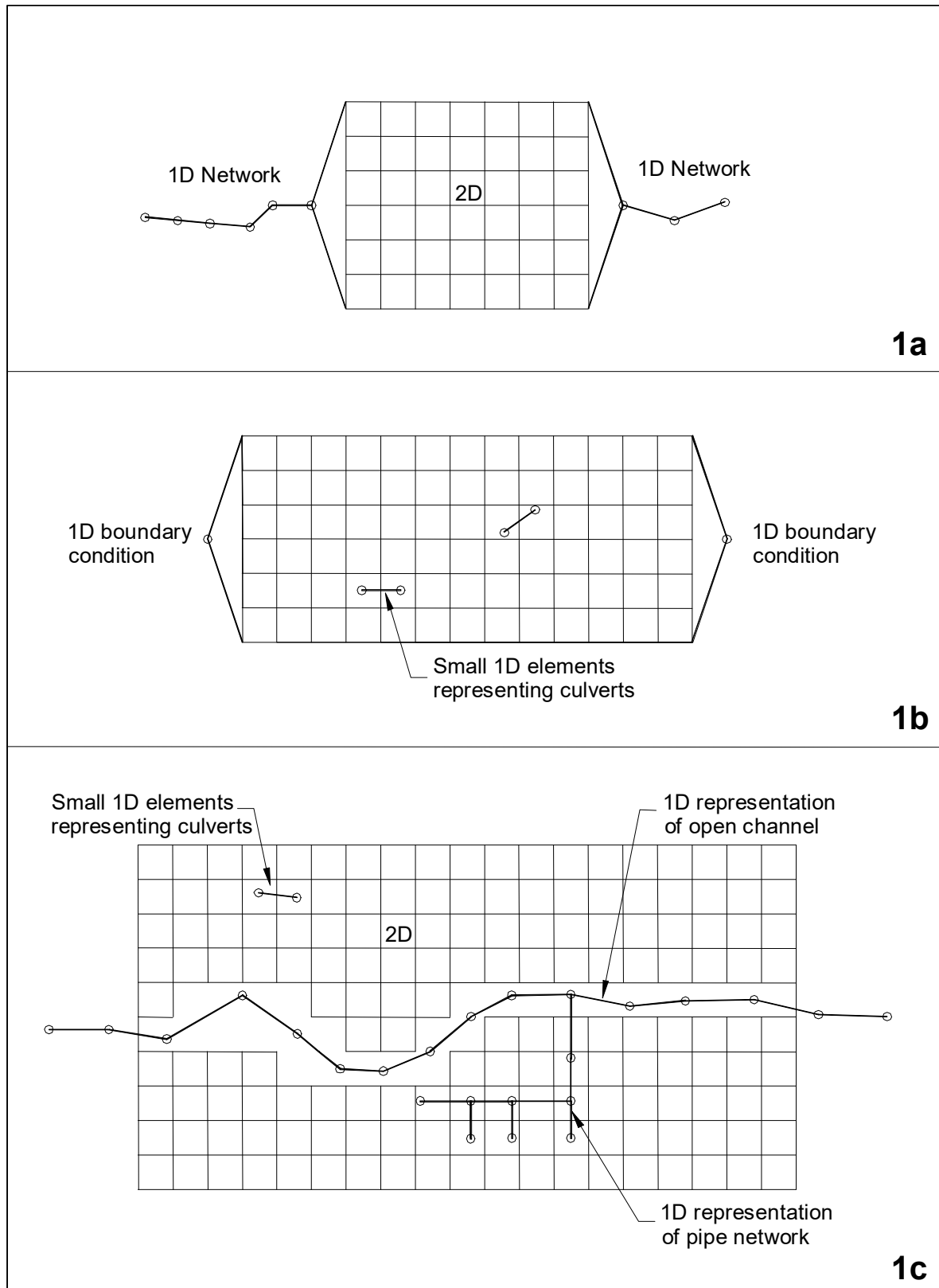


Figure 3-2 1D/2D Linking Mechanisms

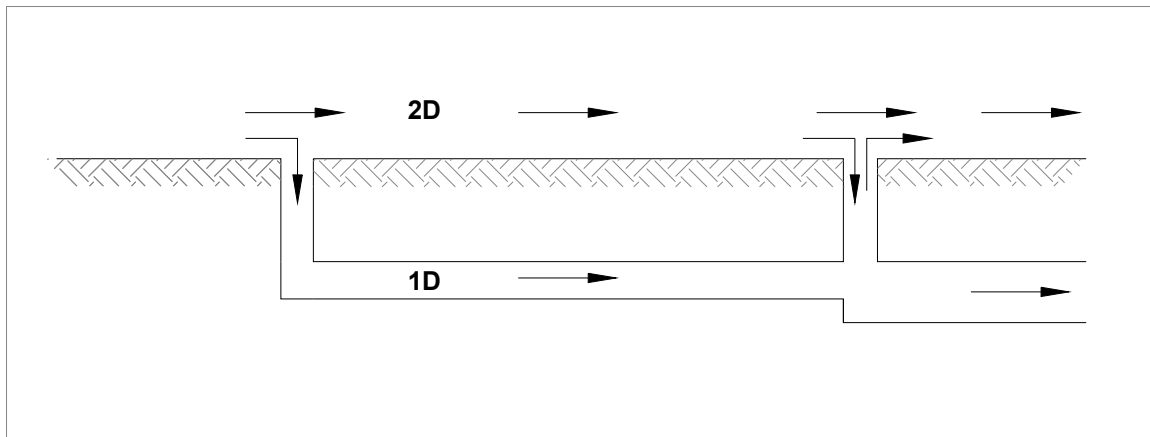


Figure 3-3 Modelling a Pipe System in 1D underneath a 2D Domain

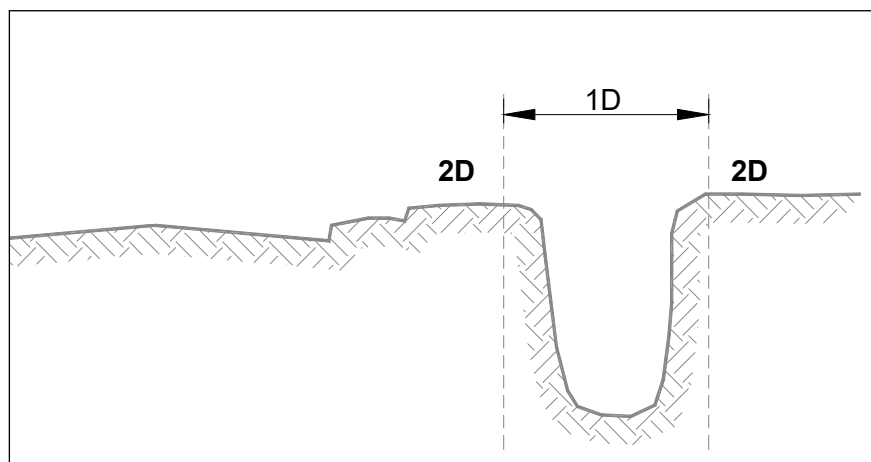


Figure 3-4 Modelling a Channel in 1D and the Floodplain in 2D

3.3 Data Requirements

The minimum data requirements for setting up a 2D/1D hydraulic model are:

- 1 A DTM with sufficient resolution and accuracy to depict the topography of all flowpaths and storage areas in the 2D domain(s). The vertical accuracy depends on the modelling objectives and budget constraints, however, for large scale models $\pm 0.2\text{m}$ is preferred, whilst for fine-scale urban models $< \pm 0.1\text{m}$ is recommended. The vertical accuracy is dependent on the typical depths of inundation in key areas.
- 2 Cross-sections for any 1D flowpaths.
- 3 If bed resistance varies over the model, geo-corrected aerial photography or other GIS layer from which material (land-use) zones are digitised for setting Manning's n values.
- 4 Boundary conditions (eg. ocean water levels, catchment inflows, rainfall, evaporation, etc).
- 5 Calibration data locations as points in a GIS layer. Peak levels should be attached as attributes to the calibration points.
- 6 Surveys of key hydraulic controls such as levees / embankments (3D breaklines), culverts, bridges, etc.

3.4 Calibration and Sensitivity

Models are usually calibrated against known flood or tidal conditions with the bed resistance coefficient (eg. Manning's n) adjusted until calculated water levels and flows are consistent with recorded field measurements. Where there is poor or insufficient topographic data the calibration procedure may also involve adjustments to the model topography to provide an adequate representation of the recorded flow behaviour. This is more common in 1D domains (where there is a choice of cross-sections to define a flowpath). There is usually little opportunity to adjust topography (from that surveyed) in 2D domains.

Ideally, the model would be calibrated for conditions similar to those under investigation although this is not always possible, particularly when major floods are being considered. In these situations, a sensitivity analyses maybe carried out by increasing and decreasing calibration factors such as Manning's n .

3.5 Model Resolution

3.5.1 2D Cell Size

The cell sizes of 2D domains need to be sufficiently small to reproduce the hydraulic behaviour. Refer to Section 3.1 above for further discussion.

3.5.2 1D Network Definition

The adequacy of the 1D domains is primarily dependent on the network representation adopted. In general, the finer the resolution the more accurate the model, but the longer the computing time. For stability reasons, the timestep for computation is normally controlled by the minimum channel length (see Section 3.6.2). The end result may require a compromise between the level of detail and the computational effort.

Prior to Build 2005-05-AN, for 2D/1D models it is highly preferable that the 1D solution does not dictate the timestep as all domains use the same timestep. As of Build 2005-05-AN, different timesteps can be specified for 1D and 2D domains largely removing this constraint.

The first step in setting up a model is to define the flow patterns and to use each identified flow path as the basis for a channel of the network. Following this step the flow paths are linked at junctions, or nodes, and each node is considered as a storage element, which accepts the flow from the adjoining channels. In this way, the model is built up as a series of interconnected channels and nodes with the channels representing the flow resistance characteristics.

For compatibility with the mathematical assumptions, the channels would ideally have more or less uniform cross-sections with constant bottom slope and a minimum of longitudinal curvature. In practice this requirement cannot always be met, particularly where a fine resolution of detail is not required in a portion of the study area. In this case, a flow path is represented by an “equivalent” channel. Experience has indicated that in most cases an adequate calibration can be achieved by deriving a single channel equivalent to a number of series or parallel channels using the steady state Manning's relation for deriving the equivalent channel characteristics.

All nodes and channels are labelled with an ID. No two nodes or two channels can have the same ID. A node and a channel can have the same ID.

3.6 Computational Timestep

The selection of the timestep is critically important for the success of a model. The run time is directly proportional to the number of timesteps required to calculate model behaviour for the required time period, while the computations may become unstable and meaningless if the timestep is greater than a limiting value. This is known as the Courant stability criterion.

3.6.1 2D Domains (Courant Number)

For the 2D scheme, the Courant Number generally needs to be less than 10 and is typically around 5 for most real-world applications (Syme 1991). The computation timestep in the .tcf file (see [Timestep](#)) should be set in accordance with this criterion as given in the equation below.

$$C_r = \frac{\Delta t \sqrt{2gH}}{\Delta x} \quad \text{2-D Square Grid (1)}$$

where

Δt = timestep, s

Δx = length of model element, m

g = acceleration due to gravity, ms^{-2}

H = depth of water, m

As a rule, the timestep is typically half the cell size. For steep models with high Froude numbers and supercritical flow, smaller timesteps may be required. It is strongly advised to not simply reduce the timestep if the model is unstable, but rather to establish why it is unstable and, in most instances, correct or adjust the model topography, initial conditions or boundary conditions to remove the instability.

If the model is operating at high Courant numbers (>10), sensitivity testing with smaller timesteps to demonstrate no measurable change in results should be carried out.

The occurrence of high mass errors is also an indicator of using too high a timestep.

3.6.2 1D Domains (Courant Number)

For the 1D channels the Courant criterion is expressed in the form:

$$C_r = \frac{\Delta t \sqrt{gH}}{\Delta x} \quad \text{1-D Scheme (2)}$$

The time step selected should not be greater than the minimum value for any channel (except non-inertial channels such as bridges, culverts, etc). Accuracy of the results is also influenced by time step. The limiting value adopted is usually a compromise between accuracy, stability and simulation time, and sensitivity checks are recommended. The occurrence of mass errors may indicate the use of too high a timestep.

Typical timestep values are 60 or 120 seconds for a model with a minimum channel length of 500 metres, down to 1 second for 1D domains with small pipes. Where a few channels must be much shorter than the rest, it may be economical to specify them as non-inertial channels. The timestep can then be chosen on the requirements of the shortest remaining channel. Care should be exercised when specifying non-inertial channels to ensure that errors are not introduced by the non-inertial representation, particularly if these channels are in a region of particular interest. Any approximations can usually be assessed by a few selected runs without the non-inertial approximation and with the necessary shorter time step.

3.6.3 2D/1D Models

Prior to Build 2005-05-AN, 2D/1D models use the same timestep in both 1D and 2D domains. In these cases, it is highly preferable that the 1D domains do not control the timestep, as 99% of the computational effort is usually in solving the 2D domains.

As of Build 2005-05-AN, different timesteps can be specified for the 1D and 2D domains, offering much greater flexibility in setting timesteps and model resolutions.

3.7 Eddy Viscosity

Two options exist for specifying eddy viscosity for the 2D domains to approximate the effect of fine-scale turbulence. Use the [Viscosity Formulation](#) and [Viscosity Coefficient](#) commands to set the formulation and coefficient.

The first method ([Viscosity Formulation](#) == CONSTANT) applies a constant value throughout the model, irrespective of velocity gradients and variations. This is generally satisfactory when the cell size is much greater than the depth or when other terms are dominant (eg. high bed resistance). The recommended coefficient for the constant formulation is 1 m²/s.

The second (default) method ([Viscosity Formulation](#) == SMAGORINSKY) applies the Smagorinsky formulation as given by the equation below. This formulation is preferred over the CONSTANT option as the coefficient is recalculated every timestep at every cell mid-side according to the change in velocity magnitude and direction. This results in higher coefficients being applied where there is greater turbulence.

Note, if the formulation is changed, the user must also reset the coefficient using [Viscosity Coefficient](#).

As of Build 2008-08-AA, a constant component (C_c in the equation below) can be applied if using [Viscosity Formulation](#) == SMAGORINSKY. The default setting is to have a Smagorinsky coefficient (C_s) of 0.2 and a constant component coefficient (C_c) of 0.1. This gives improved flow patterns around bluff objects. Further testing and research into flow patterns around fine-scale objects is planned.

Smagorinsky Formulation used by TUFLOW:

$$\mu = C_c + C_s A_c \sqrt{\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \frac{1}{2} \left(\left| \frac{\partial u}{\partial y} \right| + \left| \frac{\partial v}{\partial x} \right| \right)^2}$$

where

u and v = Depth averaged velocity components in X and Y directions

x and y = Distance in X and Y directions

μ = Horizontal diffusion of momentum (viscosity) coefficient

A_c = Area of Cell

C_c = Constant Coefficient (default = 0.1)

C_s = Smagorinsky Coefficient (default = 0.2)

A number of other improvements were made as of Build 2008-08-AA as listed below. Note that if [Defaults](#) == PRE 2008-08 is specified, these improvements are not activated.

- Enhanced treatment of the viscosity term at dry boundaries. Builds prior to 2008-08-AA could experience slight flow separation along an oblique dry boundary. This is demonstrated in the top image of Figure 3-5 which shows the velocity magnitude contoured for steady-state flow along a grid orientated 45° to the direction of flow. The blue shades indicate slightly lower velocities. The effect is more pronounced for fine grids (less than 1m) and

low bed resistance, and can cause an over prediction of the water surface slope due to the dry boundary edge effects. The enhanced treatment in Build 2008-08-AA, produces the velocity contouring as shown in the bottom image. No separation occurs, and the velocity increases gradually from left to right as the water depth gradually shallows (the model has a horizontal bed). The correct water surface slope is produced when compared with theory.

- The Smagorinsky formula is now cell-centred (was previously centred around the top right hand cell corner, which is a hang-over of when all input data was centred at the top right corner, ie. the ZH point). The latter improvement now guarantees that symmetry is achieved in hydraulic results (when using a symmetrical model) if using the Smagorinsky formula.



Flow separation along a dry oblique boundary for builds prior to 2008-08-AA



Correct velocity distribution using Build 2008-08-AA

Figure 3-5 Effect of Enhanced Dry Boundary Viscosity Term Treatment

4 Data Input

Section Contents

4	DATA INPUT	4-1
4.1	Control Files – Rules and Notation	4-4
4.2	Simulation Control Files	4-6
4.2.1	TUFLOW Control File (.tcf File)	4-6
4.2.2	1D Domains or ESTRY.exe Control File (.ecf File)	4-8
4.2.3	Run Time And Output Controls	4-9
4.3	GIS Layers	4-10
4.3.1	"GIS" or "MI" Commands	4-10
4.3.2	"RowCol" or "MID" Commands	4-11
4.4	2D Domains (.tgc File)	4-13
4.4.1	The .tgc (Geometry Control) File	4-13
4.4.2	2D Grid Orientation and Dimensions	4-14
4.4.3	2D Cell Codes	4-14
4.4.4	ZC, ZU, ZV and ZH Points (Zpts)	4-15
4.4.5	Building the Topography Layer by Layer	4-17
4.4.6	Zpt (2d_zpt) Layers	4-20
4.4.7	3D Breakline (2d_zln) Layers	4-21
4.4.8	3D TIN (2d_ztin) Layers	4-21
4.4.8.1	<i>Haul Road Example of Using Create TIN Zpts</i>	4-23
4.4.8.2	<i>Highway Embankment Removal Example</i>	4-26
4.4.9	Z Shape (2d_zsh) Layers	4-28
4.4.10	Variable Z Shape (2d_vzsh) Layers	4-33
4.4.10.1	<i>Variable Z Shape Example: Breaching of an Embankment</i>	4-35
4.4.11	Using Multiple Layers and Points Layers	4-39
4.4.11.1	<i>Points Only Layers</i>	4-39
4.4.12	Bed Resistance (Materials)	4-41
4.4.12.1	<i>Materials Files</i>	4-42
4.4.12.2	<i>.tmf Format</i>	4-42
4.4.12.3	<i>.csv Format (Manning's n vs Depth Curves)</i>	4-43
4.4.13	Multiple 2D Domains	4-46
4.5	1D Domains (Networks)	4-48
4.5.1	Nodes and Pits (1d_nwk Layers)	4-48

4.5.1.1	<i>Pit Channels</i>	4-49
4.5.1.2	<i>Pit Inlet Database</i>	4-50
4.5.1.3	<i>Connecting Pits and Nodes to 2D Domains</i>	4-52
4.5.2	Manholes (Automatic and 1d_mh Layer)	4-54
4.5.2.1	<i>Automatically Assigned Manholes</i>	4-54
4.5.2.2	<i>Manually Assigned Manholes (1d_mh Layer)</i>	4-55
4.5.2.3	<i>Digitising Culverts Connected to Manholes</i>	4-57
4.5.2.4	<i>Fixed Manhole Loss Approach</i>	4-57
4.5.2.5	<i>Engelhund Manhole Loss Approach</i>	4-57
4.5.3	Channels	4-60
4.5.4	1d_nwk Attributes	4-64
4.5.5	How are Nodes and Channels Processed?	4-74
4.6	1D Topography	4-75
4.6.1	Channel Hydraulic Properties (CS) Tables	4-75
4.6.2	Node Storage (NA) Tables	4-77
4.6.2.1	<i>Storage (NA) Tables</i>	4-77
4.6.2.2	<i>Using Channel Widths</i>	4-77
4.6.2.3	<i>Storage Above Structure Obverts</i>	4-78
4.6.2.4	<i>Procedure for Assigning NA Tables</i>	4-78
4.6.3	Free-form Tabular Input (1d_tab, 1d_xs, 1d_na, 1d_bg Layers)	4-79
4.6.4	End Cross-Sections	4-82
4.6.5	Interpolated Cross-Sections	4-82
4.6.6	XZ Relative Resistances	4-84
4.6.6.1	<i>Relative Resistance Factor (R)</i>	4-84
4.6.6.2	<i>Material Values (M)</i>	4-84
4.6.6.3	<i>Manning's n Values (N)</i>	4-85
4.6.6.4	<i>Position Flag (P)</i>	4-85
4.6.7	Reducing Conveyance with Height	4-85
4.6.8	Effective Area versus Total Area	4-85
4.7	Hydraulic Structures and Supercritical Flow	4-87
4.7.1	How to Model Bridges and Box Culverts	4-87
4.7.2	2D Flow Constrictions (FC)	4-91
4.7.2.1	<i>2d_fcsh and 2d_fc Layers</i>	4-91
4.7.2.2	<i>Applying FC Attributes</i>	4-96
4.7.2.3	<i>Layered Flow Constrictions (2d_lfcsh Layers)</i>	4-99
4.7.3	2D Upstream Controlled Flow (Weirs and Supercritical Flow)	4-102
4.7.4	1D Hydraulic Structures	4-103

4.7.4.1	<i>Adjustment of Contraction and Expansion Losses</i>	4-103
4.7.4.2	<i>Bridges</i>	4-105
4.7.4.3	<i>Culverts and Pipes</i>	4-106
4.7.4.4	<i>Weirs</i>	4-111
4.7.4.5	<i>Variable Geometry Channels</i>	4-112
4.7.4.6	<i>Non-Inertial Channels</i>	4-112
4.7.4.7	<i>M Channels (User Defined Flow Matrix)</i>	4-113
4.7.4.8	<i>WV Channels (Variable Geometry Weir)</i>	4-114
4.8	Customised Output	4-116
4.8.1	Time-Series Output (2d_po, 2d_lp) from 2D Domains	4-116
4.8.2	Evacuation Routes (2d_zshr)	4-119
4.9	Initial Water Levels (IWL) and Restart Files	4-127
4.9.1	2D Domains	4-127
4.9.2	1D Domains	4-128
4.10	Boundary Conditions and Linking 2D/1D Models	4-129
4.10.1	Boundary Condition (BC) Database	4-129
4.10.2	BC Database Example	4-132
4.10.3	Using the BC Event Name Command	4-134
4.10.4	Recommended BC Arrangements	4-136
4.10.5	Linking 1D and 2D Domains	4-138
4.10.5.1	<i>Linking ESTRY (TUFLOW) 1D Domains</i>	4-138
4.10.5.2	<i>Linking ISIS 1D Domains</i>	4-138
4.10.6	1d_bc Layers	4-141
4.10.7	2d_bc, 2d_sa and 2d_rf Layers	4-144
4.10.7.1	<i>2d_sa Trigger Option</i>	4-145
4.11	Presenting 1D Domains in 2D Output (1d_wll)	4-158
4.11.1	WLL Method A	4-158
4.11.2	WLL Method B	4-159
4.11.2.1	<i>Water Level Lines (WLL)</i>	4-159
4.11.2.2	<i>Water Level Line Points (WLLp)</i>	4-160
4.12	Managing Multiple Events and Scenarios	4-162
4.12.1	Modelling Multiple Events Using an Event File	4-162
4.12.2	Modelling Different Scenarios Using If Scenario	4-166
4.13	Data Processing	4-169
4.13.1	Hierarchy	4-169
4.13.2	XF Files	4-170
4.14	UltraEdit	4-172

4.1 Control Files – Rules and Notation

Control files, such as the .ecf, .tcf, .tbc and .tgc files, are command or keyword driven text files. The commands are entered free form, based on the rules described below. Comments may be entered at any line or after a command. The commands are listed in the index in Appendix G.

An example of a command is:

```
Start Time == 10. ! Simulation starts at 10:00am on 2/9/1962
```

which sets the simulation start time to 10 hours. The text to the right of the “!” is treated as a comment and not used by TUFLOW when interpreting the line.

If using UltraEdit, refer to Section 12.2 for automatic colour coding of files for easy viewing.

The style of input is totally flexible bar a few rules. Commands are not case sensitive and can be repeated as often as needed. This offers significant flexibility and effectiveness when modelling, particularly in building 1D and 2D model topography. Note that a repeat occurrence of a command may overwrite the effect of previous occurrences of the same command.

The rules are:

- A few characters are reserved for special purposes as described in Table 4.1.
- Only one command can occur on a single line.
- A few commands rely on another command being previously specified. These are documented where appropriate.

Table 4.1 Reserved Characters – Text Files

Reserved Character(s)	Description
“#” or “!”	A “#” or “!” causes the rest of the line from that point on to be ignored. Useful for “commenting-out” unwanted commands, and for all that modelling documentation.
==	A “==” following a command indicates the start of the parameter(s) for the command. Where there is more than one parameter, the parameter values are read as free-field formatted, ie. are space or comma delimited.

Additional text can be placed before and/or after a command. For example, a line containing the command [Start Time](#) to set the start time of a simulation to 10 hours can be written as “Start Time == 10” or “Start Time (h) == 10”. The “(h)” text is not a requirement, but is useful to indicate that the units are hours. Alternatively, “Start Time == 10 ! hours” would be acceptable, noting the use of the comment delimiter “!”.

The notation used to document commands and valid parameter values are presented in Table 4.2.

Table 4.2 Notation Used in Command Documentation – Text Files

Documentation Notation	Description
< ... >	Greater than and less than symbols are used to indicate a variable parameter. For example, the commonly used <file> example is described below.
<file>	Is a filename (can include an absolute or relative path, or a URL). Examples are: boundaries.tbc (must be located in same folder as .tcf file) ..\model\boundaries.tbc (this is a relative path – the “..” indicates to move up a level) L:\jb99\tuflow\model\boundaries.tbc (this is an absolute path) \\wbm\ivers\jb99\tuflow\model\boundaries.tbc (this is a URL)
[{Op1} Op2]	The square brackets “[” and “]” surround parameter options. The “ ” symbol separates the options. The “{” and “}” brackets indicate the default option. This option is applied if the command is not used. For example, the options for the Store Maximums and Minimums command are: [ON ON MAXIMUMS ONLY {OFF}] where the default is OFF.
spaces	Spaces can occur in commands and parameter options. If a space occurs in a command, it is only one (1) space, not two or more spaces in succession. Spaces can occur in file and path names.

4.2 Simulation Control Files

4.2.1 TUFLOW Control File (.tcf File)

The TUFLOW Control File or .tcf file sets simulation parameters and directs input from other data sources. It is the top of the tree, with all input files accessed via the .tcf file or files referred to from the .tcf file. An example of a simple .tcf file is shown further below.

The final .tcf file must reference as a minimum:

- one .tgc file using [Geometry Control File](#) for each 2D domain;
- one .tbc file using [BC Control File](#) for each 2D domain;
- an .ecf file using [ESTRY Control File](#) if there are any 1D domains (this file is now optional as of the 2010-10 release – see below); and
- a .tmf file using [Read Materials File](#) if material (land-use) polygons are being used.

For the 2010-10 release, it is now possible to incorporate 1D (.ecf) commands into the .tcf file, with the use of an .ecf file now optional. 1D (.ecf) commands can be included in the .tcf file between [Start 1D Domain](#) and [End 1D Domain](#) block(s). For the new .tcf file 1D (.ecf) commands must be prefixed by [1D](#). Also note that [ESTRY Control File](#) is now recognised every time if specified more than once (in previous releases only the last occurrence would apply).

Other mandatory or most commonly used commands are: [BC Database](#); [End Time](#); [Map Output Data Types](#); [Map Output Interval](#); [MI Projection](#); [Output Folder](#); [Start Time](#); [Store Maximums and Minimums](#); [Time Series Output Interval](#); [Timestep](#); [Write Check Files](#); [Write Empty MI Files](#);

The [Read File](#) command is extremely useful for placing commands that remain unchanged or are common for a group of simulations in another file (eg. the [MI Projection](#) command will be the same for all runs within the same study area). This reduces the size/clutter of .tcf files and allows easy global changes to a group of simulations to be made.

Other commonly used or useful commands are: [BC Event Name](#); [BC Event Text](#); [Cell Wet/Dry Depth](#); [Cell Side Wet/Dry Depth](#); [Instability Water Level](#); [Read GIS FC](#); [Read GIS IWL](#); [Read GIS PO](#); [Screen/Log Display Interval](#); [Set IWL](#); [Start Map Output](#); [Start Time Series Output](#); [Viscosity Coefficient](#); [Viscosity Formulation](#); [Write PO Online](#).

In UltraEdit, the commands and comments can be colour coded for easier viewing (see Section 12.2).

```
# This is an example of a simple .tcf file
! Comments are shown after a "!" or "#" character.
! Blank lines are ignored.  Commands are not case sensitive.

! Set the geographic projection
MI Projection == ..\model\mi\Projection.mif

BC Control File == ..\model\boundaries.tbc  ! boundary control file
Estry Control File == model.ecf  ! linked ESTRY model control file
Geometry Control File == ..\model\topography.tgc  ! topography control file
```

```
Start Time (h) == 0.  
End Time (h) == 12.  
Timestep (s) == 5  
  
Read Materials File == ..\model\n_values.tmf ! .tmf is for Tuflow Materials File
```

Appendix A lists and describes .tcf commands and their parameters.

4.2.2 1D Domains or ESTRY.exe Control File (.ecf File)

The 1D domains or ESTRY.exe Control File (.ecf file) sets simulation parameters and directs input from other data sources for all 1D domains. An example of a simple .ecf file for a **1D only model** (ie. no 2D linkage and simulated using ESTRY.exe) is shown below. The example as is used if there are 1D domains in a 2D/1D model is shown further down.

```
# This is an example of a simple .ecf file for a 1D only model run

! Set the geographic projection
MI Projection == ..\model\mi\Projection.mif

! Set simulation time parameters
Start Time (h) == 0.0
End Time (h) == 10.0
TimeStep (s) == 30
Start Output (h) == 0.0
Output Interval (h) == 0.5

! Read in the 1D network
XS Database == m11.txt ! using a MIKE 11 processed data file for X-sects
Read GIS Network == ..\model\mi\1d_nwk_example.mif

! Set the initial water level
Set IWL == 1.

! Read in the boundary condition locations and values
BC Database == ..\bc_dbase\bc_dbase.csv
BC Event Text == __event__
BC Event Name == Q100
Read GIS BC == ..\model\mi\1d_bc_example.mif
```

For a 2D/1D model, the control file for the 1D domains for the same .ecf file above would look something like the below. Note that a number of the commands are not needed as they would have been specified in the .tcf file. Commands that are only relevant for 1D only models are indicated with a “1D Only” underneath the command in Appendix B.

```
# This is an example of a simple .ecf file used for a 2D/1D model run

! Set simulation time parameters
Start Output (h) == 0.0
Output Interval (h) == 0.5
! Read in the 1D network
XS Database == m11.txt ! using a MIKE 11 processed data file for X-sects
Read GIS Network == ..\model\mi\1d_nwk_example.mif

! Set the initial water level
Set IWL == 1.

! Read in the boundary condition locations and values
Read GIS BC == ..\model\mi\1d_bc_example.mif
```

Appendix B lists and describes .ecf commands and their parameters.

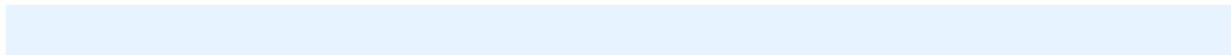
4.2.3 Run Time And Output Controls

All time-dependent data must be referred to an arbitrary time reference, which is defined by the simulation time commands.

For 2D/1D models these are [Start Time](#), [End Time](#) and [Timestep](#) in the .tcf file. For 1D Only models these are [Start Time](#), [End Time](#) and [Timestep](#) in the .ecf file.

The starting time and finishing times specify the period in hours for which calculations are made. The timestep is the calculation interval in seconds, which is dependent on various conditions as described in Section 3.6. Prior to Build 2005-05-AN, for 2D/1D models the same timestep is used for both 2D and 1D schemes. In these cases, it is highly preferable that the 1D domains do not control the timestep, as 99% of the computational effort is in usually in solving the 2D domains.

The output data is controlled by the times set using [Start Map Output](#) and [Start Time Series Output](#) for the 2D domains, and [Start Output](#) for the 1D domain. All outputs are limited to the period between these times and the end time. In determining the maximum and minimum hydraulic values, every calculation time step is considered (see [Store Maximums and Minimums](#) for 2D domains, while for the 1D domains the maximums and minimums are always output).



4.3 GIS Layers

GIS data layers are transferred into and out of TUFLOW using the MapInfo data exchange .mif or ArcGIS .shp format. These formats are industry standards and publicly available. The .mif format is in text (ASCII) form, making it particularly easy to work with. Most mainstream CAD/GIS platforms recognise these formats.

.shp file recognition for GIS input layers was introduced for the 2010-10 release. Releases prior to 2010-10 do not recognise .shp layers.

All GIS layers imported or exported by TUFLOW must be in the **same geographic projection**. For .mif layers use the [MI Projection](#) and [Write Empty MI Files](#) commands (see first few steps of Section 6.2, Setting up a New Model). Similar commands for .shp formats will be available in a future release.

TUFLOW interprets GIS data and the data objects (points, lines, etc) as described in the following sections.

To appreciate how TUFLOW interprets GIS data it is important to understand the following.

- .mif or .shp files contain the geometrical (map) data about the objects.
- .mid or .dbf files contain the attribute data of the objects.

If using builds prior to Build 2007-07-AA, please note the following:

- The syntax of the objects in the .mif files is case sensitive to TUFLOW. Therefore .mif files exported by GIS/CAD software that use different case syntax to that used by MapInfo, are now correctly interpreted by TUFLOW. Prior to this build, the syntax of MIF objects was case sensitive (according to how MapInfo writes the files).
- The case sensitive syntax for MIF Objects prior to Build 2007-07-AA is:
none; Arc; Ellipse; Line; Pline Multiple; Pline; Point; Rect; Region; Roundrect; Text; Multipoint; Center; Collection
- TUFLOW would stop if a region object did not have its centre explicitly specified in the .mif file using “Center”. As of Build 2007-07-AA, a region’s centre is automatically calculated if it is not specified (some CAD/GIS software when exporting .mif files do not specify the centre of a region object). Note that the centre of a region is only used by a few of TUFLOW’s GIS layers such 2d_fc and 2d_po layers (see Table 4.3).

4.3.1 “GIS” or “MI” Commands

Commands containing “GIS” or “MI” read and/or write a GIS layer. GIS and MI are interchangeable (the GIS syntax was introduced for the 2010-10 release with the introduction of .shp files). The format of the GIS layer is based on the filename’s extension. For example, [Read GIS Zpts](#) and [Read MI Zpts](#) are identical.

GIS or MI commands read both the .mif and .mid files (if a .mif extension is specified), and the .shp and .dbf files (if .shp is specified).

The geographic location of objects for GIS or MI commands **is important** (unlike for RowCol or MID commands as discussed below). The geographic position of the object controls which part of the TUFLOW model they effect.

For backward compatibility for MI only (not GIS), when specifying the .mif/.mid filename, the extension may be omitted, or either of the .mif or .mid extensions may be used. For example, all of the lines below would be interpreted in the same way:

```
Read MI Code == ..\model\mi\2d_code_buildings.mif
Read MI Code == ..\model\mi\2d_code_buildings.mid
Read MI Code == ..\model\mi\2d_code_buildings
```

Table 4.3 defines the different data objects supported.

When digitising objects, it is preferable that they do not snap to the 2D cell sides or corners as this may produce indeterminate effects.

As of Build 2007-07-AA, a much faster algorithm was incorporated for interrogating whether a cell or point falls within a region/polygon. The new feature can markedly reduce the startup time of large models, and allows rapid interpretation of polygons with tens of thousands of vertices. To use the old algorithm see [Inside Region](#).

4.3.2 "RowCol" or "MID" Commands

Commands containing "RowCol" or "MID" (eg. [Read RowCol Zpts](#)) only read the attribute data file (ie. .mid or .dbf file). The .mif or .shp file is not used. These commands rely on the first two columns of the attribute data to define the row and column of a 2D cell reference (ie. n,m or row,column). Attribute data in subsequent columns depends on the type of GIS layer. It is not necessary for the user to create these layers manually, as TUFLOW produces them, or they can be easily created from the GIS check files that TUFLOW writes.

Creating/moving an object using GIS in a layer that is read by a "RowCol" or "MID" command should never occur and has misleading effects.

The RowCol or MID option is normally only used for Zpts (see [Read RowCol Zpts](#)) and sometimes for materials.

The filename must specify the .mid or .dbf extension (.csv may also be used as of Build 2010-10-AA). For example, the following lines are effectively the same.

```
Read RowCol Zpts == ..\model\gis\2d_zpt_DTM.mid
Read RowCol Zpts == ..\model\gis\2d_zpt_DTM.dbf
Read RowCol Zpts == ..\model\gis\2d_zpt_DTM.csv
```

Table 4.3 TUFLOW Interpretation of GIS Objects

Object Type	TUFLOW Interpretation
Used Objects	
Point	Refers to the 2D cell that the point falls within or a 1D node. Points snapped to the sides or corners of a 2D cell may give uncertain outcomes as to which cell the point refers to.
Line (straight line)	Continuous line of 2D cells, 1D channel, connects other objects, alignment of a 3D breakline and other applications. Note, the algorithm for selecting the 2D cells has varied with different TUFLOW builds. See Line Cell Selection and Boundary Cell Selection .
Pline (line with one or more segments)	As for Line above.
Region (polygon)	For 2D cells either: - Modifies any 2D cell or cell mid-side/corner (eg. Zpt) that falls within the region. If the command is modifying a whole 2D cell, it uses the cell's centre to determine whether the cell falls inside or outside of the region. If the cell's centre, mid-side or corner lies exactly on the region perimeter, uncertain outcomes may occur. Holes within a region are accepted except for polygon objects in shape layers used for TIN boundaries. - Or, just uses the region's centroid. Examples are the original flow constriction layers (2d_fc) and time-series output locations (2d_po). For 1D selects 1D nodes within the region. If the 1D node falls exactly on the region perimeter uncertain outcomes may occur.
Multiple (Combined) Objects	In later versions of TUFLOW, multiple point, polyline and region objects are generally accepted (ERROR or WARNING messages are given if not the case).
Unused (Ignored) Objects	
Arc	Ignored (do not use).
Collections	Not supported. Collections are groups of objects of differing type.
Ellipse	Ignored (do not use).
none	These objects are ignored and most commonly occur when a line of attribute data is added that is not associated with an object. In MapInfo, this occurs when a line of data is added directly to a Browser Window (ie. no object was digitised).
Roundrect (Rounded Rectangle)	Ignored (do not use).
Rect (Rectangle)	Ignored (do not use).
Text	Ignored.

4.4 2D Domains (.tgc File)

2D domains are created by building them through a series of commands contained in .tgc files. The .tgc file contains, or accesses from other files, information on the size and orientation of the grid, grid cell codes, bed/ground elevations, bed material type or flow resistance value, and optional data such as ripple height, wave climate, wind field, etc.

A 2D domain is automatically discretised as a grid of square cells. Each cell is given characteristics relating to the topography such as ground/bathymetry elevation, bed resistance value and initial water level, etc.

Only one .tgc file per 2D domain is specified in the .tcf file using [Geometry Control File](#).

4.4.1 The .tgc (Geometry Control) File

Rather than contain all the 2D grid information in one file, the .tgc file is a series of commands that **builds** the model. The commands are applied in sequential order, therefore, it is possible to override previous information with new data to modify the model in selected areas. This is very useful where a base data set exists, over which areas need to be modified to represent other scenarios such as a proposed development. This eliminates or minimises data duplication.

The commands can occur in any order (as long as it is a logical one!).

If an unrecognisable command occurs, TUFLOW stops and displays the unrecognisable text.

Notes & Tips:

- 1 Any command can be repeated any number of times.
- 2 Commands are executed in the order they occur. If the data for a 2D cell or Zpt is supplied more than once, the last data read is that used, ie. the latter data for a cell overrides any previous data for that cell.
- 3 The .mid file is a comma delimited text file. It can be created not only by exporting a MapInfo table but also by using Excel, a text editor or a purpose written translator.
- 4 The .mid file accessed by a [Read GIS](#) or [Read RowCol](#) command does not have to contain data for the entire model. If you wish to modify just a few cell values or Zpts, the file only needs to contain these cells/Zpts.
- 5 Use [Write Check Files](#) commands to cross-check and carry out quality control checks on the final 2D grid and Zpts.

An example of a .tgc file is shown below.

```
# Setup the location of the 2d domain
Read GIS Location == ..\model\gis\2d_loc_my_model.mif ! Locate 2D domain
Cell Size == 10. ! Set cell size to 10m

# Setup the base topography
Set Code == 1 ! Set everywhere as water
Read GIS Code BC == ..\model\gis\2d_bc_my_model.mif ! Read codes from 2d_bc
```



```
# Setup the base topography
Set Zpts == 100. ! Set elevations everywhere to 100 m
Read RowCol Zpts == ..\model\gis\2d_zpt_dtm.mid ! Read the DTM Zpts
Read GIS Z Lines == ..\model\gis\2d_zlr_levees.mif ! Apply 3D lines along levees
Read GIS Zpts == ..\model\gis\2d_zpt_fill.mif ! Proposed filling of floodplain

# Setup the materials
Set Mat == 1 ! Set default material value to 1
Read GIS Mat == ..\model\gis\2d_mat_land use.mif ! Read materials distribution
```

4.4.2 2D Grid Orientation and Dimensions

Each 2D domain is a rectangle at any orientation. The orientation and dimensions are defined using .tgc file commands. For the orientation it is recommended that the X-axis falls between 90° and –90° of East as it is preferable to view the 2D grid within this range and some post-processing software only operate within this range.

Several options are available for setting the grid location and orientation as a result of a number of new commands being introduced over the years. In all cases, [Cell Size](#) must be specified. The options are:

- Using a four-sided polygon in a GIS layer to define the 2D grid orientation and dimensions (see [Read GIS Location](#)).
- Using a line (two vertices only) in a GIS layer to define the orientation of the X-axis (see [Read GIS Location](#)), and [Grid Size \(N,M\)](#) or [Grid Size \(X,Y\)](#) to set the 2D grid X and Y dimensions.
- Using [Origin](#), [Orientation](#) or [Orientation Angle](#), and [Grid Size \(N,M\)](#) or [Grid Size \(X,Y\)](#). No GIS layers are required for this option.

It is not essential at any point to specify dimensions that are an exact multiple of [Cell Size](#).

4.4.3 2D Cell Codes

Each cell in a 2D domain is assigned a code to indicate its role. It must have a value of one of the types in Table 4.4. As of Build 2002-01-AC, the default code value is one (1) or “water”.

Commands used to modify the cell codes are [Set Code](#), [Read GIS Code](#) (or [Read GIS Code BC](#)), [Read RowCol Code](#) in the .tgc file, and [Read GIS BC](#) in the .tbc file automatically sets the Boundary Cell code of 2 along external boundaries.

As of Build 2008-08-AA, a useful option for setting the cell codes for 2D-2D linked domains is the INVERT flag (see [Read GIS Code](#) or [Read GIS Code BC](#)), which allows the same code layer/polygons to be used to both 2D domains.

Table 4.4 Cell Codes

Type	Code	Description
Null Cells	-1	Inactive cells used to deactivate cells within the active domain. Null cells are often preferred to land cells as they are not excluded when TUFLOW outputs in SMS format. For two simulations to be compared in SMS, they must have exactly the same mesh. If an area in a model is removed (eg. filling part of a floodplain), use null cells or raise the ground elevations in preference to using land cells so that the two simulations can be compared. Note: In earlier versions of TUFLOW null cells were used to indicate the outside side of an external boundary – this is no longer the case. Cells on the outside of a boundary can be either a land or a null cell.
Land or Redundant Cells	0	Land cells are cells that are totally removed from the computation. The name “land” comes from coastal hydraulic studies where the land was the permanently dry area. Maximising the area of land cells reduces computation time and output file sizes.
Water or Active Cells	1	Water cells are active cells that can wet and dry.
Boundary Cells	2	Boundary cells indicate water cells that have an external boundary (including some types of 2D/1D dynamic links). At an external boundary there must be a water cell on one side and a null or land cell on the other. Note: It is not necessary to manually specify each boundary cell. Boundary lines are digitised in the GIS and TUFLOW automatically assigns the boundary code to the cells (see Section 4.10.7 and Read GIS BC).

4.4.4 ZC, ZU, ZV and ZH Points (Zpts)

The model topography is defined by elevations at the cell centres, mid-sides and corners. Each cell has the following elevations assigned to it as shown in Figure 4-1:

- “C” Zpt (ZC) – middle of cell
- “U” Zpt (ZU) – middle right of cell
- “V” Zpt (ZV) – middle top of cell
- “H” Zpt (ZH) – top right hand corner of cell

One of most important aspects of TUFLOW modelling is to understand the roles of the elevation (Zpt) points.

The ZC point:

- defines the volume of active water (cell volume is based on a flat square cell that wets and dries at a height of ZC plus the [Cell Wet/Dry Depth](#));
- controls when a cell becomes wet and dry (note that cell sides can also wet and dry); and

- is used to determine the bed slope when testing for the upstream controlled flow regime (see Section 4.7.3).

The ZU and ZV points:

- control how water is conveyed from one cell to another;
- are where the momentum equation terms are centred and where upstream controlled flow regimes are applied;
- are deactivated if the cell has dried (based on the ZC point) and cannot flow; and
- can wet and dry independently of the cell wetting or drying (see [Cell Wet/Dry Depth](#) and [Cell Side Wet/Dry Depth](#)) which allows the modelling of “thin” obstructions such as fences and thin embankments relative to the cell size (eg. a concrete levee).

ZH points:

- play no role hydraulically;
- are, by default, the only elevations to be written to the SMS .2dm mesh file (by default, all output is interpolated/extrapolated to the cell corners. (Note: As of Build 2008-08-AA, the [Map Output Format](#) == SMS HIGH RES option is available for outputting elevations and hydraulic calculations at the cell centres, mid-sides and corners.)

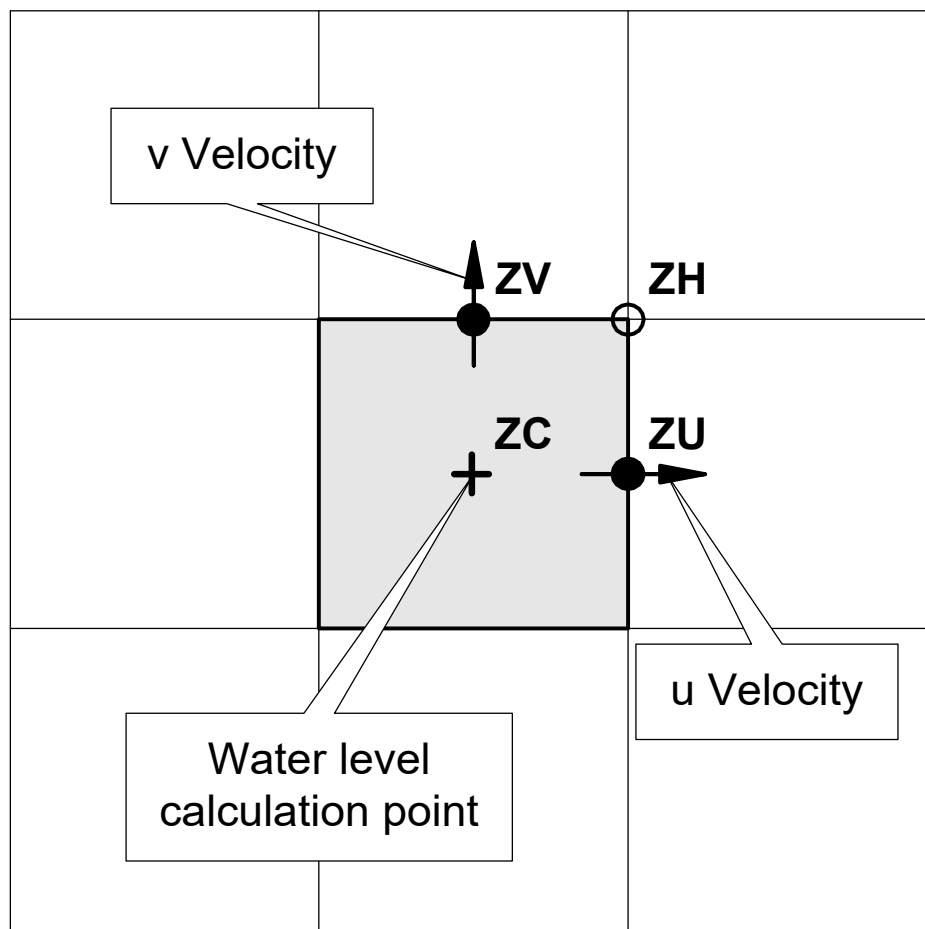


Figure 4-1 Location of Zpts and Computation Points

4.4.5 Building the Topography Layer by Layer

A powerful feature of TUFLOW is its capacity to build the 2D elevations from any number of GIS layers and/or TINs. The general approach is as follows, noting that a number of the capabilities and commands mentioned below are only available from Build 2008-08-AA onwards.

- 1 A default elevation is specified first in the .tgc file using [Set Zpt](#). A flood-free elevation is usually specified.
- 2 The elevations sampled from a DEM using [Read RowCol Zpts](#) is next. These elevations will override those previously specified wherever they exist. As of Build 2008-08-AA, there is an alternative option to interpolate elevations from a SMS or 12D TIN (see [Read TIN Zpts](#)) or to have TUFLOW create a TIN (see [Create TIN Zpts](#)).
- 3 If there are areas where there are missing or erroneous data, these areas are easily interpolated across using [Read GIS Z Shape](#) regions or polygons. Simply digitise a polygon around the missing or erroneous data and TUFLOW interpolates elevations across the region based on the existing Zpts around the perimeter of the polygon. An example is shown in Section 4.4.9. (Note: missing or erroneous data often occurs with aerial surveys where the sampling in areas of thick vegetation, water or the removal of elevated objects (eg. buildings) gives a poor representation of the topography.)
- 4 Sometimes there is a need to remove cut and fill works from the topography. For example, if modelling an historical event to calibrate the model, some of the levees, road embankments and developments may not have existed at the time of the event, and need to be removed. This is easily done using [Read GIS Z Shape](#) by simply digitising polygons around the various works.
- 5 The base elevations set up in the previous step(s) can be modified to represent hydraulic controls, proposed works, failure of a flood defence wall, etc. Some examples are:
 - (a) The crests of road/rail embankments, levees, fences and other solid obstructions are easily inserted using [Read GIS Z Line](#), [Read GIS Z Shape](#) or [Read GIS Z HX Line](#). [Read GIS Z Shape](#) is particularly powerful as 3D lines can be given a thickness making it very easy to quickly raise, or lower, elevations along a road alignment or a diversion channel where the width of the embankment/channel is wider than the 2D cell size.
 - (b) The proposed cut and fill for a development or other works can be incorporated using [Read GIS Zpts](#), [Read GIS Z Shape](#), [Read TIN Zpts](#) or [Create TIN Zpts](#). These powerful commands can set elevations based on regions. Within regions TINs can be generated from points and lines, and the perimeter of the TIN can be automatically merged with the existing Zpts. A TIN of the cut and fill produced by SMS or 12D can be read directly if so desired using [Read GIS Zpts](#).
- 6 If there is a need to simulate the failure of a flood defence wall or road/rail embankment, or the collapse of fences, [Read GIS Variable Z Shape](#) can be used to control the collapse of the embankment or fences over time. The collapse can be triggered to occur at a specified time or when a water level, or a water level difference between two locations, elsewhere in the model is reached.

A 2D domain's Zpts are built up using one or more of the commands shown in Table 4.5.

Table 4.5 2D Zpt Commands

Command	Description
Set Zpt	Sets all Zpts over the whole 2D domain to the same value. Useful for providing an initial elevation prior to other commands as some Zpts in inactive (land) parts of the model may not receive a value. The default value for all Zpts is 99999.0. As of Build 2007-07-AA, every Zpt must be assigned a value, essentially making this command mandatory.
Read RowCol Zpts	Normally used to set the Zpts generated from a DTM. Use the Write MI Zpts command for TUFLOW to create a GIS layer of Zpts with no elevations (TUFLOW only writes out Zpts at active (non-land) cells). This layer is then imported to a GIS, each Zpt assigned an elevation from a DTM and then exported for use by the Read RowCol Zpts command. Note the use of RowCol (or MID) in the command, which indicates that only the .mid file or .dbf file is read (ie. do not specify a .mif or .shp file when using this command).
Read GIS Zpts	Note, this command is different to the Read RowCol Zpts command above. It is typically used for simple modifications of sections of the topography. Examples are filling an area (defined by a region or polygon object) to the same elevation, or dredging (lowering) a section of river using Read GIS Zpts ADD. As of Build 2008-08-AA, Read GIS Z Shape offers greater functionality and maybe preferable to using Read RowCol Zpts to modify Zpts.
Read GIS Z Line	Reads 3D breaklines (defined as a polyline with elevation points) to modify the nearest Zpts to the height of the line. A powerful command for ensuring the crest height of ridges (levees, embankments, etc) is correctly modelled. A number of options exist for this command. Also see Allow Dangling Z Lines and Pause When Polyline Does Not Find Zpt. As of Build 2008-08-AA, Read GIS Z Shape offers greater functionality and maybe preferable in some instances.
Read GIS Z HX Line	Similar to Read GIS Z Line , but uses HX lines and ZP points in a 2d_bc layer (see Table 4.33) to adjust the 2D cell elevations along HX lines.
Create TIN Zpts	Useful for having TUFLOW create a TIN using polygons for TIN boundaries and points and lines within the polygons to create the TIN. If no points are snapped to the perimeter vertices of a polygon, the elevations around the polygon's perimeter are merged with the current Zpt values. The resulting TIN can optionally be exported to SMS and 12D TIN formats.
Read TIN Zpts	Reading of a TIN in SMS or 12D format to set the Zpt values within the TIN.

Command	Description
Read GIS Z Shape	Powerful command to modify Zpt values using points, lines and polygons. Lines can vary in width from just the cell sides (thin), whole cells (thick) or be assigned a width (thickness) in metres. TINs are created within the polygons and incorporate elevations from points and lines that fall within the polygon. The perimeter of the polygon can be merged with the current Zpt values in part, or in its entirety. Read GIS Z Shape and Create TIN Zpts are excellent for removing bad data areas and for filling in null areas where the aerial survey has provided poor or no data. Another example is to remove buildings from a DEM.
Read GIS Variable Z Shape	Allows the user to define the eroded 3D shape of a section of the 2D domain, specify the period for collapse, and how the collapse is triggered (ie. at a specified time or when a water level is reached or when a water level difference is exceeded). Raising of the Zpts over time is also permitted (eg. to model the influence of a landslide filling a river).
Maximum Points Maximum Vertices	Use these commands to increase the maximum number of elevation points or maximum number of vertices in a single polyline or polygon.
Default Land Z	Now rarely used in lieu of Set Zpt.
Interpolate ZC Interpolate ZHC Interpolate ZUV Interpolate ZUVC Interpolate ZUVH	Allows the interpolation of Zpts from other types of Zpts. Now rarely used as nearly all models assign values directly to all the Zpts. The original TUFLOW code only required input of ZH points, and Interpolate ZUVC provided a tool for interpolating the other Zpts. Models with “bumpy” terrain, such as that from airborne laser surveys, might benefit from using Interpolate ZHC or Interpolate ZUV. Models through urban areas where the DTM includes the buildings may benefit from using Interpolate ZC ALL LOWER, which reduces the amount of cells that become blocked out due to high ZC elevations from buildings.
ZC == MIN(ZU,ZV)	Rarely used.

4.4.6 Zpt (2d_zpt) Layers

2d_zpt layers contain points at the ZC, ZU, ZV and ZH locations, with each point being assigned an elevation. The layer is usually initially generated by TUFLOW using [Write MI Zpts](#), which creates the .mif and .mid files, and elevations are assigned to the fourth (Elevation) attribute using a DEM in 3D surface modelling software (for example, by using the Point Inspection Analysis tool in Vertical Mapper). The 2d_zpt attributes are listed in Table 4.6.

Unlike nearly all other layers, TUFLOW only reads the .mid file of the 2d_zpt layer. The .mid file contains the attribute data in comma delimited format. [Read RowCol Zpts](#) (note the MID) reads the attribute data and locates the elevation within the 2D domain using the n, m and Type attributes. n and m (row and column) defines the 2D cell, and Type defines whether it is the ZC, ZU, ZV or ZH elevation point. The fourth attribute is the elevation value (note, that it does not have to be called Elevation, in fact it is good practice to label the fourth attribute after the DEM for data traceability).

[Read RowCol Zpts](#) can be used as many times as needed. Usually the first time is to read in the DEM elevations. If there are elevations that need to be manually modified, the points can be selected in the GIS, saved as a separate layer (note, only save the points that need to be modified), and the Elevation attribute values edited (do not change any of the first three attribute values). In the .tgc file, a second instance of [Read RowCol Zpts](#) can be used to read in the new elevation values to override the original ones.

Table 4.6 2d_zpt Attribute Descriptions

No.	GIS Attribute	Description	Type
1	n	2D grid row	Integer
2	m	2D grid column	Integer
3	Type	One of “C”, “U”, “V” or “H” to indicate whether the elevation is at the ZC, ZU, ZV or ZH location as shown in Figure 4-1.	Char(1)
4	Elevation	Elevation of the point.	Float

4.4.7 3D Breakline (2d_zln) Layers

2d_zln layers are used for adjusting the closest Zpt elevations to lines representing the crests or highpoint of levees, flood defences, roads, railways and other embankments that obstruct flow. The lines can be 3D in shape and height using elevation points. For further information see [Read GIS Z Line](#).

Prior to Build 2008-08-AA, 2d_zln layers using [Read GIS Z Line](#) were the most powerful method for easily manipulating the 2D geometry to represent embankments and gullies. They continue to be supported, however, users should familiarise themselves with the range of new capabilities incorporated into Build 2008-08-AA as described in the following sections. One of the advantages of continuing to use 2d_zln layers is that they only require one attribute (Z or Elevation) and are therefore simpler to use.

Users should also be aware of the new [Read GIS Z HX Line](#) for assigning Zpt elevations along HX lines.

4.4.8 3D TIN (2d_ztin) Layers

TINs (triangulations) of elevation points and 3D lines within a polygon can be carried out using [Create TIN Zpts](#). This is particularly useful for modifying the Zpt elevations where there have been, or are proposed, changes to the base DEM Zpt values.

This feature can also be used as an alternative to using [Read RowCol Zpts](#) for the base DEM elevations, although it is noted that if dealing with large data sets, it is likely to be much more efficient to use 3D surface modelling software to triangulate the data, and [Read RowCol Zpts](#) or [Read TIN Zpts](#) to read the data into TUFLOW.

The protocols applied are:

- 1 A TIN is created for each polygon in the 2d_ztin layer.
- 2 Any points found within a polygon are used when generating the TIN.
- 3 Any lines are converted to points, and those points falling within the polygon are used for the TIN creation. Lines are converted to points as follows:
 - (a) All vertices (nodes) of the line are converted to points.
 - (b) The dMax attribute is used to insert additional vertices between the line's vertices. For example, if dMax is set to 10, then additional intermediate vertices are inserted at least every 10 metres (or less) between the existing vertices where the distance between the existing vertices exceeds 10m. If the dMax attribute does not exist or is zero, half the 2D domain's [Cell Size](#) is used as the dMax value.
 - (c) If there are any points snapped to the line's vertices, the elevations of these points are used to set the elevations at all the vertices generated along the line. In this way, a 3D breakline effect can be produced within the TIN. If there are no points snapped to the line, the line's elevation is used giving the effect of a horizontal line.

4 The perimeter of the polygon/TIN can either be merged with the current Zpt values or have its own values as follows:

- (a) If there are no points snapped to the perimeter of the polygon, the elevations of the polygon's perimeter vertices, and of any automatically inserted vertices, are based on the current Zpt values (ie. the Zpt values assigned by any prior commands).
- (b) If there are one or more points snapped to the polygon's perimeter vertices, the perimeter is not merged with the Zpt values, and the elevations of the snapped points are used to assign elevations to the perimeter vertices and any automatically inserted vertices.
- (c) The frequency of any automatically inserted points around the perimeter is controlled by the dMax attribute. If the dMax attribute does not exist or is zero, half the 2D domain's [Cell Size](#) is used.

A useful quality control option of [Create TIN Zpts](#) is the WRITE TIN option. If this option is specified, a SMS .tin file is written for each TIN generated, and the triangles are written to the 2d_sh_obj_check.mif layer. This means that the TIN can be cross-checked in SMS, viewed in 3D, and edited and modified if desired. [Read TIN Zpts](#) can be used to assign Zpt elevations from the modified SMS TIN.

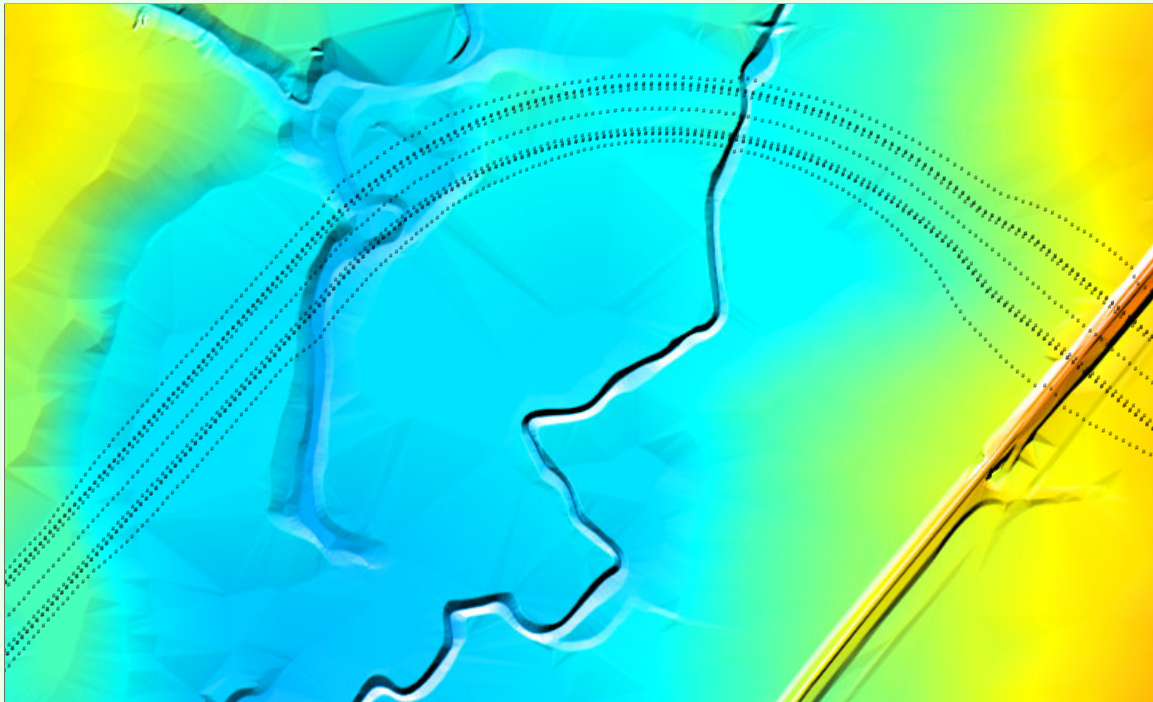
Table 4.7 2d_ztin Attribute Descriptions

No.	GIS Attribute	Description	Type
1	Z	Point: Elevation of the point. Line: Elevation of the line. Ignored if there are any points snapped to the line's vertices. Polygon: Not used.	Float
2	dMax (optional)	Point: Not used. Line: Maximum distance between automatically added intermediate vertices. If set to zero or this attribute does not exist, half the 2D domain's Cell Size is used. If less than zero no intermediate vertices are inserted. Polygon: Same as for Line above.	Float

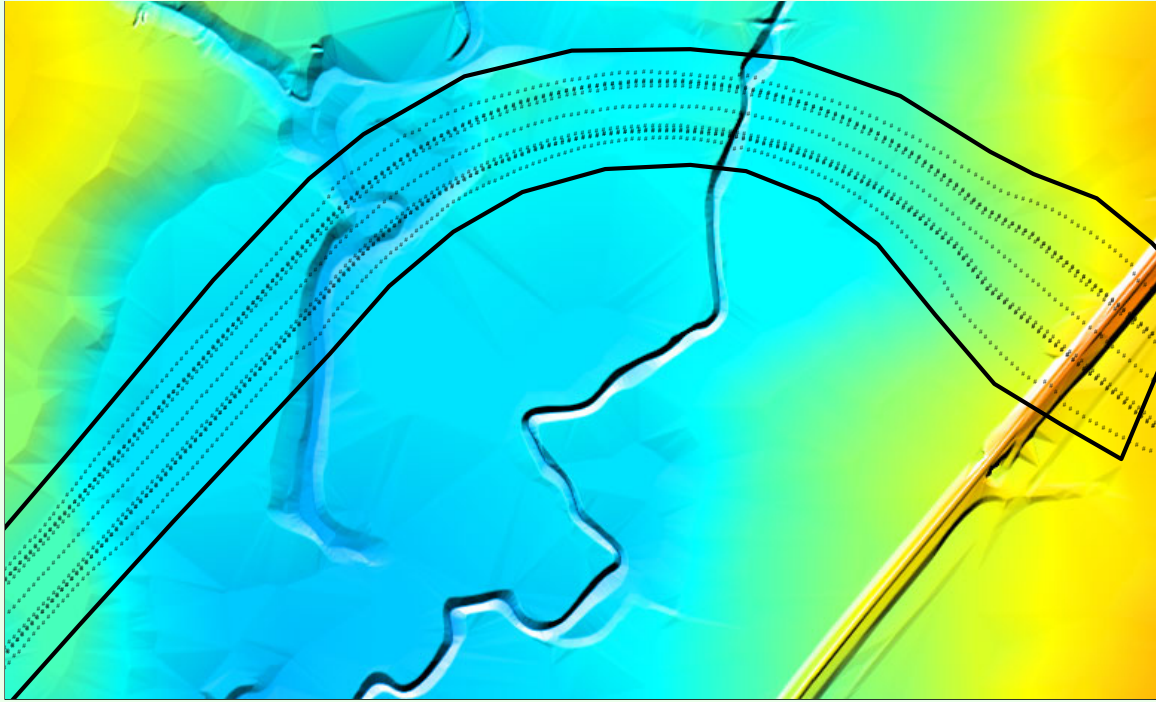
4.4.8.1 Haul Road Example of Using Create TIN Zpts

The images and text below illustrate an example of using [Create TIN Zpts](#) to very easily incorporate a proposed haul road into a TUFLOW model. [Read GIS Z Shape](#) can also be used in a similar manner. The data for this example was provided courtesy of Hatch (www.hatch.com).

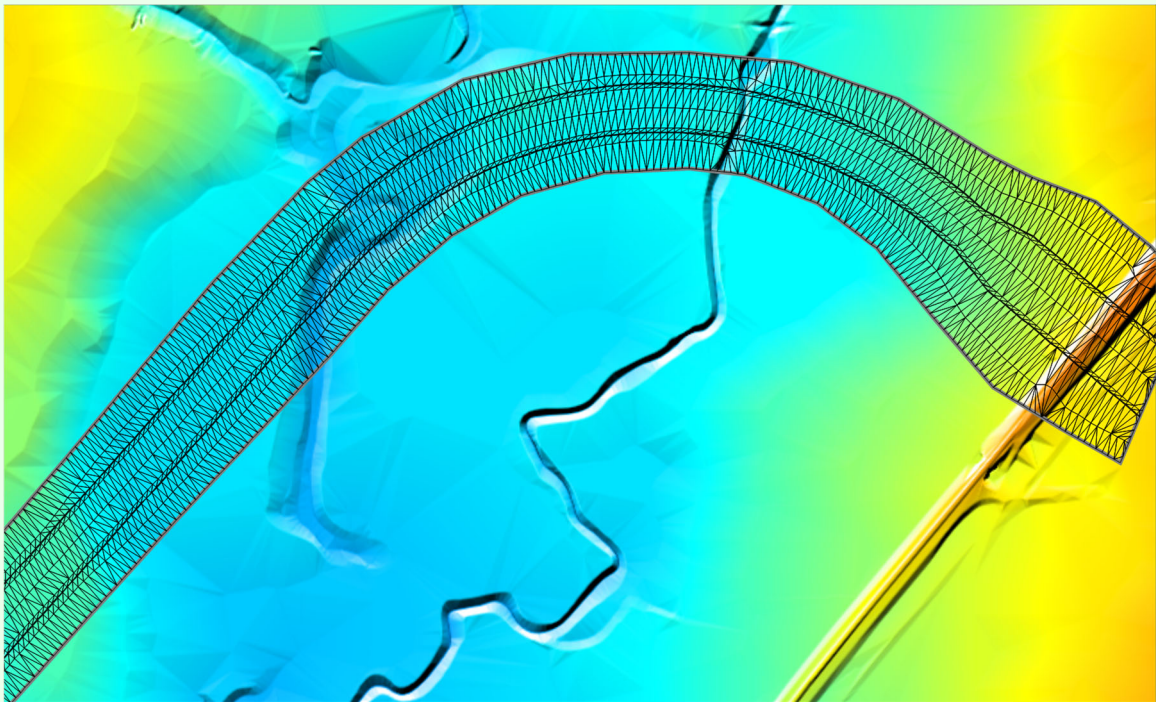
The image below shows the base DEM and a layer of points provided from a civil design CAD package in XYZ format. The points represent a proposed haul road to be built across the floodplain.



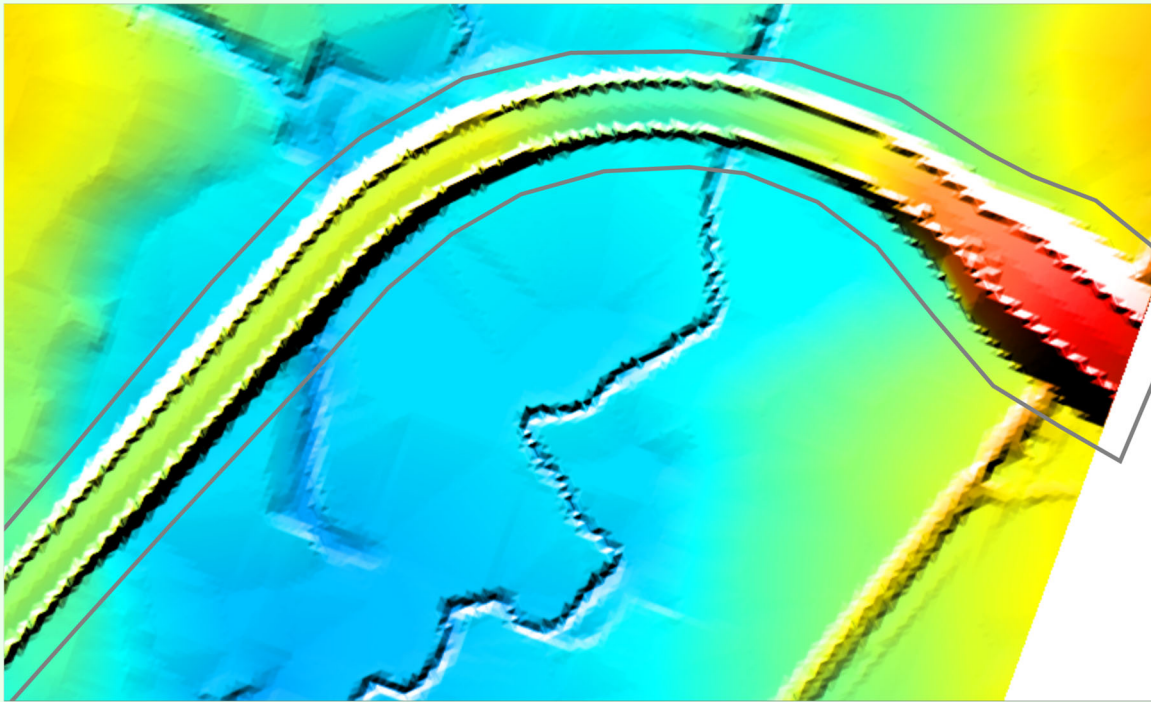
The points were copied into a MapInfo layer named `2d_ztin_haul_road_pts.tab`, with the only attribute being the Elevation attribute that contains the Z values (see Section 4.4.11). A new layer was created and named `2d_ztin_haul_road.tab`, and the polygon shown as the black solid line in the image below was digitised. Note the polygon does not need to snap to any of the points and can be digitised roughly. Alternatively the points and polygon can be placed in the same layer if desired.



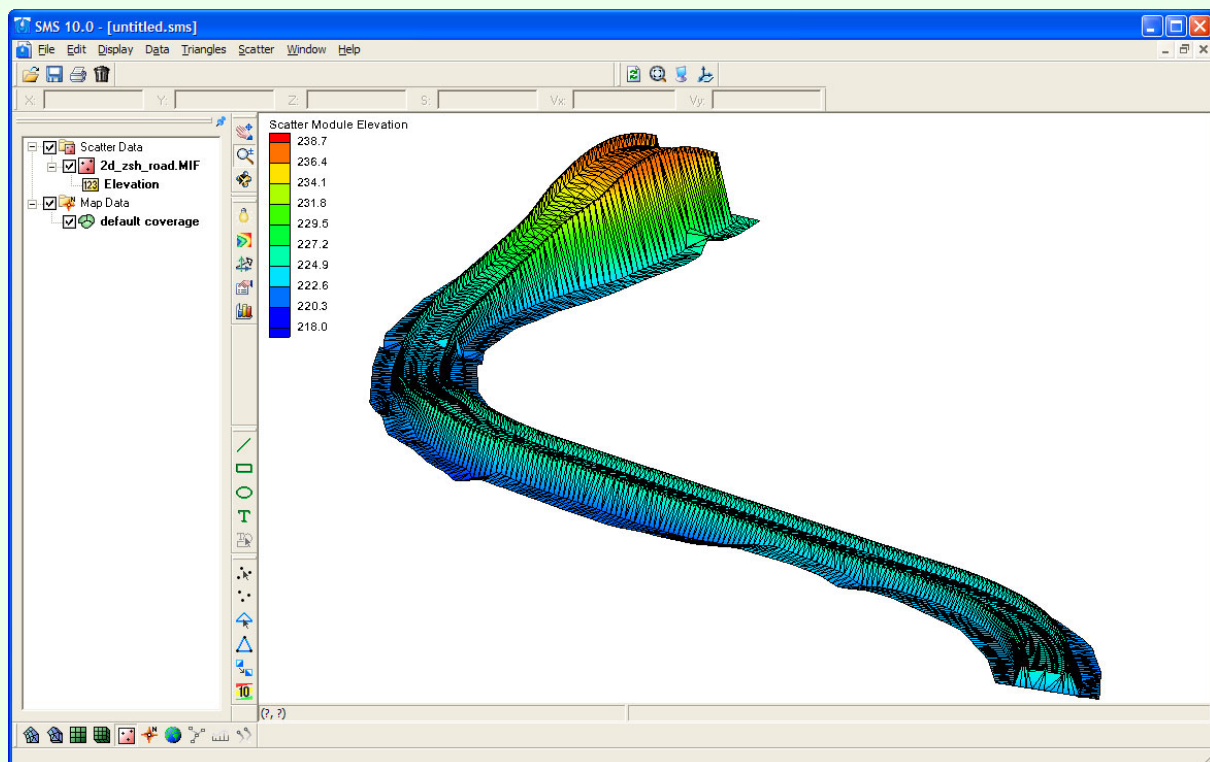
Create TIN Zpts == 2d_ztin_haul_road.mif was used to have TUFLOW create a TIN within the polygon as shown in the image below, and override the DEM Zpt values.



The image below is of a DEM created from the TUFLOW Zpts with the polygon shown in grey. As can be seen the haul road TIN has merged seamlessly with the existing Zpts from the base DEM.



The WRITE TIN option (see [Create TIN Zpts](#)) can be used to write the TUFLOW TIN as a SMS .tin file. This file can be opened in SMS for viewing and editing the TIN. The image below shows the haul road TIN in 3D. If the TIN is modified, [Read TIN Zpts](#) can be used to read the TIN back into TUFLOW.



4.4.8.2 Highway Embankment Removal Example

Figure 4-2 presents another example where a new highway, which exists in the DEM, needed to be removed because the calibration flood events occurred before the highway was built. To remove the highway was a simple matter of digitising a polygon around the highway. All attributes were left as their defaults, and there was no need to specify any elevation points. Either [Create TIN Zpts](#) or [Read GIS Z Shape](#) can be used for this purpose.

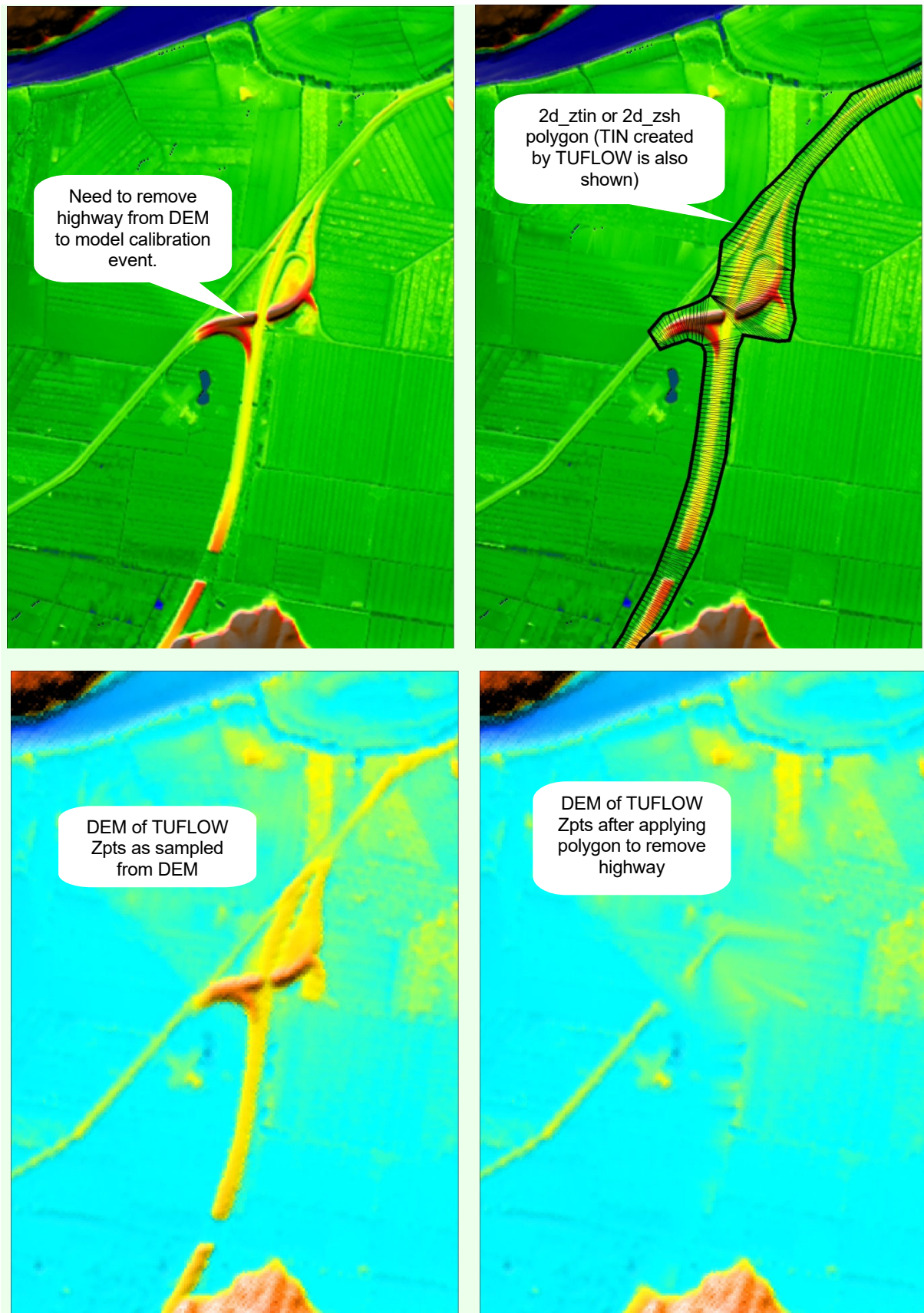


Figure 4-2 Example of Using a 2d_ztin or 2d_zsh Layer to Remove a Highway Embankment

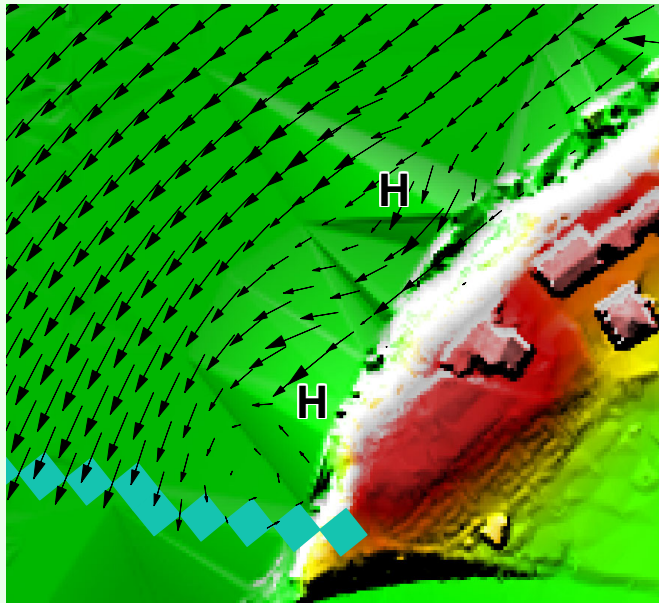
4.4.9 Z Shape (2d_zsh) Layers

[Read GIS Z Shape](#) offers a wide range of options for manipulating and modifying the Zpt values. These include 3D breaklines and TINs. Table 4.8 provides a description of the different attributes.

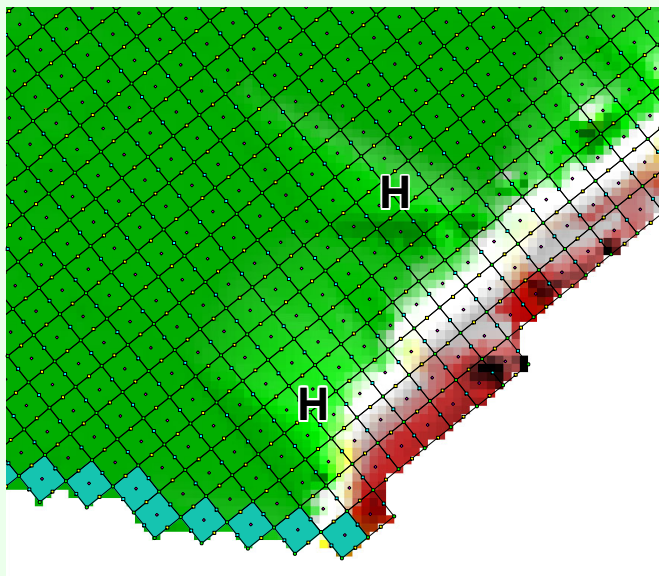
When a mixture of different shapes and shape options occur within the same layer the following protocols are used to control how Zpt values are modified.

- 1 The order in which objects are processed is:
 - (a) **Polygons:** All polygons are triangulated according to the process described for [Create TIN Zpts](#) in Section 4.4.8. The only difference to note is that if a line is to be used for the TIN generation, the Shape_Option attribute for the line must include the keyword “TIN”.
 - (b) **Wide Lines:** Wide lines are lines that have a Shape_Width_or_dMax attribute value greater than 1.5 times the 2D [Cell Size](#). A buffer polygon is created along the line, and all Zpts falling within the buffer polygon are assigned elevations based on a perpendicular intersection with the line. Note that wide lines are processed in the order that they occur in the GIS layer, so if the buffer polygons of two wide lines overlap each other, the latter one prevails. In this situation it would be wise to separate the two lines into two different layers. Buffer polygons can be viewed in the 2d_sh_obj_check.mif layer.
 - (c) **Thin and Thick Lines:** Thin lines have a Shape_Width_or_dMax value of zero and Thick lines a value less than or equal to 1.5 times the 2D [Cell Size](#). For a more detailed description of Thin and Thick lines see [Read GIS Z Line](#). Thin and Thick lines are applied depending on the Shape_Option attribute setting as follows:
 - (i) All ADD lines are applied first.
 - (ii) Then lines without any option (these will modify all Zpts affected by the line).
 - (iii) Followed by GULLY, LOWER or MIN lines.
 - (iv) And finally any RIDGE, RAISE or MAX lines.
- 2 The priority can of course be further controlled by using different layers and controlling when those layers are processed in the .tgc file.
- 3 Note that [Read GIS Z Shape](#) uses a new GULLY line approach (for thin and thick lines) compared with the [Read GIS Z Line](#) approach. The new approach assigns Zpt values that are interpolated using the perpendicular or nearest point along the breakline.

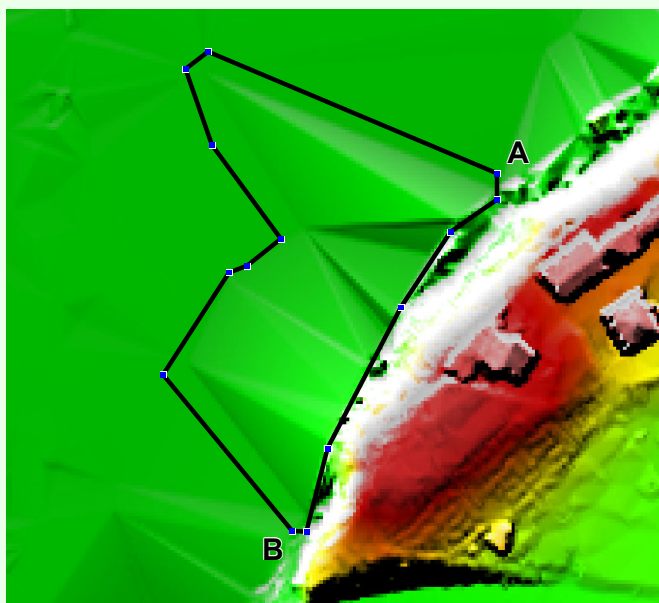
An example of using [Read GIS Z Shape](#) to simply modify Zpt elevations is given below.



The image shows a couple of false ridges indicated by the H letters that were caused by a poor triangulation of the TIN software used to create the DEM. These ridges caused unrealistic flow patterns as shown by the velocity vectors. The blue cells at the bottom are the downstream boundary cells.

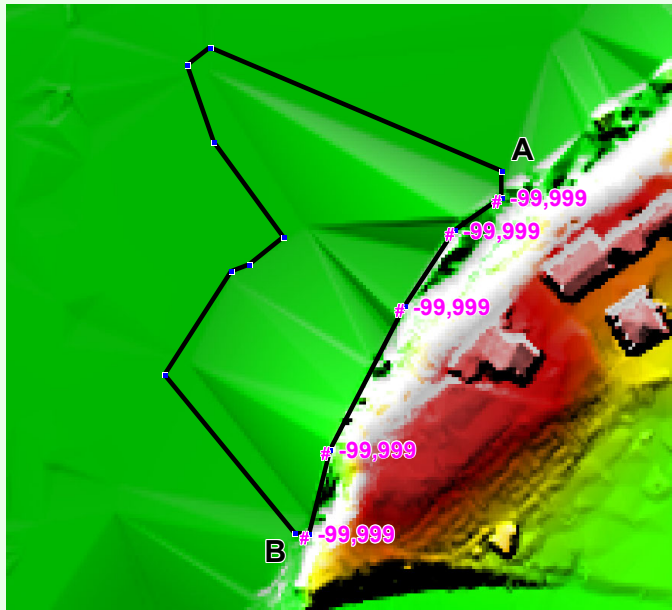


This image is of a DEM produced from the 2d_zpt_check.mif layer (which is shown as the points). This is how TUFLOW sees the DEM data with the false ridges clearly shown.

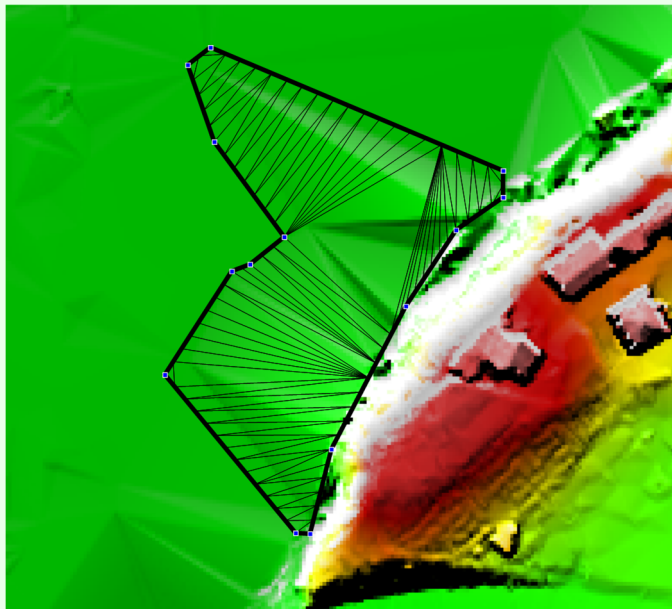


To remove the ridges, import an empty 2d_zsh layer, and digitise a polygon around the ridges as shown. By default (ie. using the default attribute values), the elevations assigned around the perimeter of the polygon are interpolated from the current Zpt values.

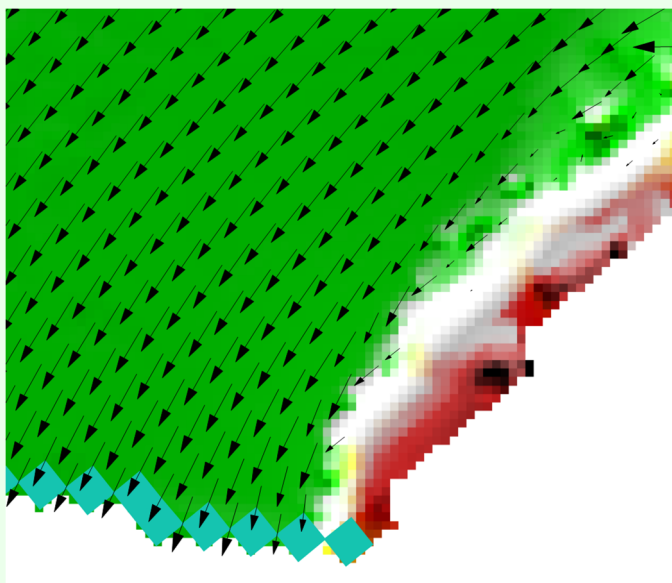
One problem with this approach is that the elevations along the right hand side (ie. along the edge of the floodplain between Locations A and B) are interpolated from the high Zpts along this boundary.



To solve this problem, digitise points either into the 2s_zsh layer or into a 2d_zsh..._pts layer (see Section 4.4.11) that snap to the vertices of the polygon where the high elevations occur. Assign a Z attribute of -99999 to each point as shown in the image. The -99999 indicates to not interpolate an elevation from the existing Zpts. Instead, the elevations at Locations A and B are used to interpolate elevations at vertices where -99999 has been assigned.



Use [Read GIS Z Shape](#) to process the polygon and points and generate the TIN as shown in the image. The TIN can be viewed by importing the 2d_sh_obj_check.mif layer.



This image shows a DEM created from the 2d_zpt_check.mif layer with the above 2d_zsh layer applied. As can be seen, the ridges have been removed and the flow patterns are now realistic.

Table 4.8 Z Shape (2d_zsh) Attribute Descriptions

No.	GIS Attribute	Description	Type
1	Z	Point: Elevation (or change in elevation for ADD option) of the point. Points are only used to assign elevations along lines and polygon perimeters, or for the creation of TINs within polygons. They do not modify Zpts directly. An elevation of -99999 has a special meaning when the point is snapped to a vertex of a polygon. The -99999 indicates to ignore the elevation at that vertex and of any automatically inserted vertices between that vertex and the two neighbouring vertices. Instead the elevations are based on the elevations of the neighbouring vertices. If a neighbouring vertex also has a -99999 point snapped to it, the next vertex is used, and so on. This feature is very useful, as illustrated in the example above. Line: Elevation of the line if no points are snapped to the line, otherwise ignored. Polygon: Used to set the elevation of the polygon perimeter only if NO MERGE option is specified and there are no points snapped to the perimeter. Otherwise ignored.	Float
2	dZ	Point: Change in elevation (Z) value at the point. To leave Z unchanged, set to zero. Useful for easily adjusting the height of the shape object without having to change the original Z value. Line: Change in elevation (Z) value for all vertices/points along the line irrespective of whether there are points snapped to the line or not. To leave elevations unchanged, set to zero. For example, to raise an embankment by 0.2m, set dZ to 0.2. Polygon: Adjusts the height of all points inside or snapped to the perimeter by dZ. Only adjusts the elevations of the polygon perimeter if NO MERGE option is specified, otherwise ignored.	Float

No.	GIS Attribute	Description	Type
3	Shape_Width_or_dMax	Point: Not used. Line (no TIN): Provided “TIN” is not specified for the Shape_Option attribute, this attribute specifies the width (thickness) of the breakline in metres. If equal to zero, only the elevations along the cell sides are adjusted (ie. a thin breakline). If less than or equal to 1.5 times the 2D Cell Size, a line of whole cells is adjusted (ie. a thick line). Otherwise all Zpts within a distance of half Shape_Width_or_dMax are adjusted. Line (TIN): If “TIN” is specified for the Shape_Option attribute, this attribute controls the maximum distance between automatically added intermediate vertices. If set to zero half the 2D domain’s Cell Size is used. If less than zero no intermediate vertices are inserted. Polygon: The maximum distance between intermediate points inserted around the polygon’s perimeter to interpolate elevations from the current Zpts. If equal to zero, half the 2D Cell Size is used. If less than zero no intermediate vertices are inserted. Not used if the NO MERGE option is specified.	Float

No.	GIS Attribute	Description	Type
4	Shape_Options	Point: Not used. Line or Polygon: ADD: Adds the shape's "elevations" to the current Zpts. If ADD is specified, any automatic merging around the region perimeter is ignored (to merge the perimeter with existing Zpts when using the ADD option, specify zero Z and dZ values for the region). Use Build 2008-08-AK or later if using the ADD option due to bug fixes. MAX, RIDGE or RAISE: Only changes a Zpt elevation if the Z Shape elevation at the Zpt is higher. MIN, GULLY or LOWER: Only changes a Zpt elevation if the Z Shape elevation at the Zpt is lower. Line Only: TIN: Indicates the line is only to be used for generation of TINS from polygons (therefore, only sections of TIN lines that fall within the polygon(s) are used). Polygon Only: If none of the options below are specified, the elevations at perimeter vertices that do not have an elevation point snapped to them are merged with the current Zpt values. MERGE ALL: Ignores elevations from any points snapped to the perimeter and merges all perimeter vertices with the current Zpt values. NO MERGE: Does not merge the perimeter elevations with the current Zpt values.	Char

4.4.10 Variable Z Shape (2d_vzsh) Layers

The Zpts can be varied in elevation over time to simulate, for example, breaching of embankments, or even the filling of a river due to a landslide. The 2d_vzsh layer is used to define the final topographic shape at the end of the breach and/or fill period. The first four attributes are the same as for the 2d_zsh layer, plus there are some additional attributes to define how/when the breach commences and for how long. The breach/fill can be triggered by: a specified time; when the water level reaches a specified height at a specified (trigger) location; or the water level difference between two triggers exceeds a specified amount.

Table 4.9 presents the attributes of a 2d_vzsh layer, and an example is discussed and illustrated in the next section.

This new feature as of Build 2008-08-AA, should be used instead of the 2d_bc VG option (unless the rate of change of the erosion/fill is non-linear) as 2d_vzsh layers are much easier to set up and manage.

Table 4.9 Variable Z Shape (2d_vzsh) Attribute Descriptions

No.	GIS Attribute	Description	Type
1	Z	Same as for Table 4.8.	Float
2	dZ	Same as for Table 4.8. The exception is for a thin line, for which the final elevations along the line can be set to dZ metres above the current Zpt values. For example, if a thin line represents a fence that is being collapsed, and the finished height of the collapsed fence is to be 0.2m above existing ground levels, specify a dZ value of 0.2. Note this only applies if the NO MERGE Shape_Option has not been specified.	Float
3	Shape_Width_or_dMax	Same as for Table 4.8.	Float
4	Shape_Options	Point: TRIGGER: Indicates the point is not an elevation point, but a trigger location. The trigger must be given a name using the Trigger_1 attribute. Line or Polygon: Same as for Table 4.8. except the ADD option is not supported. Thin Line: NO MERGE: For thin lines (Shape_Width_or_dMax = 0), the final elevations along the line are as specified. If NO MERGE is not specified for a thin line, the final elevations are set to be the same as the current Zpt values plus the dZ value.	Char(20)
5	Trigger_1	Point: If Shape_Options is set to TRIGGER, enter the name of the trigger location. The name can contain any characters and can include spaces. Otherwise not used. Line or Polygon: To commence the failure at a specified time leave blank. To commence failure based on reaching a water level elsewhere in the model, enter the name of the trigger location. Thin Line: For thin lines there are two special options as follows. Specify DEPTH to have the failure commence once the depth of water adjacent to the cell side exceeds the amount specified for Trigger_Value. Specify DEPTH DIFF to have the failure commence once the difference in water level across the cell side exceeds the amount specified for Trigger_Value.	Char(20)

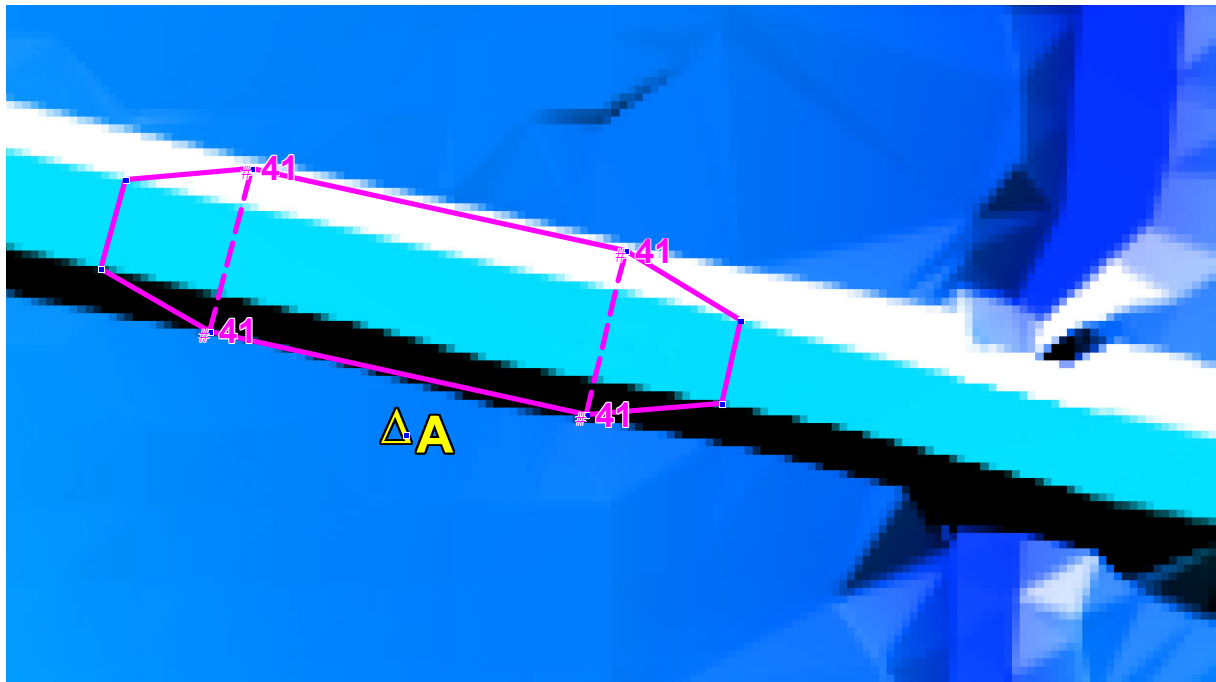
No.	GIS Attribute	Description	Type
6	Trigger_2	Point: Not used. Line or Polygon: The name of a second trigger location (only needed if the breach is to be initiated on a water level difference between two trigger locations).	Char(20)
7	Trigger_Value	Point: Not used. Line or Polygon: If Trigger_1 is blank, the simulation time in hours that the breach is to commence. If Trigger_1 is specified and Trigger_2 is left blank, the water level in metres at Trigger_1 that needs to be reached to trigger the failure. If both Trigger_1 and Trigger_2 are specified, the water level difference in metres between Trigger_1 and Trigger_2 that needs to be exceeded to trigger the failure. The water level difference is taken as the absolute of the difference between Trigger_1 and Trigger_2, so there is no need to specify a negative value. Thin Line: If “DEPTH” is specified for Trigger_1, the depth in metres adjacent to the cell side that needs to be exceeded to trigger the failure at the cell side. If “DEPTH DIFF” is specified for Trigger_1, the water level difference in metres across the cell side that needs to be exceeded to trigger the failure.	Float
8	Period	Point: Not used. Line or Polygon: Time in hours over which the variation in Zpt elevations occurs.	Float

4.4.10.1 Variable Z Shape Example: Breaching of an Embankment

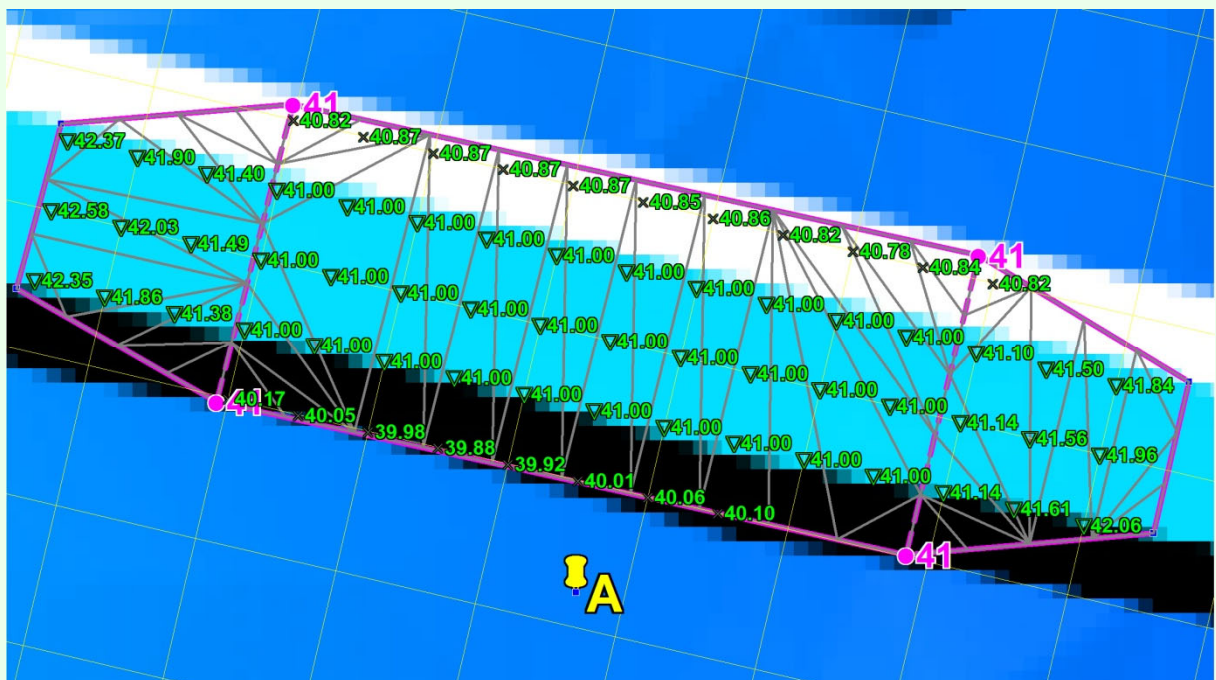
The image below shows an example of a 2d_vzsh layer. The solid magenta line is the polygon, the magenta dashed lines are polylines used to enforce TIN breaklines, and the four magenta points all have an elevation of 41.0m. The Shape_Options attribute for the polygon was set to MIN (this means that the Zpt elevation can only be lowered (ie. eroded), and the dashed lines have Shape_Options of TIN (to indicate that they are to be used for TIN generation, and not for Z lines). The vertices of the polygon that do not have a point snapped to them are assigned an elevation based on the existing Zpt values. The polygon vertices with the points snapped to them are assigned the elevation of the point (in this case, all at 41.0m). The elevation of the dashed lines will be constant at 41m as they are snapped to the 41m points.

The only other object in the layer is the yellow pin point labelled A. This is a trigger point and its only attribute values are: TRIGGER for Shape_Options; and A for Trigger_1. The magenta polygon also has attribute values of: A for Trigger_1 (this indicates that the erosion trigger is based on the water level at Trigger Location A); a Trigger_Value of 42.0 (ie. when the water level at location A reaches

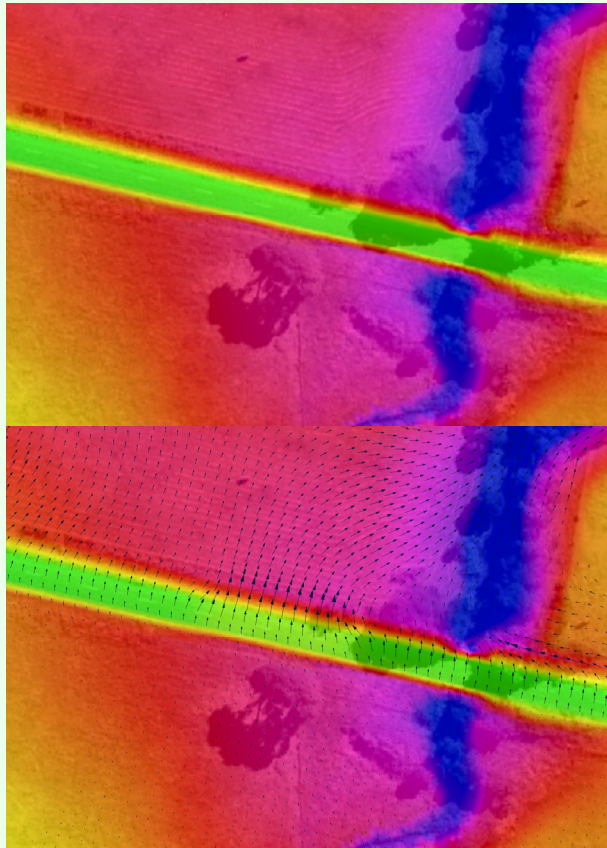
42m, start the erosion); and a Period value of 1.0 indicating that the erosion takes one hour to complete.

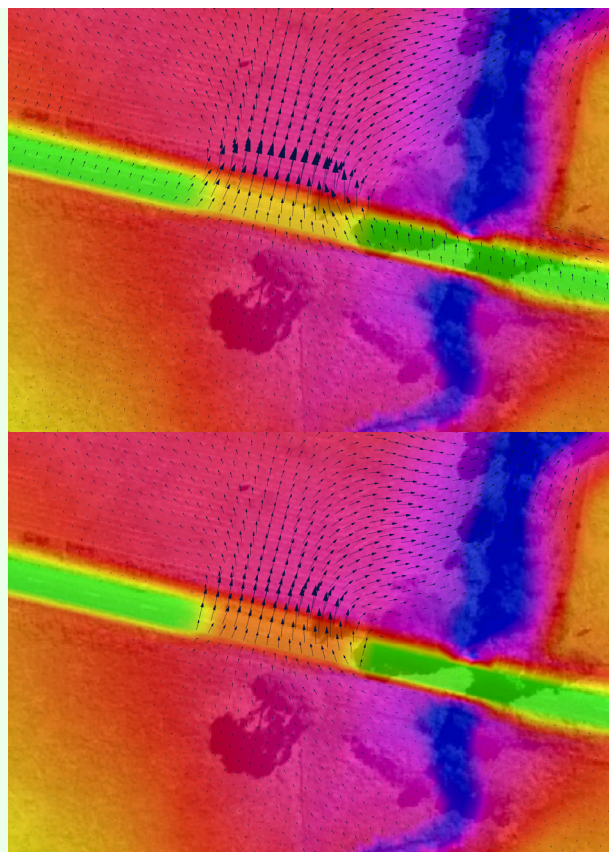


The final eroded Zpt values are based on the TIN created by TUFLOW (see grey triangles in the image below). The central section will be horizontal at 41m sloping up either end to elevations based on the road level. The 2d_vzsh_zpt_check.mif layer is useful to view the Zpts effected by the variable Z shape. This layer is also shown in the image below. The green triangles indicate that the Zpt level is to be eroded, and the crosses indicate no change (this is because of the MIN Shape_Option). The final eroded Zpt values are labelled in the image below. Other useful attributes are also available in this layer.



The images below show the breach modelled using the example above. Each image is in half hour intervals. The colour shading is of the elevations (specify ZH as a [Map Output Data Types](#) to be able to view the changes in ground level over time). The images below are from SMS using the [Map Output Format](#) == SMS HIGH RES option.





4.4.11 Using Multiple Layers and Points Layers

For the 2010-10 release, GIS layers used for [Read GIS Z Line](#), [Read GIS Z HX Line](#), [Create TIN Zpts](#), [Read GIS Z Shape](#), [Read GIS Variable Z Shape](#), [Read GIS FC Shape](#) and [Read GIS Layered FC Shape](#) can be split into more than one layer to more easily manage the variety of data these commands sometimes use.

For example, one layer may contain the elevation points, another the TIN lines and polygons and another the 3D Z lines. This is useful in terms of managing the data, and especially when interrogating and/or viewing the data in GIS. It is also more compatible with the .shp format, which does not allow layers to contain a mixture of different object types (eg. points and lines cannot exist in the same .shp layer).

A maximum of nine (9) layers per command line is allowed, and each layer is separated by a vertical bar (“|”). You can even use a mixture of .mif and .shp files if desired (but probably not recommended!).

For the 2008-08 release a second points only layer was permitted (see section below).

4.4.11.1 Points Only Layers

To minimise the number of attributes, some/all points may optionally be placed into a separate layer with less attributes as discussed below. This keeps the data sets more simple and easier to manage and interrogate.

For the 2008-08 release, a separate points layer must be the second layer in the command line. For the 2010-10 release, more than one separate points layer can be specified and can be in any location except for the first layer. In the 2010-10 release a layer is treated as a separate points layer if it has less attributes than the minimum required for the command.

For most commands there is only one attribute for the points layer, ie. Elevation (or Z) as described in Table 4.10. The exceptions are for [Read GIS FC Shape](#), which requires the first two attributes of Table 4.19, and [Read GIS Layered FC Shape](#), which requires the four attributes in Table 4.22.

The automatic detection of a second points layer using the “_pts” extension is still supported to preserve backward compatibility, however, it’s recommended that this option not be used (it is preferable to enter the filename of the second layer so that it is clear as to which layers are being used). The logic below for the second layer therefore still applies, although not recommended. Using the line below as an example.

```
Read GIS Z Shape == mi\2d_zsh_haul_road.mif
```

- 1 The specified layer, 2d_zsh_haul_road.mif, is opened. This layer may or may not contain elevation points. If any elevation points exist they are used.
- 2 A separate points layer can optionally be used to specify additional points or all of the points. The layer can be specified in one of two ways:

- (a) Entering the pathname of the points layer after the main layer. A “|” must be used to separate the two layers. The points layer must be the second layer specified. For example the example above would look like:

```
Read GIS Z Shape == mi\2d_zsh_haul_road.mif | mi\2d_zsh_haul_road_points.mif
```

- (b) Alternatively, name the points layer the same as the main layer, but with a “_pts” extension. If a layer exists with the “_pts” extension, TUFLOW automatically assumes this layer is associated with the main layer, and includes all points within this layer when applying the above commands. For this example, the layer would be named 2d_zsh_haul_road_pts.mif.
- (c) The first approach (a) above prevails over the second (b) if both apply.
- (d) If neither (a) or (b) apply, TUFLOW assumes there is no separate points layer.

Table 4.10 ..._pts Attribute Descriptions

No.	GIS Attribute	Description	Type
1	Z	Elevation (or change in elevation for ADD option) of the point.	Float

4.4.12 Bed Resistance (Materials)

The bed resistance values for 2D domains are created by using GIS layers containing polygons of different bed resistance. Bed resistance formulation is set to one of Manning's n , Manning's M ($1/n$) or Chezy using [Bed Resistance Values](#) in the .tcf file. The default is Manning's n . Chezy can be specified as a direct value or by bed ripple heights. Manning's n is the only supported option. The other formulations were developed for coastal applications and have not been fully tested on the PC version of TUFLOW. Manning's n values can also be varied with depth.

The recommended approach is to use materials to define how the bed roughness over the model varies. Each material represents a different roughness category. GIS layers of land-use or vegetation often make excellent material layers. Examples of different materials are River In-Bank, River Banks, Pasture, Roads, Buildings, Forest, Mangroves, etc. Each material is assigned a constant Manning's n value, or varying Manning's n with depth.

Prior to Build 2007-07-AA, the bed resistance values were by default spatially located at the cell centres. Values at the cell mid-sides (where the momentum equation, and hence the bed resistance formula, is applied) were based on averaging the Manning's M ($1/n$) value at the two cell centres either side of the mid-side.

As of Build 2007-07-AA, the default is to interrogate the material polygons directly at the 2D cell mid-sides. This produces a higher resolution representation of the bed roughness, and gives improved flow patterns and results especially in urban areas where large and sudden variations in bed roughness occur due to the presence of roads (very slippery) and buildings (major obstruction, ie. very rough).

Build 2007-07-AA also offers two alternatives to direct interrogation of material/roughness values at cell-sides. The cell centre values can still be used with the cell mid-side values being averaged as either the average of the n values or the average of the M values (the latter being the default prior to Build 2007-07-AA). To use these options see [Bed Resistance Cell Sides](#).

The most common approach is to digitise one or more materials layers (2d_mat) and assign Manning's n values to the materials using [Read Materials File](#). This approach allows the easy adjustment of Manning's n values during model calibration.

In creating the base 2d_mat layer, it is good practice to not digitise the most common or the most difficult to digitise material, and use the following commands in the .tgc file (see Section 4.4.1).

- Use [Set Mat](#) to set the most common material to all cells in a 2D domain.
- Use [Read GIS Mat](#) to allocate the remaining material values.

The [Read GIS Mat](#) command may be used as many times thereafter to further modify the materials in parts of a 2D domain.

Note that as of Build 2007-07-AA, the default material value is now zero (previously one). As a material value of zero is not allowed, every cell and cell-side must now be assigned a material value using [Set Mat](#) and [Read GIS Mat](#) in the .tgc file (it is good practice to always set a default materials value using the [Set Mat](#) as the first material command in the .tgc file). For backward compatibility, [Change Zero Material Values to One](#) can be used to set any zero material values to one.

As of Build 2007-07-AA, material values must be within the range 1 to 32,767 (in previous builds a value up to 99,999 could be used).

If using the Chezy formula, a number of commands have been setup to provide backward compatibility. These are [Depth/Ripple Height Factor Limit](#) and [Recalculate Chezy Interval](#).

4.4.12.1 Materials Files

Materials file(s) contain information on a material's roughness and optionally infiltration if using direct rainfall. The file is referenced using [Read Materials File](#) and can be in one of two formats (.tmf and .csv) as described below.

As of the 2009-07 Release the reading of materials files has been reworked, as follows.

- More than one materials file may now be specified by repeat occurrences of [Read Materials File](#).
- Both .tmf and the new .csv formats are supported.
- Any combination of .tmf and .csv files can be used.
- Up to 100 materials in total are allowed.

Prior to the 2009-07 Release, only one .tmf file could be used and the .csv format option is not available.

The .csv format offers greater flexibility for future options, and supports curves of Manning's n versus depth.

4.4.12.2 .tmf Format

The .tmf format is a text file containing Manning's n and other information for different materials (land-uses). The file can contain comments using the “#” and/or “!” delimiters at any location.

The first number is the Mat (Material ID) number, which must be an integer.

The second value is the Manning's n value.

The third and fourth numbers are optional and set the initial and continuing loss rates in mm and mm/h if using [Read GIS RF](#).

The fifth to eighth numbers are also optional and are used to change the Manning's n value with depth. The four values, y_1 , n_1 , y_2 , n_2 , are applied as follows: below y_1 metres deep, a Manning's n value of n_1 is applied; above y_2 , n_2 is applied, and between y_1 and y_2 , the Manning's n value is interpolated between n_1 and n_2 according to [Bed Resistance Depth Interpolation](#). Initial and continuing loss values must be specified in columns 3 and 4 as described above (enter these as values as zero if not using direct rainfall). **Note, if the four values are specified, the Manning's n value in the second column is ignored and not used.**

The material values may also be used to define bed resistance values across 1D XZ cross-sections (see Section 4.6.6.2).

The file format is shown in the examples below. See [Set Mat](#), [Read GIS Mat](#) and [Read RowCol Mat](#) for assigning the material IDs to the 2D domains.

```
! Comments and blank lines are allowed in this file
! First value is the Mat value
! Second is the Manning's n value
! Maximum of 100 different materials
1, 0.03    ! waterways
2, 0.08    ! river banks
11, 0.06   ! grazing land
12, 0.04   ! parks and gardens
13, 0.15   ! sugar cane
14, 0.12   ! natural forest
15, 0.02   ! roads
```

To include the initial loss (mm) and the continuing loss rate (mm/h) optionally enter a third and fourth value as shown below. If an IL is specified, a CL must also be specified otherwise an ERROR occurs. Both can be omitted, in which case, they are both set to zero.

```
1, 0.03    ! waterways
2, 0.08, 20, 2    ! river banks
11, 0.06, 20, 2    ! grazing land
12, 0.04, 5, 1.5   ! parks and gardens
13, 0.15, 10, 2    ! sugar cane
14, 0.12, 30, 2.5  ! natural forest
15, 0.02, 2, 0     ! roads
```

To vary n values with depth (m) using two pairs of values optionally enter values in the fifth to eighth columns as shown below in lines 2 and 3 (Materials 2 and 11) below. IL and CL values must be entered (use zero if not relevant).

```
1, 0.03    ! waterways
2, 0.08, 20, 2, 0.3, 0.15, 0.5, 0.08    ! river banks with long grass 0.3m high
11, 0.06, 20, 2, 0.1, 0.1, 0.15, 0.06    ! grazing land
12, 0.04, 5, 1.5   ! parks and gardens
13, 0.15, 10, 2    ! sugar cane
14, 0.12, 30, 2.5  ! natural forest
15, 0.02, 2, 0     ! roads
```

4.4.12.3 .csv Format (Manning's n vs Depth Curves)

The.csv format is intended to be generated from a database of materials and associated data in a .xls file in a similar manner to BC databases, using TUFLOW Tools.xls macros to export to the .csv format.

Column 1: The Material IDs must be in the first column as shown in Figure 4-3. Integer numbers must still be used for Material IDs, although work has commenced to handle non-numeric Material IDs.

Column 2: The second column contains information on the bed resistance values (usually Manning's n). The options available as of the 2009-07 Release are:

- Single n value (the n value is assumed constant at all water depths).

- Two pairs of depth and n values (see the y_1 , n_1 , y_2 , n_2 option described above for the .tmf format, ie. Columns 5 to 8). The values must be separated by commas.
- A .csv file containing n vs depth values. 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 curve's data. However, you can specify the columns to be used by placing the column label after the .csv filename using a vertical bar "|" to separate the column label from the .csv filename. If only one column label is given, this is assumed to be the depth column and the next column must contain the Manning's n values. 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. To view how the n values vary with depth over time, use the "n" option for [Map Output Data Types](#).

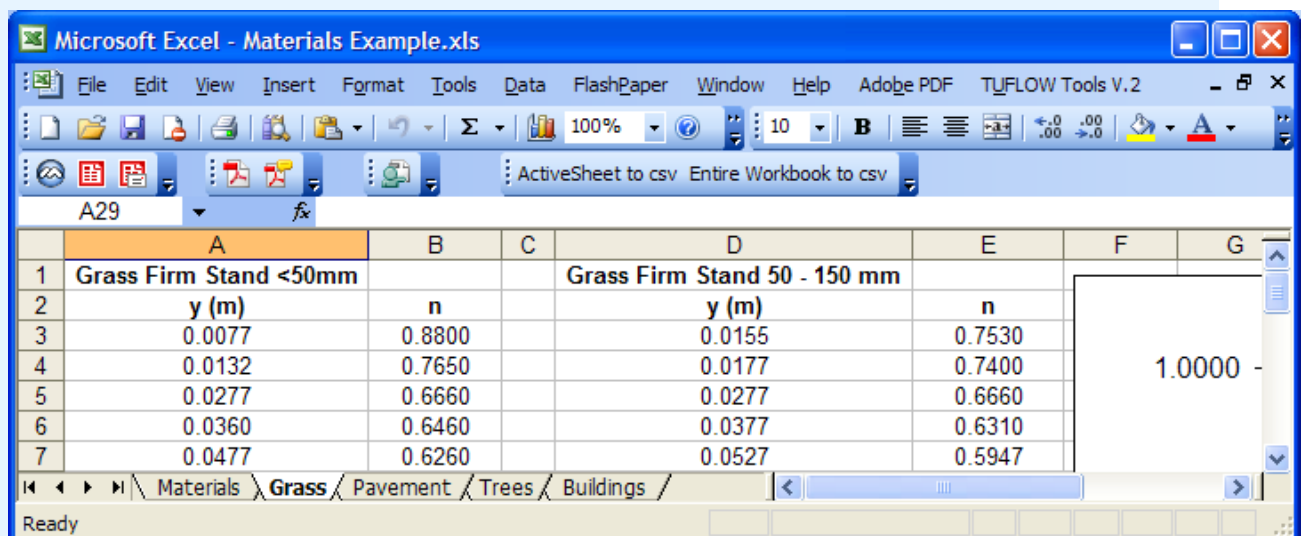
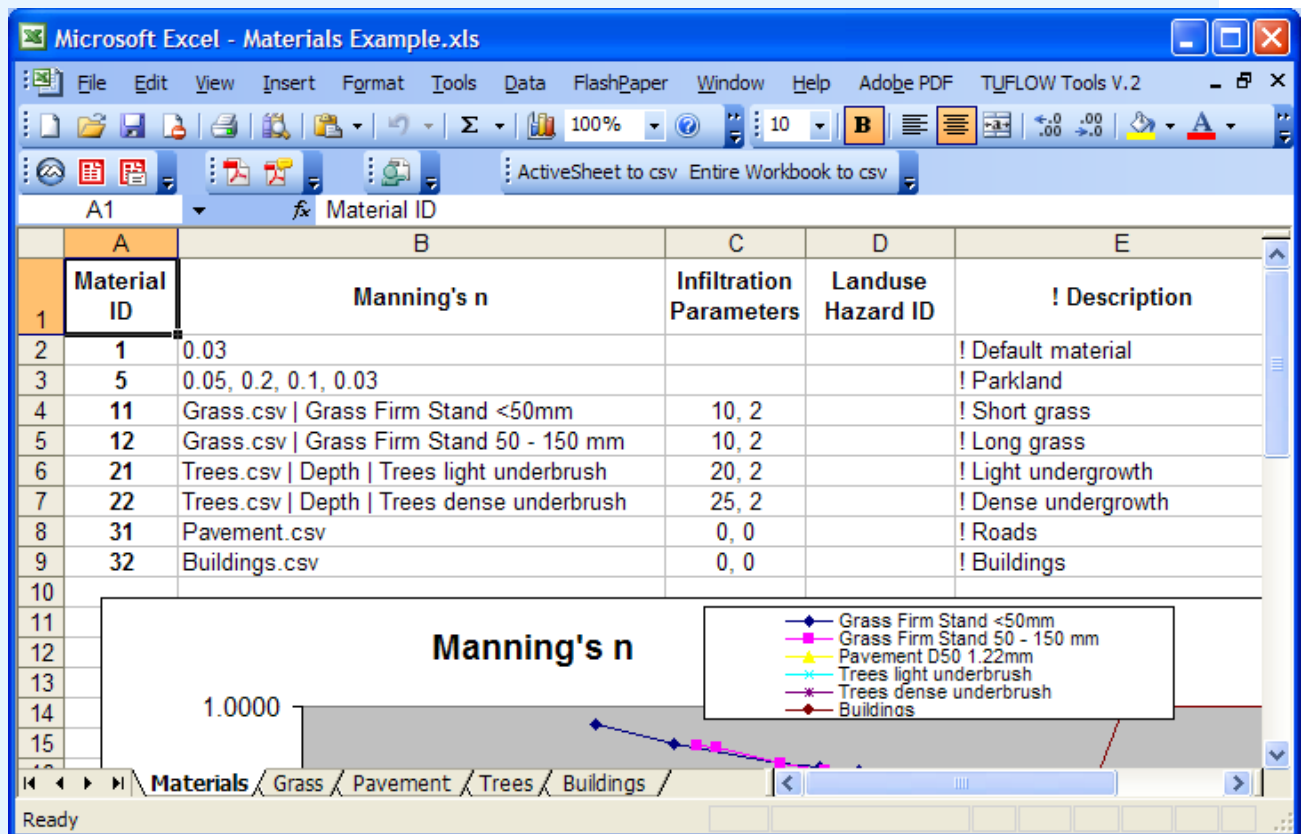
Column 3: The third column of the .csv format is reserved for the infiltration parameters. At present, only the initial loss/continuing loss option is available and is entered as two comma delimited numbers in a similar manner to the third and fourth values in the .tmf format.

Column 4: The fourth column is reserved for a future 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.

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 y_1 , n_1 , y_2 , n_2 values (as per .tmf format approach) and no infiltration parameters.
- Materials 11 and 12 will source n vs y curves from a file called 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 used to create Grass.csv 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.



The figure consists of two screenshots of the Microsoft Excel application window titled 'Materials Example.xls'. The top screenshot shows the 'Trees' sheet with the following data:

	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	0.190	0.360				

The bottom screenshot shows the 'Pavement' sheet with the following data:

	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	0.050524082							

Figure 4-3 Example of Materials .csv File Format

4.4.13 Multiple 2D Domains

Any number of 2D domains of different cell size and orientation can be combined to form one model. The 2D domains can be linked by 1D domains, with 2D/2D linking under development and planned for a future release. For example, a 1D domain of a river system may have several 2D domains embedded to represent several townships where a more detailed analysis is required. The combination of 1D and 2D domains forms one overall model. Direct 2D to 2D linking can be achieved by using the 2d_bc 2D link Type (see Table 4.32 and Table 4.33).

To have access to multiple 2D domains requires having the Multiple 2D Domains Module (visit www.tuflow.com or contact sales@tuflow.com if you do not have this module).

To specify more than one 2D domain use [Start 2D Domain](#) and [End 2D Domain](#) in the .tcf file to start and end blocks of commands applicable for each 2D domain.

The mandatory .tcf commands that occur within a 2D domain block are:

[BC Control File](#)

[Geometry Control File](#)

[Timestep](#)

Optional commands that can be used are:

[Cell Wet/Dry Depth](#)

[Cell Side Wet/Dry Depth](#)

[Set IWL](#)

[Read GIS FC](#)

[Read GIS GLO](#)

[Read GIS IWL](#)

[Read RowCol IWL](#)

[Read GIS LP](#)

[Read GIS PO](#)

[Instability Water Level](#)

Note that specifying one of the above commands outside a Start/End 2D Domain block does not apply that command to all 2D domains. For example, specifying [Cell Wet/Dry Depth](#) outside a block will not set the [Cell Wet/Dry Depth](#) value to all 2D domains, and causes ERROR 2107 to occur.

An example of using 2D domain blocks is given below.

```
# This is an example of a simple .tcf file with multiple 2D domains
MI Projection == ..\model\mi\Projection.mif

Start 2D Domain == East Domain
    BC Control File == ..\model\east domain.tbc
    Geometry Control File == ..\model\east domain.tgc
    Timestep (s) == 10
    Set IWL == 1
End 2D Domain

Start 2D Domain == West Domain
    BC Control File == ..\model\west domain.tbc
    Geometry Control File == ..\model\west domain.tgc
    Timestep (s) == 5
    Set IWL == 1
End 2D Domain

Start Time (h) == 0.
End Time (h) == 12.

Read Materials File == ..\model\n_values.tmf ! .tmf is for TufLOW Materials File
```

4.5 1D Domains (Networks)

1D domains are made up of a network of channels and nodes where:

- Channels represent the conveyance of the flowpaths.
- Nodes represent the junctions of channels and the storage capacity of the network.

There are no constraints on the complexity of the network with any number of channels being able to connect to a single node. Each channel is connected to two nodes; one at the channel's upstream end the other at its downstream end. **As of Build 2002-08-AC, the digitising of nodes is optional.**

Note: There must be at least one (1) channel in at least one of the 1D domains. This can be a dummy channel separate to the rest of the domains.

1D networks are created using one or more GIS layers. As of the 2010-10 release, the original fixed field format entry is no longer supported.

4.5.1 Nodes and Pits (1d_nwk Layers)

Nodes are created in a several ways:

- digitised points in a 1d_nwk layer;
- from a pit in a 1d_nwk layer;
- from a manhole in a 1d_mh layer (see Section 4.5.2), or
- if none of the above, automatically using channel ends.

Note: all points in 1d_nwk and 1d_mh layers are interpreted as being a node. Pits were introduced in Build 2005-05-AN, and extended options were also made available for nodes in this build. Manholes were introduced in the 2010-10 release.

As of Build 2010-10-AB, any number of pits may be specified at the same location.

Pits automatically insert a short channel called a pit channel to transfer water between the 2D domain on the surface and a 1D pipe network underground (see Section 4.5.1.1 below). Two nodes are created: one at ground level for connection to the 2D domain and one at the pipe junction.

Both nodes and pits can be automatically connected to 2D domains - see Section 4.5.1.3.

If a node has not been digitised at the end of a channel a new node is automatically created. The ID of automatic nodes is the first ten characters of the Channel ID with a “.1” or “.2” extension. “.1” is used if the node is at the start of the channel and “.2” if at the end. If more than one channel is connected to the created node, the Channel ID that occurs first alphanumerically is used. The automatic creation of nodes can be switched off using [Create Nodes](#).

Note: The use of the term “node” in this manual refers to both manually digitised nodes and nodes automatically created at the ends of channels where no digitised nodes exist. The term “pit channel” refers to the small channel automatically inserted at a pit node.

Prior to Build 2005-05-AN, the only attribute required for a manually created node was its ID (see Table 4.14). Several other attributes are now optionally available as discussed below and detailed in Table 4.14. It is also now not mandatory to specify an ID.

If an ID is specified it must be unique amongst all nodes, and up to 12 characters in length. It may contain any character except for quotes and commas. As a general rule, spaces and special characters (eg. “\”), should be avoided although they are accepted. A node ID can be used for a channel (ie. a channel and node can have the same ID), but not for another node. The Ignore attribute can be used to ignore a node. It is recommended for the 1d_nwk Type attribute that “Node” is entered to easily distinguish nodes from channels when querying objects in the GIS, or alternatively place the nodes and pits in separate 1d_nwk layers. Prior to Build 2005-05-AC, all other 1d_nwk attributes were not used for nodes.

If using .shp layers, because .shp layers cannot contain points and lines in the same layer, all point objects (nodes, pits and manholes) will have to be in different layer(s) to the channels (which are line objects).

Since Build 2005-05-AN, a number of attributes may now, optionally, be used to more easily create a 1D network. These are described in Table 4.14 and are listed below:

- Assign upstream and downstream inverts at channel ends using the DS_Invert attribute of a node.
- Add additional storage (as a surface area) to a node using the Len_or_ANA (previously referred to as Length) attribute, or automatically create a nodal area table if no node storage is specified.
- Automatically insert a small, zero length channel, called a pit channel, to convey water to/from a 2D overland domain to a 1D pipe network (see Section 4.5.1.1 below).

For specifying node storages see Section 4.6.2.

4.5.1.1 Pits and Pit Channels

As discussed in Section 4.5.1, pit channels are created from 1d_nwk point objects that have the Type attribute (see Table 4.14) set to C, Q, R or W (note, the Q option was added for Build 2008-08-AA).

The pit channel ID is given a “.P” extension based on the ID assigned or specified to the node. The upstream (ground) node ID of the pit channel is given a “.0” extension. The ground elevation is specified in the US_Invert attribute, and the bottom elevation of the pit is specified in the DS_Invert attribute. The Conn_2D (previously Topo_ID) attribute can be used to specify if this pit is to be connected to an overland 2D domain (this saves having to create 2D SX links in a 2d_bc layer).

Pit channels are zero length channels designed to convey water to/from a 2D overland domain to a 1D pipe network. They do not contribute to any storage in the system (noting that the Len_or_ANA attribute is used to specify the surface area or storage of the pit).

If a pit channel cannot be connected to an overland 2D domain (because, for example, it does not fall on an active cell), a WARNING is issued (see Section 7.2.2) and the pit channel will remain unconnected.

As of Build 2010-10-AB, pits do not have to be snapped to a channel end, and can be automatically connected to the nearest node (channel end) by using [Pit Search Distance](#).

In the various 1D output and the 1d_nwk check files, the pit channels that are snapped to a channel end are displayed as a small channel orientated in the north-south axis, not a node (point). The display length of the channel is, by default, set to 10 m, however, this can be changed using [Pit Channel Offset](#). For pits connected using [Pit Search Distance](#), the pit is displayed as extending from their location to the their connected node.

4.5.1.2 Pit Inlet Database

As of Build 2008-08-AA, pit channels can be specified as a Q Type and the flow through the pit is controlled by a depth-discharge curve extracted from the [Pit Inlet Database](#). The database is similar to the [BC Database](#) and is described below.

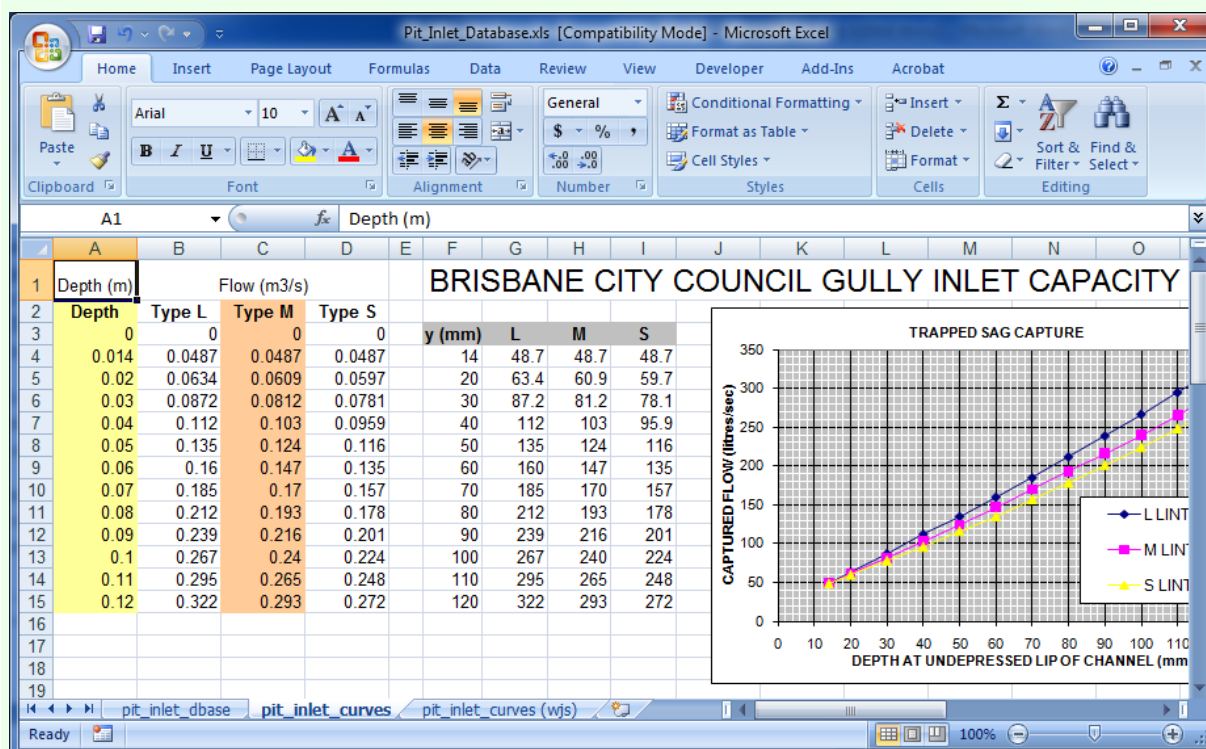
The [Pit Inlet Database](#) is a .csv file usually set up and managed in a Microsoft Excel spreadsheet. The database file sources depth-discharge curves in other .csv files that are also usually set up and managed in the same Excel spreadsheet.

The image below shows an example of a [Pit Inlet Database](#) .csv file. Note that the first column must contain the pit inlet type as referenced by the 1d_nwk Inlet_Type attribute (see Table 4.14). The second column contains the name of the source .csv file that contains the depth-discharge curve. The third and fourth columns are the heading labels of the depth and discharge columns respectively in the source .csv file.

The fifth and sixth columns are the inlet's nominated full flow area in m² and flow width in m. Note that the flow area is only used to output a velocity for the pit channel (ie. it does not have any influence over the hydraulic calculations). The width is used for the automatic selection of 2D cells when connecting to a 2D domain (see Section 4.5.1.3) and for extending the depth discharge curve.

For the example shown below, if the 1d_nwk Inlet_Type attribute is set to "M", TUFLOW searches the first column (Name) until it finds "M" as highlighted in green in the image below. The source .csv file, pit_inlet_curves.csv is opened and the occurrence of the labels "Depth" (shaded yellow) and "Type M" (orange) on the same line in pit_inlet_curves.csv are searched for. The numeric values for these two columns, as shown in the second image below in yellow and orange, are read and applied as the depth-discharge relationship for the Q pit channel.

	A	B	C	D	E	F
1	Name	Source	Depth_Col	Flow_Col	Area (m2)	Width (m)
2	Type L	pit_inlet_curves.csv	Depth	Type L	0.72	4.8
3	Type M	pit_inlet_curves.csv	Depth	Type M	0.54	3.6
4	Type S	pit_inlet_curves.csv	Depth	Type S	0.36	2.4



The depth-discharge curve is applied and extrapolated as follows:

- The first coordinate on the curve must be 0,0, ie. zero flow at zero depth.
- The depth and flow must be in metres and m^3/s .
- If required the curve is extrapolated. An “effective” flow width of the pit inlet is calculated that gives the same flow at the top of the curve as would orifice flow based on the Width value provide in the database .csv file. The orifice flow equation is used to extend the curve indefinitely.
- The depth value used to derive the discharge through the pit channel is taken as the water level in the 2D domain less the pit invert (usually the ground/2D level). If the downstream

water level of the pit channel rises above the pit invert, the depth value used for deriving the pit channel flow is the difference in water level. This provides a smooth transition from upstream controlled flow to drowned flow. If the pit channel starts to be surcharged (ie. flow is reversed), the flow is extracted from the depth-discharge curve using the downstream water level less the upstream water level (pit invert if dry) as the depth value, and is applied as a negative discharge so that flow surcharges onto the 2D domain.

Table 4.11 presents the different Q pit flow regimes output to the .eof file next to the velocity and flow values, and to the _TSF and _TSL output layers (see Sections 7.4.1 and 7.4.4).

Table 4.11 Q Pit Flow Regimes

Regime	Description
N	The depth-discharge point is within the bounds of the Q pit curve provided.
O	The depth is over (exceeds) the highest depth in the Q pit curve and the extrapolated orifice flow equation is being used.
R	Reverse (negative) flow is occurring.

4.5.1.3 Connecting Pits and Nodes to 2D Domains

As of Build 2008-08-AA, both nodes and pits can be automatically connected to 2D domains using the 1d_nwk Conn_2D (previously Topo_ID) attribute (see Table 4.14). Previously only pits could be connected using this attribute, and the connection was always to a single 2D cell.

If Conn_2D is set to “SX”, TUFLOW automatically connects the upstream end of the pit channel or node to the 2D domain. The 2D cell must be active (Code = 1). If no active 2D cells are found from any 2D domain, a WARNING 2122 or 2123 is issued. If there is overlapping 2D domains and more than one 2D cell is active a WARNING 2037 is given.

As of Build 2010-10-AB, [Pit Default 2D Connection](#) can be used to set the global default value for the 1d_nwk Conn_2D attribute.

The default (as of Build 2008-08-AA) is to connect to one or more 2D cells at pit connections up to a limit of ten (previously only one 2D cell would be selected). The number of 2D cells selected is a function of the flow width of the pit inlet (including the effects of the Number_of and Width_or_Dia 1d_nwk attributes), or the total flow width of the channels connected to the node. The total width of the 2D cells selected is always more than the pit inlet width or width of channels. For example, a width of 3.6m on a 2m grid selects two cells. The advantage of selecting more than one 2D cell is to provide enhanced stability at the 1D/2D connection when the 1D flow width exceeds the 2D cell size.

The number of 2D cells connected can be controlled by the user using the 1d_nwk Conn_No attribute, and the approach for selecting the 2nd, 3rd, ... cells can also be controlled using the G and S flags for the Conn_2D attribute as follows.

- The 1d_nwk Conn_No attribute can be used to change the number of 2D cells connected as follows.
 - A positive value adds that number of 2D cells to the automatic number. In the example above, if Conn_No is set to 1, three (2 + 1) 2D cells are connected.
 - If Conn_No is negative, this ignores the automatic approach and fixes the number of 2D cells connected to the absolute value of Conn_No. For example, a value of -1 would only connect the pit or node to one (1) 2D cell irrespective of the width.
- If more than one 2D cell is being connected, there are two approaches available for how the 2D cells are selected.
 - For pits, the default is the G (on Grade) option that selects the 2nd, 3rd, ... cells by traversing to the next highest 2D cell that neighbours the 2D cell(s) already selected (this includes diagonal cells, ie. all 8 cells around a cell are considered). These cells are also automatically lowered to the same height as the first cell so that their ZC value is at or below the pit inlet.
 - For nodes, the default is the S (Sag) option that selects the 2nd, 3rd, ... cells by traversing to the next lowest 2D cell. The ZC values of these cells are not changed as they are already at or below the node invert.
 - The above approaches can be changed using the G (Grade) and S (Sag) flags for the 1d_nwk Conn_2D attribute. The G approach is that adopted by default for pits and S for nodes. To override the default approach, specify G or S as appropriate. For example, if the S approach is preferred for a pit that is draining a depression, specify a S flag (ie. "SXS") for the Conn_2D attribute. Note SX must come first before an optional flag is specified.
 - Note: If [Defaults](#) == PRE 2008-08 is specified, the original approach of only selecting one 2D cell is used, irrespective of the Conn_2D and Conn_No values.

Note that the conventional approach of using 2d_bc SX points or lines can still be used to connect 1D channel ends. This gives the user complete control over which 2D cells are selected and may still be preferred in situations where large 1D structures are being connected to 2D cells.

4.5.2 Manholes (Automatic and 1d_mh Layer)

Manholes are used at culvert junctions to dissipate energy due to:

- Expansion/contraction of flow within the manhole chamber and outlet culverts.
- Change in direction of the culverts (eg. at a bend).
- Change in height and/or invert level of the adjoining culverts.

The presence of a manhole will override the exit loss of any culvert discharging into the manhole and the entrance loss of any culvert taking flow out of the manhole. Therefore the corresponding Entry Loss and Exit Loss attributes of the culverts in the 1d_nwk layer are not used (see the notes on these attributes in Table 4.14 for more information).

In the 2010-10 release there are three different types of manholes:

- “C” for circular chambers.
- “R” for rectangular chambers.
- “J” for culvert junctions without a chamber.

And three different energy loss approaches:

- “NO” no special manhole losses (the standard entry/exit losses of the connected culverts apply).
- “EN” for the Engelund loss approach.
- “FX” for the Fixed loss approach.

Manholes were introduced for the 2010-10 release.

4.5.2.1 Automatically Assigned Manholes

By default, manholes are automatically created at all culvert junctions (nodes) where the following conditions are met.

- There is at least one incoming and one outgoing culvert – a culvert is a C, I or R channel.
- There are no open channels connected (ie. no bridges, weirs or any other channel that is not a culvert).
- Pit channels can be connected, but are not included in any of the calculations for determining manhole energy losses.

Note: Models run using builds prior to the 2010-10 release that have underground culvert networks may experience significantly different results, as the default setting of [Manholes at All Culvert Junctions == ON](#) typically applies greater energy losses compared with using the default settings for prior releases. To not have manholes automatically created use [Manholes at All Culvert Junctions == OFF](#). However, manual specification of energy losses via the outlet culvert’s 1d_nwk Form_loss attribute, or setting the losses as being fixed (use [Structure Losses == FIX](#) or specify the “F” option – see “F” in Table 4.13) maybe required wherever there is a manhole chamber, bend or other reason for energy dissipation.

The type of manhole assigned is set using [Manhole Default Type](#). If [Manhole Default Type](#) is not specified the default is to assign a C, R or J type based on the size and configuration of culverts connected to the manhole (see [Manhole Default Type](#) for more information).

The minimum diameter/width of a manhole chamber is controlled by [Manhole Minimum Dimension](#), and the default side clearance and clearance between culverts is set using [Manhole Default Side Clearance](#).

The default energy loss approach is controlled by [Manhole Default Loss Approach](#).

4.5.2.2 Manually Assigned Manholes (1d_mh Layer)

Manholes are manually assigned to nodes using a 1d_mh layer and [Read GIS Manhole](#). Any number of 1d_mh layers can be specified. If a manhole is repeated either in the same layer or a subsequent layer, the last one processed prevails. The attributes of a 1d_mh layer are described below in Table 4.12.

A manhole in a 1d_mh layer will override any automatically created manhole. As of Build 2010-10-AB, if a manually specified manhole occurs at the same location more than once, the last occurrence will prevail (ie. the last manhole at a node will override any previously specified manholes at the same node).

To remove an automatically created manhole, digitise a point at the node in a 1d_mh layer and set the Loss Approach to “NO” (ie. no manhole energy losses). Note that culvert inlet/outlet losses will apply instead in accordance with Section 4.7.4.1.

Table 4.12 1D Manhole (1d_mh) Attribute Descriptions

No.	GIS Attribute	Description	Type
1	ID	Unique identifier up to 12 characters in length (see ID for 1d_nwk layers in Table 4.14). The ID is used to label the node that is created at the location of the manhole. If left blank, then the ID will be automatically assigned according to the rules for an automatically created node as discussed in Section 4.5.1.	Char(12)
2	Type	“C” for circular shaped chamber. “J” for a junction, but no manhole chamber. “R” for rectangular shaped. If left blank, the global default type set by Manhole Default Type is used.	Char(4)

3	Loss_Approach	“NO” for no manhole loss approach (the standard entry/exit losses of the connected culverts applies). “EN” for the Engelhund loss approach. “FX” for the Fixed loss approach. If left blank, the global default type set by Manhole Default Loss Approach is used.	Char(4)
4	Ignore	If set to true (ie. “T”), the manhole is ignored and makes no contribution to the final network. Otherwise set to “F”.	Logical
5	Invert_Level	The bottom or bed elevation of the manhole. Is also used to set the upstream and downstream inverts of connected culverts, if the culverts have not been assigned an invert. If set to -99999, not used.	Float
6	Flow_Width	The flow width in metres of the manhole (ie. the width of the manhole perpendicular to the dominant direction of flow). For a C manhole this would be the diameter. Used to calculate the manhole flow area and velocity through the manhole.	Float
7	Flow_Length	The flow length in metres of the manhole (ie. the length of the manhole in the dominant direction of flow). For a C manhole this attribute is not used and the diameter (Flow_Width attribute) is used. As of the 2010-10 release, only used to calculate the storage of the node at the manhole.	Float
8	ANA	Additional storage surface area in m ² for the node at the manhole. Usually set to zero.	Float
9	K_fixed	Fixed component of the calculated manhole loss coefficient. If using the Fixed approach, this is the only coefficient applied.	Float
10	Km	Manhole exit coefficient used by the Engelhund approach. If set to zero (0), for C manholes the Manhole Default C Exit Coefficient is applied, and for R manholes Manhole Default R Exit Coefficient is used.	Float
11	K_Bend_Max	The upper limit of the combination of K_{θ} and K_{drop} in the Engelhund approach. If set to zero (0), Manhole K Maximum Bend/Drop is applied.	Float
12	C_reserved	Reserved for future use – leave blank.	Char(12)
13	N1_reserved	Reserved for future use – leave as zero (0).	Float
14	N2_reserved	Reserved for future use – leave as zero (0).	Float
14	N3_reserved	Reserved for future use – leave as zero (0).	Float

4.5.2.3 Digitising Culverts Connected to Manholes

Whether a culvert is an inlet or outlet culvert at a manhole during the simulation is based on the flow direction in the culvert, not the digitised direction. Therefore, if reverse flow occurs this is taken into account in determining the loss coefficients to be applied. However, the pre-processing of the model assumes that the culverts have been digitised in the predominant direction of flow. A variety of checks are made such as whether both an inlet and an outlet culvert are connected to a manhole, and ERRORS or WARNINGS and CHECKS are issued if there is an incompatibility. Therefore, it is strongly recommended that the culverts are digitised consistently in the downstream direction.

4.5.2.4 Fixed Manhole Loss Approach

The Fixed loss approach applies the `1d_mh K_fixed` coefficient as an energy loss on the outlet culvert(s). This is in accordance with publications that quote K values for different manholes based on the velocity in the outlet culvert.

If `K_fixed` is 0.5 or less the energy loss is applied as an inlet loss to the outlet culvert. If `K_fixed` exceeds 0.5, 0.5 is applied as an inlet loss, and the remainder is applied as a form or bend loss. The inlet loss component is shown as the first value in the `_TSL` output layer, and any remainder as the second (middle) value. Any `Form_Loss` value assigned to the outlet culvert via the `1d_nwk` layer is also applied and will be included in the `_TSL` middle value.

By default, `K_fixed` is set to zero, therefore, to assign a `K_fixed` value to any manhole using the Fixed loss approach will require using a `1d_mh` layer.

Note that `K_fixed` is also used by the Engelhund loss approach as an additional (calibration) energy loss if required.

4.5.2.5 Engelhund Manhole Loss Approach

The Engelhund energy loss approach at manholes is based on the work of Engelhund, and used by the MOUSE software (DHI, 2009). The approach adopted has been further adapted and enhanced through advice from, and discussions with, staff from Technical Services Branch of the Gold Coast City Council, Queensland, Australia.

The Engelhund approach provides an automatic method for determining the following energy loss coefficients. The coefficients calculated and their equations are presented below. Of note is that the coefficients are recalculated every timestep, and therefore vary depending on the flow distribution between inlet and outlet culverts and the depth of water within the manhole. The values calculated are output to the `_TSL` layer so they can be reviewed/checked over time.

- K_{entry} covers the expansion of flow within the manhole at the outlet of an inlet culvert. The coefficient is applied as the exit loss of the inlet culvert, and its value over time is shown as the third value for inlet culvert in the `_TSL` output layer.
- K_{θ} represents the losses due to a change in direction (ie. a bend) between inlet and outlet culverts.
- K_{drop} is the loss coefficient due to a change in invert level and culvert height between inlet and outlet culverts.

- K_{Θ} and K_{drop} are added and applied as an energy loss for each outlet culvert. The value calculated is the middle value for outlet culverts in the _TSL output layer. Any Form_Loss value assigned to the outlet culvert via the 1d_nwk layer is also applied and will also be included in the _TSL middle value.
- K_{exit} covers the contraction from the manhole and re-expansion of flow within the entrance of an outlet culvert. It is applied as an entrance loss of the outlet culvert, and is the first value for outlet culverts in the _TSL output layer.

The angle, Θ , between inlet and outlet culverts used to calculate K_{Θ} is based on the digitised lines of the culverts. For the inlet culvert the last two vertices of the line are used and for the outlet culvert the first two vertices. If the culvert line is digitised so that it has an unrepresentative first/last segment angle (this may occur where two parallel culverts have been digitised apart to make it easier to visualise), a 1d_nwk connector (X Type) can be used to connect the start/end of the culvert to the node, so that the first/last segment is a correct representation of the culvert angle relative to the other culverts.

The default value for K_m is dependent on whether the manhole is of a C or R type, and is set using [Manhole Default C Exit Coefficient](#) or [Manhole Default R Exit Coefficient](#).

The equations used for the Engelhund loss approach are provided below.

Q_p = Flow in culvert

Q_{im} = Total flow in to manhole

Q_{om} = Total flow out of manhole

y_i = Height of inlet culvert

y_o = Height of outlet culvert

h_i = Inlet culvert invert

h_o = Outlet culvert invert

θ = Angle in degrees of inlet culvert relative to outlet culvert

($\theta = 0^\circ$ when the culverts are in line, $\theta = 90^\circ$ when the outlet culvert is at right angles)

Q_p = Flow in outlet culvert

W_m = Flow width in manhole = 1d_mhwidth attribute

y_m = Depth of water in manhole

A_m = Flow area in manhole

A'_m = Effective flow area in manhole

A_p = Flow area of culvert

K_m = Km 1d_mh attribute

K_b = Bend loss coefficient (1d_nwkForm_Loss attribute)

K_f = Fixed loss (1d_mhK_Fixed attribute)

K_{Bend_Max} = Upper limit to sum of K_θ and K_{drop} (1d_mhK_Bend_Max attribute)

$$V_m = \frac{Q_{om}}{A_m}$$

$$Q_f = \min\left(\frac{Q_p}{Q_{om}}, 1.0\right)$$

$$A'_m = \frac{W_m y_m Q_{om}}{Q_{im}}$$

$$K_{entry} = \left[1 - \min\left(\frac{V_m}{V_p}, 1\right)\right]^2 \quad (\text{applied as an exit loss on inlet culvert})$$

$$K_\theta = \sum_{\text{inlet pipes}} \left[Q_f \min\left(\frac{\theta^2}{90^2}, 4\right) \right]$$

$$K_{drop} = \sum_{\text{inlet pipes}} \left[\min\left(\max\left(\frac{Q_f(h_o - h_i)(h_o + y_o - h_i - y_i)}{y_o y_i}, 0\right), 2\right) \right]$$

$$K_{exit} = K_m \left(1 - \min\left(\frac{A_p}{A'_m}, 1\right)\right) \quad (\text{applied as inlet loss on outlet culvert})$$

$$K_{outlet\ pipe} = \min(K_\theta + K_{drop}, K_{Bend\_Max}) + K_b + K_f \quad (\text{applied as loss using outlet culvert velocity})$$

4.5.3 Channels

A channel is defined by a length, a Manning's n value, a table of hydraulic properties versus elevation and other parameters depending on the type of channel. Channels can also represent hydraulic structures such as bridges, culverts, weirs, etc. Section 4.6.1 describes the options for defining the hydraulic properties table.

Channels are specified as lines or polylines in a 1d_nwk GIS layer. To connect channels the ends of the channels must be snapped. (Note: As of Build 2002-08-AC, it is not a requirement that both ends of the channel must be snapped to a node – see [Create Nodes](#)). The channels and any digitised nodes may be in the same GIS layer or other layer(s).

Subsequent 1d_nwk layers are used to modify the network at individual objects. For example, if a culvert is to be upgraded in size, then rather than making a copy of the whole 1d_nwk layer, select the culvert channel, save it as another 1d_nwk layer and modify the channel to represent the upgraded culvert. Use [Read GIS Network](#) twice to first read in the base 1d_nwk layer, then the 1d_nwk layer with the single channel representing the upgraded culvert. Provided the channel has the same ID and is snapped to the same nodes, it will override the original culvert channel. Using this approach minimises data duplication and, if executed logically and in a well documented manner, is a very effective approach to modelling.

The attributes required (see Table 4.14) depend on the channel type (see Table 4.13).

Channel flow direction is positive in the direction the line/polyline is digitised. This is visualised in the GIS using a line style that has arrows or other symbolism indicating the line direction.

If a channel has a very steep gradient, critical flow problems may be avoided by specifying that the section properties be calculated from the conditions at the upstream end of the channel. “S” channels test for the occurrence of upstream controlled flow and automatically switch between the two regimes, and is the preferred type for all open channels (the other open channel types are kept for backward compatibility).

The water level in a node is not permitted to fall below the nodes bed level, so if an adjoining channel has an effective invert below this level a permanent phantom flow would flow from the empty node. To prevent this happening the input geometry is checked and any occurrences are reported as an ERROR.

Table 4.13 1D Channel and Node Types

Channel/Node	Type	Description
Primary Channel (Line) or Node (Point) Types		
Normal	(leave blank)	Point: No special node type. The node attributes can be used for various purposes as discussed in Table 4.14. Line: A normal flow channel defined by its length, bed resistance and hydraulic properties. The channel can wet and dry, however, for overbank areas (eg. tidal flats or floodplains) gradient (G) channels should be used. For steep channels that may experience supercritical flow, use S channels. Note: For open channels it is recommended to use the S Type.
Bridge	B	Point: Not used. Line: A bridge structure. See Section 4.7.4.2.
Circular Culvert	C	Point: Circular (in the vertical) pit channel inlet. Line: A pipe or circular culvert. See Section 4.7.4.3.
-	F	Reserved (do not use).
Gradient	G	Point: Not used. Line: Similar to a Normal channel, except when the water level at one end of the channel falls below the channel bed, the channel invokes a free-overfall algorithm that keeps water flowing without using negative depths. The algorithm takes into account both the channel's bed resistance and upstream controlled weir flow at the downstream end. Gradient channels are designed for overbank areas such as tidal flats and floodplains. The upstream and downstream bed invert attributes must be specified to define the slope of the channel.
Irregular Culvert	I	Point: Not used. Line: An irregular shaped culvert. See Section 4.7.4.3.
Matrix Flow Channel	M	Point: Not used. Line: User defined flow channel using a flow matrix. See Section 4.7.4.7.
Depth-Discharge Pit Inlet	Q	Point: A pit channel whose flow is defined by a depth-discharge curve from a database of curves (see Section 4.5.1.2). Line: Not used.
Rectangular Culvert	R	Point: A rectangular (in the vertical) pit channel inlet. Line: A box or rectangular culvert. See Section 4.7.4.3.

Channel/Node	Type	Description
Open Channel	S	Point: Not used. Line: Similar to Normal (Blank) and G channels, except switches into upstream controlled, friction only mode (ie. no inertia terms) for higher Froude numbers (see Froude Check). This allows steep flow regimes such as super-critical flow to be represented. See also Froude Depth Adjustment. This is the preferred 1D open channel as it incorporates all flow regimes, therefore, use this channel in preference to Normal (Blank) and G channels. Upstream and downstream bed invert attributes must be specified to define the slope of the channel, or the inverts can be taken from the channel's cross-sections by specifying -99999 for the inverts. Note: This feature was introduced in Build 2002-08-AC.
Weir	W	Point: A weir (in the vertical) pit channel inlet. Line: A weir structure. See Section 4.7.4.4. For a variable geometry weir see Section 4.7.4.8.
Connector	X	Point: Not used. Line: Connects the end of one channel to another. This is particularly useful for connecting a side tributary or pipe into the main flowpath. It also allows a different end cross-section or WLL to be specified for the side channel, rather than using the end cross-section on the main channel. The direction of the connector line is important. The line must start at the side channel and end at the main channel. If two or more connectors are used at the same location (ie. to connect two or more side channels to a main channel) their ends must all snap to the same main channel.
Additional/Optional Channel (Line) Flags		
Adjust Structure Losses	A	Uses the equations and methodology in Section 4.7.4.1 to adjust the inlet and outlet losses of a culvert or bridge channel according to the approach and departure velocities. This flag overrides Structure Losses if set to FIX. For example, to adjust the losses for a rectangular culvert specify a Type attribute of "RA".
Downstream Controlled	D	For culverts, limits the flow regimes to the downstream controlled ones (see Table 4.23), unless it is a zero length channel (ie. channel length less than 0.01m).
Fix Structure Losses	F	Do not adjust the inlet and outlet losses of a culvert or bridge channel according to the approach and departure velocities. This flag overrides the Structure Losses setting if set to ADJUST (the default). See Section 4.7.4.1. For example, to fix the losses for a circular culvert specify a Type attribute of "CF".

Channel/Node	Type	Description
Non-inertial channel	N	Normal and gradient channels can be specified as non-inertial by including a “N” in the Type attribute. A non-inertial channel has the inertia term suppressed from the momentum equation. Note: Prior to Build 2002-08-AC this flag was “S”. Any models with channels using this feature will have to have “S” flags changed to “N”.
Uni-directional (all channels)	U	Any channel can be defined as uni-directional by including a “U” in the Type attribute. Water will only flow in the positive direction of the channel.
Variable Geometry	V	Normal and gradient channel cross-sections can vary over time by using a variable channel definition. Include a “V” in the Type attribute and see Section 4.7.4.5 for more details.
Weir over the Top	W	If a “W” is specified in conjunction with a B, C or R channel (eg. BW, CW or RW), a weir channel is automatically inserted to represent the flow overtopping the structure. This saves having to digitise the weir separately. To use this option requires adding the 10 optional attributes to the 1d_nwk layer as detailed in Table 4.15. Some of these attributes are used to specify the weir parameters. Available at Build 2005-05-AN.

4.5.4 1d_nwk Attributes

Table 4.14 presents the attributes for 1d_nwk layers. Note the names of these attributes have been changed for Build 2008-08-AA to reflect new features and changes. The name of the attribute is immaterial as far as TUFLOW is concerned as TUFLOW only requires that attributes (of any layer) are in the correct order and of the correct type (ie. Char(), Float, Integer, etc). Like all TUFLOW layers, the attributes can be named as the modeller wishes, so older layers that do not have the same attribute names as documented in this manual will still work fine.

Table 4.14 1D Model Network (1d_nwk) Attribute Descriptions

No.	GIS Attribute	Description	Type
1	ID	Unique identifier up to 12 characters in length. It may contain any character except for quotes and commas, and cannot be blank. As a general rule, spaces and special characters (eg. “\”) should be avoided, although they are accepted. The same ID can be used for a channel and a node, but no two nodes and no two channels can have the same ID. Since Build 2005-05-AN, digitised nodes can have their ID left blank and TUFLOW will assign an ID. Note: Prior to Build 2002-06-AD, ID was a positive integer number. 1d_nwk layers with ID as an integer do <u>not</u> have to be changed to a character field, unless non-integer IDs are to be used.	Char(12)
2	Type (previously Channel_Type)	Node or Pit: Used to specify a Pit Channel as one of C, Q, R or W as per Table 4.13. Not used for Nodes although it can be recommended to type in “Node” for easy identification. Channel: The channel type as specified using the flags in Table 4.13. For X (connectors), no other attributes are required.	Char(4)
3	Ignore	If set to true (ie. “T”), the node or channel is ignored and makes no contribution to the final network. Otherwise set to “F”.	Logical
4	UCS (Use Channel Storage at nodes. Previously Use_Chan_Storage_at _Node)	Node or Pit: Not used. Channel: If left blank or set to Yes (“Y” or “y”) or True (“T” or “t”), the storage based on the width of the channel over half the channel length is assigned to both of the two nodes connected to the channel. If set to No (“N” or “n”) or False (“F” or “f”), the channel width does not contribute to the node’s storage. See Section 4.6.2.2 for further discussion. As of Build 2008-08-AA, this attribute has been	Char(1)

No.	GIS Attribute	Description	Type
		changed from a logical to a char(1). If left blank, the MapInfo default for a character attribute, it is treated as Yes or True. Note that if using the old format of a logical attribute the MapInfo default of "F" (False) has the opposite effect. As of Build 2008-08-AA, this attribute is also now correctly shown in the 1d_nwk_check layer.	
5	Len_or_ANA	Node or Pit: Since Build 2005-05-AN, adds the value specified as additional nodal area (surface area in m²). If no nodal area data exists for the node, either by using the UCS attribute above, or using a NA table, TUFLOW automatically creates an NA table of constant surface area. As of Build 2007-07-AA, if a negative value is specified, this value is used as a multiplier of the node storage. For example, a value of -1.5 increases the nodal storage table (NA table) by 50% (or to reduce storage use a value between -1 and 0). Increasing storage can be useful to stabilise problematic 1D nodes, provided that the added storage does not adversely distort the results. This multiplication is applied after any affect of Minimum Channel Storage Length. Minimum NA is applied after the multiplication. Prior to Build 2005-05-AN, not used. Blank, C, G, I, R, S Channel Type: The length of the channel in meters. If the length is less than zero, except for the special values below, the length of the line/polyline is used. Note, not used to specify the length of a pit channel (which is assumed to have zero length). If Length is -99999., the length from the MIKE 11 link channel is used. B, W Channel Type: Only used in determining nodal storages if the UCS attribute is set to "Y" or "T". Not used in conveyance calculations.	Float
6	n_or_n_F (previously Manning_n)	Node or Pit: Not used. Blank, C, G, I, R, S Channel Type: The Manning's n value of the channel. If using materials to define the bed resistance from XZ tables (see Section 4.6.3), n_or_n_F should be set to one (1) as it becomes a multiplication factor of the materials' Manning's n values. It may be adjusted as part of the calibration process. B, W Channel Type: Not used.	Float

No.	GIS Attribute	Description	Type
7	US_Invert (previously Upstream_Invert)	Node or Pit: Since Build 2005-05-AN, used to specify the ground elevation of the pit. This is used to set the upstream and downstream elevation of the pit channel. If no pit channel is specified, it is not used. If set to -99999, the ground elevation is set to the ZC elevation of the 2D cell that the pit falls within (provided there is a 2D SX connection – see Conn_2D attribute below). Note: as of Build 2008-08-AA, if Conn_2D is set to “SXL”, US_Invert is used as the amount by which to lower the 2D cell and the pit channel invert is set to this level. Prior to Build 2005-05-AN, not used. C, G, I, R, S Channel Type: The upstream bed or invert elevation of the channel in meters. For G and S channels, if -99999 is specified, the bed of the channel cross-section is used. Since Build 2005-05-AN, if a manually created node exists at the upstream end of the channel, and -99999 is specified, the upstream invert is set to the downstream invert specified for the node (or pit), provided this value is greater than -99999. As of Build 2008-08-AA, if a culvert (C, I or R channel) invert has a value of -99999 (after any application of node/pit DS_Invert values), the invert is interpolated by searching upstream and downstream for the nearest specified inverts, and the invert is linearly interpolated. If Defaults = PRE 2008-08 is specified, the invert interpolation is switched off. Interpolate Culvert Inverts can also be used to switch this feature ON or OFF. W Channel Type: As of Build 2006-03-AB, sets the weir invert (see Apply All Inverts). Note that the invert is taken as the maximum of the US_Invert and the DS_Invert attributes. Use -99999 to use the bed of the cross-section as the invert. Prior to Build 2005-05-AN, attribute not used. If there is no cross-section specified, the maximum of US_Invert and DS_Invert is used in conjunction with the Diameter_or_Width attribute to define a rectangular section 5m high. Prior to Build 2003-03-AE, the automatic height given to a weir was 100m – if no cross-section is specified). This was changed to 5m so that automatic generation of node storage areas from channel widths were within a more realistic range of elevations. Use Depth Limit Factor to allow water levels to exceed the 5m range if required. Blank, B Channel Type: As of Build 2006-03-AB, sets the upstream and downstream inverts (see Apply All Inverts). Note that the invert is taken as the	Float

No.	GIS Attribute	Description	Type
		maximum of the US_Invert and the DS_Invert attributes. Use -99999 to use the bed of the cross-section as the invert. Prior to Build 2005-05-AN, attribute not used. Note: If upgrading a model used prior to Build 2006-03-AB, any inverts for B, Blank and W channels need to be set to -99999 to ensure the inverts remain unchanged. Otherwise, the inverts for these channels are likely to be set to zero, as this is the default value set by MapInfo for the invert attributes.	
8	DS_Invert (previously Downstream_Invert)	Node or Pit: Since Build 2005-05-AN, the bottom elevation of the pit. Can also be used to set the upstream and downstream inverts of connected channels – see discussion for US_Invert for channels above. If set to -99999, not used. Prior to Build 2005-05-AN, not used. Channel: Sets the downstream invert of the channel using the same rules as for described for the US_Invert attribute above.	Float
9	Form_Loss (previously Form_or_Bend_Loss)	Node or Pit: The Form_Loss value is applied as an energy loss to all outgoing culverts (ie. culverts that are digitised as leaving the pit/node), so the Form_Loss value should be based on the outgoing pipe velocity (eg. the K values). Note, it may be necessary to zero or reduce the Entry_Loss and/or Exit_Loss attributes of culverts connected to the pit/node so that duplication of losses does not occur. This loss coefficient is not adjusted according to the approach and departure velocities as documented in Section 4.7.4.1. Prior to Build 2008-08-AA, not used. Blank, G, S Channel Type: Additional form losses (factor of dynamic head) due to bends, bridge piers, etc. Preferable to use instead of increasing Manning's n. For S channels, this only applies when not in upstream controlled friction mode. B Channel Type: Since Build 2005-05-AN, adds the value specified to the loss coefficients in the BG table. Note, the component this value adds to the loss coefficient is not subject to adjustment when using Structure Losses == ADJUST. If no BG table is specified, and the value is greater than zero, TUFLOW automatically generates a BG table of constant loss coefficient up until the bridge deck (ie. the top of the cross-section). This value is not subject to adjustment when using Structure Losses == ADJUST. Above the underside of the bridge deck (the top of	Float

No.	GIS Attribute	Description	Type
		the cross-section) a value of 1.56 is used. The 1.56 less the Form_or_Bend_Loss value is subject to adjustment when using Structure Losses == ADJUST. Prior to Build 2005-05-AN, not used and should be set to zero. C, I, R Channel Type: Since Build 2005-05-AN, an additional dynamic head loss coefficient that is applied when the culvert flow is not critical at the inlet. Note, this loss coefficient is not subject to adjustment when using Structure Losses == ADJUST, and is ideally used to model additional energy losses such as pit and bend losses. Prior to Build 2005-05-AN, not used. W Channel Type: Not used.	
10	pBlockage (Build 2003-06-AD) Divergence (Prior to Build 2003-06-AD)	Node or Pit: Not used. This field is reserved for future builds for blocking pit channels. Blank, G, S Channel Type: After Build 2003-06-AD, not used. Prior to Build 2003-06-AD, the channel width divergence factor. Rarely used and recommended to be set to zero. C, R Channel Type: After Build 2003-06-AD, the % blockage (for 10%, enter 10). For R culverts, the culvert width is reduced by the % Blockage, while for C culverts the pipe diameter is reduced by the square root of the % Blockage. Prior to Build 2003-06-AD, not used. B, I, W Channel Type: Not used. Reserved for future builds to fully or partially block B and W channels. The 1d_tab Skew attribute can be used to partially block cross-sections of these channels - see Table 4.17.	Float
11	Inlet_Type (previously Branch)	Node or Pit: For Q pit channels, the name of a pit inlet type in the Pit Inlet Database (see Section 4.5.1.2). Otherwise not used. Blank, B, G, S, W Channel Type: Somewhat redundant attribute unless using MIKE 11 or ISIS to manage 1D cross-section data. If attribute is not blank, TUFLOW searches the active cross-section database for hydraulic properties data (processed cross-section data) as follows: If a MIKE 11 database (.txt file), finds the processed data based on the Branch (now Inlet_Type attribute), Topo_ID (now Conn_2D) and XSect_ID_or_Chainage (now Conn_No) attributes. If Topo_ID is "\$LINK", searches the active MIKE 11 network (.nwk11) file for	Char(12)

No.	GIS Attribute	Description	Type
		the link cross-section details. For links, XSect_ID_or_Chainage must equal or fall within the upstream and downstream chainages of the link. If a ISIS database (.pro file), finds the processed data based on the label specified in the Topo_ID attribute. Leave the Branch attribute blank. C, I, R Channel Type: Not used.	
12	Conn_2D (previously Topo_ID)	Node or Pit: Since Build 2005-05-AN, is used to specify a “SX” or “SXZ” flag that automatically creates a 2D SX cell and connection at the 2D cell within which the 1D pit occurs. This negates the need to create SX objects in a 2d_bc layer. As of Build 2008-08-AA, the following options/changes are available: • “SX” can also be used to automatically connect 1D nodes as well as pits. • “SXL” can be specified to connect the 1D pit or node to the 2D domain and lower the 2D cell by the amount of the US_Invert attribute. The invert of the pit channel is set to the lowered 2D cell level. This is useful to help trap the water into the pit as it flows overland in the 2D domain. This feature works well in combination with the new Read GIS SA PITS option. • By default, if more than one 2D cell is automatically connected, pits are assumed to be connected using the Grade (G) approach and nodes the Sag (S) approach (see Section 4.5.1.3). To override these default approaches specify either “SXG” or “SXS”. As of Build 2010-10-AB, Pit Default 2D Connection can be used to set a global default value for Conn_2D, and up to four characters is now accepted (previously only three were accepted). “NO” can be used to not connect the node or pit to a 2D domain. Note that “SXXX” is reserved for internal checking purposes for Builds prior to 2010-10-AB (ie. do not specify SXXX for this attribute). Prior to Build 2005-05-AN, not used. Channel: See description for Inlet_Type (previously Branch) above. Although the name of this attribute has been changed, it can still be used as the Topo_ID if so desired.	Char(4)
13	Conn_No (previously XSect_ID)	Node or Pit: If “SX” is specified for Conn_2D, can be used to control the number	Integer

No.	GIS Attribute	Description	Type
	_or_Chainage)	of 2D cells connected as follows. See Section 4.5.1.3 for a discussion on how the 2D cells are selected. - A positive value increases the number of automatically selected 2D cells by the value of Conn_No. - If negative, this ignores the automatic approach and fixes the number of 2D cells connected to the absolute value of Conn_No. For example, a value of -1 would only connect the pit or node to one (1) 2D cell irrespective of the width. Channel: See description for Inlet_Type (previously Branch) above. If being used for a MIKE 11 cross-section chainage, specify to the nearest integer.	
14	Width_or_Dia (previously Diameter_or_Width)	Node: Not used. C, R, W Pit Type: As of Build 2005-05-AN, sets the diameter of a C pit channel or the width of a R or W pit channel. Q Pit Type: As of Build 2008-08-AA, used as a multiplier of the flow derived from the depth-discharge curve. For example, if Width_or_Dia equals 2, the flow is doubled. If set to zero (0), the MapInfo default, a value of 1.0 is used (ie. the discharge curve remains unchanged). C Channel Type: The pipe diameter in meters. R, W Channel Type: The width in meters of the box culvert or weir. A weir (W) may also be defined using a cross-section. See discussion for the Upstream_Invert attribute above. Blank, B, G, I, S Channel Type: Not used.	Float
15	Height_or_WF (previously Weir_Factor _or_Height)	Node or Pit: Since Build 2005-05-AN, sets the height of a R pit channel. Not used for other pit types. Prior to Build 2005-05-AN, not used. R Channel Type: The height in meters of the box culvert. W Channel Type: The calibration weir factor. If set to zero (<0.0001), it is changed to a value of 1.0. See Section 4.7.4.4. Blank, B, C, G, I, S Channel Type: Not used.	Float

No.	GIS Attribute	Description	Type
16	Number_of (previously No_of_Culverts)	Node: Not used. C, R, W Pit Type: As of Build 2005-05-AN, sets the number of pits (of same dimension) within the one pit channel. Q Pit Type: As of Build 2008-08-AA, used as a multiplier of the flow derived from the depth-discharge curve. For example, if Number_of equals 3, this assumes there are three pits lumped together and the flow is trebled. If set to zero (0), the MapInfo default, a value of 1 is used (ie. a single inlet pit is assumed). C, I, R Channel Type: The number of culvert barrels. If set to zero, one barrel is assumed. Not that prior to 2007-07-AD, if set to zero one barrel was assumed for the hydraulic conveyance calculations, but the storage contribution of the culvert was set to zero. Blank, B, G, S, W Channel Type: Not used.	Integer
17	Height_Cont (previously Culv_H_Contraction_ Coef)	Node or Pit: Since Build 2005-05-AN, sets the height contraction coefficient of a R pit channel, otherwise not used. I, R Channel Type: The height contraction coefficient for orifice flow at the inlet. Usually 0.6 for square edged entrances to 0.8 for rounded edges. If value exceeds 1.0 or is less than or equal to zero, it is set to 1.0. Not used for unsubmerged inlet flow conditions or outlet controlled flow regimes. Not used for C channels. Blank, B, C, G, S, W Channel Type: Not used.	Float
18	Width_Cont (previously Culv_W_Contraction_ _Coef)	Node or Pit: Since Build 2005-05-AN, sets the width contraction coefficient of a C or R pit channel, otherwise not used. C, I, R Channel Type: The width contraction coefficient for inlet-controlled flow. Usually 0.9 for sharp edges to 1.0 for rounded edges for R culverts. Normally set to 1.0 for C culverts. If value exceeds 1.0 or is less than or equal to zero, it is set to 1.0. Not used for outlet controlled flow regimes. Blank, B, G, S, W Channel Type: Not used.	Float
19	Entry_Loss	Node or Pit:	Float

No.	GIS Attribute	Description	Type
	(previously Culv_Entry_Loss)	Since Build 2005-05-AN, sets the entry loss coefficient of a C or R pit channel, otherwise not used. C, I, R Channel Type: The entry loss coefficient for outlet controlled flow (recommended value of 0.5). If value exceeds 1.0, it is set to 1.0. If value is less than zero (0), it is set to zero (0). If Structure Losses = ADJUST (the default) or the A flag is used (see Table 4.13), this value is adjusted according to the approach and departure velocities (see Section 4.7.4.1). If the culvert is discharging out of a manhole, this attribute is ignored and the entry loss used is that determined by the manhole energy losses formulation (see Section 4.5.2). The entry loss value used in the computation can be viewed over time using the _TSL output layer (see Section 7.4.4.1). Blank, B, G, S, W Channel Type: Not used.	
20	Exit_Loss (previously Culv_Exit_Loss)	Node or Pit: Since Build 2005-05-AN, sets the exit loss coefficient of a C or R pit channel, otherwise not used. C, I, R Channel Type: The exit loss coefficient for outlet controlled flow (recommended value of 1.0). If value exceeds 1.0, it is set to 1.0. If value is less than zero (0), it is set to zero (0). If Structure Losses = ADJUST (the default) or the A flag is used (see Table 4.13), this value is adjusted according to the approach and departure velocities (see Section 4.7.4.1). If the culvert is discharging into a manhole, this attribute is ignored and the exit loss used is that determined by the manhole energy losses formulation (see Section 4.5.2). The entry loss value used in the computation can be viewed over time using the _TSL output layer (see Section 7.4.4.1). Blank, B, G, S, W Channel Type: Not used.	Float

Table 4.15 1D Model Network (1d_nwke) OPTIONAL Attribute Descriptions

No.	Optional GIS Attribute	Description	Type
21	ES1	Not yet used (leave blank).	Char(50)
22	ES2	Not yet used (leave blank).	Char(50)
23	EN1	For BW, CW and RW channels, the flow width of weir (m) over the top of the B, C or R structure. Prior to Build 2007-04-AP, if < 0.001, uses the top width of B or R channel, or the C channel diameter. From Build 2007-04-AP onwards, the width is multiplied by the number of culverts attribute for C and R channels.	Float
24	EN2	For BW, CW and RW channels, the depth (m) of the bridge deck or culvert overlay.	Float
25	EN3	For BW, CW and RW channels, the depth of the hand rail (m). If < 0.001 assumes solid or no rail.	Float
26	EN4	For BW, CW and RW channels, % blockage of the rail (eg. 100 for solid rail, 50 for partially blocked, 0 for no rail).	Float
27	EN5	For BW, CW and RW channels, the weir calibration factor. Is set to 1.0 if < 0.001 is specified.	Float
28	EN6	Not yet used (leave as zero).	Float
29	EN7	Not yet used (leave as zero).	Float
30	EN8	Not yet used (leave as zero).	Float

4.5.5 How are Nodes and Channels Processed?

The procedure for reading and compiling the network is:

1 Nodes:

Read the location of any manually digitised nodes from the GIS layer(s) specified in the [Read GIS Network](#) command(s). Each node must have a unique ID. The network can be split up into several layers if desired. Note that a manually digitised node must only occur once in all the layers combined.

2 Channels and Cross-Sections (Except CS Tables):

Read the details of all channels from the GIS layer(s) specified by the [Read GIS Network](#) command(s). Any cross-section profiles, hydraulic properties or bridge loss coefficient tables linked via 1d_tab layer(s) (see [Read GIS Table Links](#) and Section 4.6.3) are processed. Also, any channel cross-section processed data from MIKE 11, ISIS or other external source is read at this stage. Each channel must have a unique ID. Nodes are automatically created at the ends of any channels found to not have a manually digitised node. The network can be split up into several layers if desired. You **can** overwrite an existing channel provided it has the same ID, and is connected to the same nodes as the channel being overwritten.

3 Node Storages (Except NA Tables):

Any elevation versus surface area tables at nodes linked via 1d_tab layer(s) (see [Read GIS Table Links](#) and Section 4.6.3) are processed.

4 Network Checks:

The network is checked for any incompatible node and/or channel IDs, connectivity, etc.

5 Any Fixed Field Formatted Cross-Sections (CS Tables):

Any cross-section processed data in the original fixed field format (see [CS](#) and [CS Data](#)) are read. If cross-section data is specified more than once for a channel the last cross-section data set read prevails (a “CHECK” message is provided in the .elf file whenever a channel’s cross-section data is overwritten). Every channel, except some hydraulic structures, must have a cross-section.

6 Any Fixed Field Formatted Node Storages (NA Tables):

Firstly, any node storages to be based on cross-section widths are calculated. Secondly, any node storage data in the original fixed field format is read. Note: if storage data is specified more than once for a node the last storage data read prevails.

7 Hydraulic Structures:

Additional information required for some hydraulic structures is read.

8 Boundary Conditions:

Boundary condition locations and data are read (see Section 4.10).

4.6 1D Topography

Channel and node topography are defined using tables. Channels require a table describing its hydraulic properties with height, and nodes require a table describing its storage volume (table of surface area versus height).

Channel and nodal property tables are assumed to be undefined above the highest point in the tables. If a computed water level exceeds the top of a table an ERROR occurs. If this occurs ten times for any one channel or node the run stops. [Depth Limit Factor](#) can be used to allow water levels to exceed the top of hydraulic properties and nodal area tables. Head limit checks are not applied to bridges or culverts.

4.6.1 Channel Hydraulic Properties (CS) Tables

Each channel requires a hydraulic cross-section properties table to define its conveyance. The properties are defined at a cross section positioned midway along the channel, or as of Build 2005-05-AN, can be derived from cross-sections located at the channel ends for G and S channels. The water level used for calculating the channel properties from the property tables is normally the mean of the water levels at the nodes, except for upstream controlled friction flow conditions and some hydraulic structure flow regimes.

The hydraulic properties are listed in Table 4.16. To generate the hydraulic properties, a channel requires a hydraulic properties (CS) table or a cross-section from which to calculate the properties. The exceptions are:

- For culverts (C and R) the attribute information supplied (ie. diameter, width, etc) is sufficient to define the hydraulic properties – **no cross-section properties table is required.**
- For weirs (W), if no cross-section or hydraulic properties table is specified, and a Diameter_or_Width attribute value greater than 0.01 is specified, the weir is defined as being a rectangular section 5 meters high based on the invert and width values.

Cross-section hydraulic properties tables may come from a number of sources:

- Calculated using a cross-section profile in a .csv or similar formatted file.
- A hydraulic properties table in a .csv or similar formatted file.
- External sources such as MIKE 11 processed data .txt files or ISIS .pro files.
- Traditional fixed field format (CS tables).

Cross-section profile and hydraulic properties data are accessed using a 1d_tab layer as described in Section 4.6.3.

External sources are defined using the [XS Database](#) and [M11 Network](#) commands. Cross-sections are extracted using the channel attributes Branch, Topo_ID and XSect_ID_or_Chainage as described in Table 4.14. The hydraulic properties table is automatically created from the external source. The conversion from these sources preserves all the hydraulic properties listed in Table 4.16 including any vertical variation in bed resistance (Manning's n) values.

Alternatively the hydraulic properties tables can be specified using standard fixed field formats, the documentation for which can be found in TUFLOW manuals prior to 2007 downloadable from www.tuflow.com. These tables can be entered directly into the .ecf file or preferably placed in a separate file and read using the [CS Data](#) command.

As of Build 2002-06-AB, it is possible to let the water level at a cross-section to extend above the highest elevation in the hydraulic properties table. The default is to treat any water levels that exceed the top of the table as being an instability (except for culverts and bridges), resulting in a WARNING being issued. See [Depth Limit Factor](#) for further details.

As of Build 2005-05-AN, rather than use a cross-section midway, cross-sections can be specified at the ends of G and S channels. This also saves having to enter upstream and downstream invert levels for these channels. See Section 4.6.4 for more details.

Table 4.16 Channel Cross-Section Hydraulic Properties

Property	Flag	Required	Description
Elevation	n/a	Mandatory	The water level elevation in m above the datum at which the hydraulic properties apply.
Width	n/a	Mandatory	The storage width in m.
Area	A	Optional	The effective flow area in m ² . If omitted, the area is calculated based on the elevations and widths starting at an area of zero at the lowest elevation.
Wetted Perimeter	P	Optional	The wetted perimeter in m. If omitted, the area is calculated based on the elevations and widths assuming a symmetrical channel.
Manning's n	N	Optional	The variation in Manning's n with height. Default value is the Manning's n is that assigned to the channel using the Manning_n attribute.
Manning's n Factor	F	Optional	A multiplication factor that varies with height applied to the Manning's n value. This option may be used instead of the N flag above.

4.6.2 Node Storage (NA) Tables

Storage at a node is defined using either a storage table of elevation versus surface area or using the widths of the channels. **All nodes must have their storage defined by one of these approaches.** Note that checks are also made for the following.

- The lowest elevation of a node must be below the lowest channel connected to the node.
- The highest elevation of a node is used to detect instabilities. Therefore, the highest elevation should be above the highest expected water level, unless [Depth Limit Factor](#) is used to extend storage properties above the highest elevation.
- The storage (surface area) of a node must not be zero at any level.

[Minimum NA](#) is useful for stabilising 1D nodes that have small surface areas, particularly at shallow depths or when wetting and drying.

[Minimum NA Pit](#) sets the minimum NA of the upstream (ground) nodes of all pit channels. This command was introduced for Build 2007-07-AA to differentiate upstream pit channel nodes from the [Minimum NA](#) setting. If the pit channel is connected to a 2D domain, this storage has no influence on the hydraulic computations, and increasing the value has no stability benefits.

The [Minimum NA](#) value is not applied to automatically created NA tables for upstream pit channel nodes, and for any node with a `1d_nwk Length_or_ANA` value that is greater than 0.001m^2 , provided there is no NA table that has been manually specified or created from channel storages. Backward compatibility only through using the [Defaults](#) == PRE 2007-07-AA command.

[Minimum Channel Storage Length](#) can be useful to add additional storage for stability reasons to nodes at the ends of very short channels. This command was introduced for Build 2007-07-AA.

4.6.2.1 Storage (NA) Tables

Using an elevation versus surface area table (NA table – NA stands for Nodal surface Area). This provides the opportunity to accurately define the storage of the floodplain including any backwater areas that do not act as flowpaths.

NA tables are entered using original fixed field formats as detailed in TUFLOW manuals prior to 2007 (downloadable from www.tuflow.com), or by accessing a table in a comma or space delimited text file linked to a `1d_tab` layer (see Section 4.6.3).

4.6.2.2 Using Channel Widths

The storage is calculated from one or more of the channels connected to the node. This approach does not require any specification of a NA table and is therefore the easiest. It is suited to nodes where the storage is accurately defined using the channel widths. For example, nodes connecting channels that model the in-bank flowpaths of a river. It may not be suited to, for example, floodplain areas where the storage may differ significantly from that calculated using the widths of the floodplain channels.

The channel storage approach is invoked using the UCS attribute described in Table 4.14. If the attribute is left blank or set to “Y” (yes) or “T” (true), the storage from the channel is assigned to both

of the nodes the channel is connected to. The storage is split equally to the two nodes. For each node the surface area at different elevations is calculated as the product of the channel width by half the channel length.

If the UCS attribute is set to “N” (no) or “F” (false), the storage of that channel is not used in calculating the storage at the two nodes.

Care should be taken using this option for G or S channels that have very steep slopes – check that the resulting NA table in the .eof file is satisfactory.

4.6.2.3 Storage Above Structure Obverts

As of Build 2005-05-AN: For models where the storage contributed by culverts and bridges is significant (eg. urban pipe model), use [Storage Above Structure Obvert](#) to minimise the storage contributed by these channels above their obverts (note that some storage is necessary to prevent a divide by zero in the equations).

Prior to Build 2005-05-AN: For culverts and bridges, storage based on using the channel width is also applied above the structure obvert according to the top width for B and R channels and the diameter for C channels. Also the No_of_Culverts attribute in the 1d_nwk layer was not used with a one culvert being adopted. This was a simplistic approach adopted on the basis that bridges and culverts generally have an insignificant contribution to the storage in a model. In these cases, if the additional storage above the obvert is a significant component of the overall storage in the model, separate NA tables may need to be specified where appropriate.

4.6.2.4 Procedure for Assigning NA Tables

It is important to note the logic in assigning node storage.

- 1 User defined NA tables are read from any 1d_tab NA table links. The overwrite principle applies, so that if a NA table has been previously defined, the latter NA table prevails.
- 2 User defined NA tables are read from any fixed field NA tables. The overwrite principle applies, so that if a NA table has been previously defined from Step 1 or in this step, the latter NA table prevails.
- 3 Nodes without any NA tables assigned in Steps 1 and 2 above, and have one or more connected channels that have the UCS attribute is left blank or set to “Y” or “T”, have their NA table automatically calculated from the channel widths.
- 4 As of Build 2005-05-AN, the Len_or_ANA attribute of a node (or pit) in a 1d_nwk layer can be used to add additional surface area to the NA table (see Table 4.14). If no NA table exists after the above steps, and the Len_or_ANA attribute is greater than zero, a NA table of constant surface area is created.
- 5 If there are any nodes remaining without a NA table, an ERROR occurs.

4.6.3 Free-form Tabular Input (1d_tab, 1d_xs, 1d_na, 1d_bg Layers)

Tables of cross-section profiles, cross-section hydraulic properties, nodal surface areas and bridge loss coefficients are accessed using links within a 1d_tab GIS layer. This allows these data to be entered in a free-form comma or space delimited format (as opposed to the fixed field formats) using .csv files that can be managed and edited in spreadsheet software such as Microsoft Excel.

Modellers often keep the different data sets separate and use the following prefixes. As numerous .csv files often are needed, a separate folder underneath the model folder (same level as the mi folder) is often used to store all of the .csv files and the GIS layer.

- 1d_xs for XZ cross-section profiles in an model\xs folder
- 1d_na for nodal surface area tables in a model\na folder
- 1d_bg for bridge loss coefficient tables in a model\bg folder

Table 4.17 describes each of the attributes and the method for determining which data to extract from the source file. Using the Column_1 attribute, several tables can be located in the one source file if desired.

[Read GIS Table Links](#) defines the 1d_tab layer(s) to be used for linking tabular data to nodes and channels. The method for linking is as follows:

- Lines and polylines (unlimited vertices as of Build 2002-10-AA – previously two or three vertices only) are used to link to channels. A two-point line must intersect or cross the channel line – it does not have to snap to a vertex on the channel line. If the two-point line crosses more than one channel, the channel that is closest to the mid-point of the line is selected. Three or more vertex lines must have one of the vertices snap to a vertex on the channel line and are given preference over any two-point line that crosses the channel line.
- Points are used to link to nodes. The point must be snapped to a node (or channel end if automatically creating nodes).
- Other objects are not used.

This feature was incorporated at Build 2002-08-AC and was further improved for Build 2003-03-AA during which the Flags attribute was split into two attributes: Type and Flags as described in Table 4.17.

Table 4.17 1D Table Links (1d_tab) Attributes

GIS Attribute	Description	Type
Read GIS Table Links Command		
Source	Filename (and path if needed) of the file containing the tabular data. Must be a comma or space delimited text file such as a .csv file.	Char(*)
Type	Two characters defining the type of table link as follows: “XZ” : Cross-section XZ profile (can include horizontal variations in resistance). The first column is the distance column, and the second the elevation column. Other optional columns are described under the Flags attribute below. “NA” : Nodal surface area versus height table. The first column is elevation and the second surface area in m ² . “CS” or “HW” : Cross-section hydraulic properties table. The first two columns must be elevation and width. Optional flags are described under the Flags attribute below “BG” or “LC” : Bridge loss coefficients (second column) versus elevation (first column) for bridge structures.	Char(2)
Flags	Optional flags are as follows: XZ Tables: “R”, “M” or “N” : The relative resistance (Column 3) is used to vary the bed resistance value (Manning’s n) across the section. Specify an “R” flag for relative resistance factor, an “M” flag to use a material number or an “N” flag (Build 2003-10-AA) for a Manning’s n value. (Note: As of Build 2003-03-AA, the Relative Resistance command to set the default relative resistance value as a factor or as a material is now redundant and should not be used.) “E” or “T” : Specify an “E” to use effective area or a “T” to use total area when calculating the flow area (see Section 4.6.7 and Flow Area). If neither is specified, the global value set using Flow Area is used. “P” : The position values (Column 4) are used to indicate whether an XZ point is left bank (1), mainstream (2) or right bank (3). P values must be entered as 1, 2 or 3. See Section 4.6.6.3. “A” : The addition values (Column 5) are used to raise or lower the Z value – this is useful, for example, for modelling siltation or erosion of a cross-section, raising, or adding blocked rails to, a weir cross-section, etc. NA Tables: No optional flags. CS or HW Tables: “A” : Flow area (Column 3) “P” : Wetted perimeter (Column 4) “F” or “N” : Vertical change in resistance (Column 5). Use “F” for a multiplication factor and “N” for a Manning’s n value. Build 2008-08-AC incorporates a bug fix if using “N” . “E” : Effective flow width (Column 6)	Char(8)

GIS Attribute	Description	Type
	BG or LC Tables: No optional flags.	
Column_1	Optional. Identifies a label in the Source file that is the header for the first column of data. Values are read from the first number encountered below the label until a non-number value, blank line or end of the file is encountered. If this field is left blank, the first column of data in the Source file is used.	Char(*)
Column_2	Optional. Identifies a label in the Source file that is in the header for the second column of data. If this field is left blank, the next column of data after Column_1 is used.	Char(*)
Column_3	Optional. Identifies a label in the Source file that is in the header for the third column of data. If this field is left blank, the second column of data after Column_1 is used.	Char(*)
Column_4	Optional. Defines the fourth column of data.	Char(*)
Column_5	Optional. Defines the fifth column of data.	Char(*)
Column_6	Optional. Defines the sixth column of data.	Char(*)
Z_Increment	Optional. Sets the height increment in meters to be used for calculating hydraulic properties from a XZ cross-section profile. If less than 0.01, the increment is determined automatically. Only used for XZ cross-section data.	Float
Z_Maximum	Optional. Sets the maximum elevation in meters to be used for calculating hydraulic properties from a XZ cross-section profile. If less than the lowest point in the cross-section profile, Z_Maximum is taken as the highest elevation in the profile. Only used for XZ cross-section data.	Float
Skew (in degrees)	Optional. Adjusts the cross-section properties for XZ and CS/HW data according to the skew angle. Useful where the cross-section line is surveyed oblique to the flow direction. The skew angle is zero degrees in the direction of flow and 90 degrees if surveyed a right angles(!) to the direction of flow. For example, a value of 45 adjusts the horizontal dimensions by dividing by the $\sqrt{2}$.	Float

4.6.4 End Cross-Sections

As of Build 2005-05-AN, cross-sections can be specified for G and S channels using the 1d_tab approach at the channel ends, rather than a single cross-section somewhere along the channel. This has a couple of benefits:

- The upstream and downstream inverts for G and S channels can be based on the beds of the cross-sections, thereby saving some effort to enter this information. To do this, set the Upstream_Invert and Downstream_Invert attributes in the 1d_nwk layer to -99999. If either of these attributes is greater than -99999, the invert is set to the attribute value rather than that of the cross-section bed.
- Cross-section surveys from other 1D models often have the cross-sections at the channel ends, therefore, this makes it easier to use these data.

There are a few rules on how end cross-sections are interpreted and applied as follows:

- The 1d_tab (or 1d_xs) cross-section lines must have a vertice snapped to the channel end.
- If a 1d_tab cross-section line occurs elsewhere along a G or S channel with end cross-sections, the midway cross-section prevails. This is particularly useful where two channels' ends are snapped to an end cross-section, but the end cross-section is to be applied to only one of the channels (eg. one channel is a river channel using end cross-sections, and the other is an overbank channel). For the overbank channel, specify a cross-section line somewhere along the channel, and preference will be given to this cross-section rather than the end cross-section.
- End cross-sections cannot be used to override previously defined cross-section properties for a G or S channel. You can override the end cross-sections using a midway cross-section.
- For channels other than G and S channels, end cross-sections are ignored.

4.6.5 Interpolated Cross-Sections

As of Build 2008-08-AA, cross-sections are now interpolated for channels (excluding C and R culvert channels) that have not been assigned a cross-section. A series of channels may now be digitised between two cross-sections, and the cross-section properties at each channel are linearly interpolated between the two cross-sections. The protocols applied when interpolating cross-sections and setting Manning's n values are:

- If a channel has a cross-section at each end, the processed data of these cross-sections is averaged (as per previous builds).
- If a channel has a cross-section midway, this cross-section takes priority over any end cross-sections (as per previous builds).
- If a channel only has one end cross-section, TUFLOW traverses upstream/downstream to find the next available cross-section, and uses this to interpolate the cross-section properties for that channel. The next available cross-section can be a midway or end cross-section.

- If a channel has no cross-sections attached to it, TUFLOW traverses upstream and downstream to find the nearest cross-sections, and interpolates the channel properties based on these cross-sections.
- When traversing upstream/downstream to find a cross-section:
 - If a junction (three or more channels snapped together) is reached (excluding pits and connectors), an ERROR is issued as it is not possible to determine which branch to follow. Note, channels connected to a junction using a connector (Type “X”) are not used for traversing, therefore use connectors to connect side channels to the main branch.
 - The digitised direction of the channel is important and controls the direction used to traverse upstream and downstream. Ensure the channels are digitised in a consistent direction (usually heading downstream).
- If a channel has an end cross-section only at one end, and no cross-section is found when traversing, this end cross-section is used at both ends for that channel only (as pre previous builds).
- The inverts are also interpolated using the cross-section beds (unless the inverts have been manually entered into the 1d_nwk attributes). Specify -99999 for the 1d_nwk channel inverts to have them interpolated.
- Cross-sections that are interpolated can be of any format except the old fixed field formats. This includes CS or HW 1d_tab formats (see Table 4.17).
- The Manning’s n value assigned to the channel’s cross-section is as follows:
 - If the cross-sections used for interpolation have no Manning’s n values (ie. for XZ cross-sections, M or N was not specified, or for CS/HW cross-sections, N was not specified), the 1d_nwk Manning’s n attribute of the channel is used.
 - If the cross-sections used for interpolation have Manning’s n values, the value is interpolated from the cross-section n values (at the bed) and multiplied by the 1d_nwk Manning’s n attribute of the channel. In this case the 1d_nwk n_or_n_F attribute is a multiplier that can be used to calibrate the model.
 - If one of the two cross-sections used for interpolation has a Manning’s n value, and the other does not, the n value used is interpolated using the channel’s 1d_nwk Manning’s n value and the cross-section’s n value. Ideally, the model should be set up using the same approach everywhere so that this situation does not arise as it may cause undesirable results. A WARNING is issued if this occurs.

The interpolation of cross-sections is the default as of Build 2008-08-AA. If [Defaults](#) == PRE 2008-08 is specified, the interpolation is switched off. [Interpolate Cross-Sections](#) can also be used to switch this feature ON or OFF.

4.6.6 XZ Relative Resistances

Varying the resistance across an XZ cross-section is possible by using either a relative resistance factor (R flag) or different material values (M flag). These are discussed further in the sections below.

The relative resistance value applies midway to either side of the X value (except the first and last X values where it only applies to midway to the single neighbouring X value). This is slightly different from some other 1D hydraulic modelling software that apply relative resistance values from the previous X value to the current X value or from the current to the next.

Sections of a cross-section can be “removed” by entering -1 (negative one) for a resistance value.

4.6.6.1 Relative Resistance Factor (R)

The relative resistance factor (R) is a multiplication factor applied to the primary Manning’s n value of the channel. Wherever the R value changes across the cross-section, a new parallel sub-channel is created. The total conveyance for the whole cross-section is determined by carrying out a parallel channel analysis of all the sub-channels. This approach allows the variation in bed resistance across a cross-section to be accounted for, and to force a parallel channel analyses so that conveyance does not decrease with height when the wetted perimeter suddenly increases (eg. when overbank areas just become wet).

If using effective area (see Section 4.6.8), an R of 1.0 must occur at some point in the profile to indicate the primary sub-channel. If a value of 1.0 is not found an ERROR 1070 occurs, as grossly incorrect channel velocities can occur when using effective area. The Manning’s n value of the primary sub-channel is that specified in the 1d_nwk layer for the channel. The primary sub-channel does not have to be the lowest part of the cross-section.

4.6.6.2 Material Values (M)

If using material values (M), the Manning’s n value to be applied is taken from the .tmf file (see [Read Materials File](#)). If the “P” flag is not used, the material at the lowest Z value (cross-section bed) is used as the primary material, which then corresponds to a relative resistance factor of 1.0. If no material values are specified, a material value of one (1) is applied over the whole cross-section. If the P flag and values are used, the primary material is determined as that at the lowest Z value in the mainstream channel (see Section 4.6.6.3).

When using materials, the n_or_n_F (previously Manning_n) value in the 1d_nwk layer becomes a multiplier and should be set to one (1.0). If justified, it can be adjusted for calibration purposes. For example, if a slightly higher resistance is desired along a channel, rather than setting different material values, change the n_or_n_F value in the 1d_nwk layer to, say, 1.1 to increase all Manning’s n values across the cross-section by 10%.

As of Build 2008-08-AA, negative material values in the M column of a XZM cross-section table can now be specified to disable (ie. block out or remove) sections of a cross-section. Previously a negative value would result in an ERROR message stating that that material could not be found. The negative value must be the negative of a valid Material ID in the .tmf file (see [Read Materials File](#)).

4.6.6.3 Manning's n Values (N)

If using Manning's n values (N), the n value is specified directly, noting that the Manning_n value in the 1d_nwk layer becomes a multiplier and should be set to one (1.0). See discussion above for using material values. A value of -1 ignores that section of the profile.

4.6.6.4 Position Flag (P)

The position values are used to indicate whether an XZ point is left bank (1), mainstream (2) or right bank (3). Incorporated in Build 2003-03-AA, the P value is used to indicate where the mainstream sub-channel is located. If materials (M flag) are used, the primary material is taken as that at the lowest Z value in the mainstream sub-channel. If the P flag and values are not specified, the primary material is that at the lowest Z value across the whole section.

It is intended that the P values be used for other processing and post-processing of results in future releases.

4.6.7 Reducing Conveyance with Height

The conveyance of a cross-section may reduce with height where there is a sudden increase in the wetted perimeter compared with a relatively small increase in flow area, causing the hydraulic radius to reduce despite the water level increasing. If this occurs, a WARNING is issued and it is strongly recommended that the cross-section be reviewed and corrected.

The most common cause for the reduction in conveyance with height occurs when the extent of inundation across the cross-section increases markedly during the transition from in-bank to out-of-bank flow. The reducing conveyance with height problem is usually resolved by forcing a parallel channel analysis by specifying a change in resistance using the R, M or N flag discussed in the sections above.

As of Build 2006-06-AA, [Conveyance Calculation](#) == ALL PARALLEL is the default setting. This forces a parallel channel analysis based on splitting the cross-section into parallel channels at every distance (X) value. This approach generally ensures that conveyance does not reduce with height. Using this approach to calculate the hydraulic properties has a tendency to produce a slightly more efficient cross-section (higher conveyance), similar to the resistance radius formulation used by some schemes. Therefore, a slightly higher Manning's n value (by ~10%) may be needed to achieve similar results.

4.6.8 Effective Area versus Total Area

For XZ Cross-Sections, the flow area is calculated as an effective area (E flag) or a total area (T flag). The flag will override the global setting set by [Flow Area](#).

If there is no variation in relative resistance across the cross-section there is no difference between effective and total areas. This is dependent on the relative resistance being 1.0 across the whole section. (An ERROR 1070 is produced if the relative resistance is not 1.0 somewhere along the cross-section when using effective area.)

For an open channel, the total conveyance of a cross-section is not affected by whether effective or total area is used because in the case of effective area the wetted perimeter is adjusted to compensate for the change in flow area so as to produce the same conveyance as would occur for total area. For special channels that use cross-sections such as bridges, weirs and irregular culverts, the flow area used is the effective or total area as specified. This can be useful if the effects of blockage or congestion within the section needs to be modelled.

The primary differences between using effective and total area are:

- The channel velocity calculated is the depth and width average of the primary (normally mainstream) parallel sub-channel if using effective area, and the depth and width averaged of the whole cross-section if using total area.
- Where the effective and total areas are significantly different, the channel velocities used in the 1D momentum equation will be significantly different. If the channel velocity is sufficiently high and different depending on whether effective or total area is used, the inertia terms in the 1D momentum equation may affect the results. Note the frictional (bed resistance) term in the momentum equation is NOT affected as the hydraulic properties for the cross-section are adjusted so that the total conveyance is the same irrespective of whether effective or total area is used.

Effective area gives a more reliable calculation of the mainstream velocity, and therefore, a more accurate estimate of approach and exit velocities of structures, and more appropriate velocities for advection-dispersion and sediment transport calculations. Where velocities are not high or significantly changed when using effective or total area, the water level and flow results are usually identical or very similar.

4.7 Hydraulic Structures and Supercritical Flow

4.7.1 How to Model Bridges and Box Culverts

Bridges, box culverts and other structures that constrict flow can be modelled in 2D rather than using 1D elements provided the flow width of the structure is of similar or larger size than the 2D cell size. Cells are modified in their height (invert and obvert) and width. For bridges, additional losses associated with flow reaching the underside of the deck is specified. For box culverts, the additional resistance for vertical walls is specified. Additional form losses (energy head losses) can be specified for all FCs.

Weir flow (across levees and other embankments) is modelled in 2D domains by default, but can be changed using options in the [Free Overfall](#) command. Weirs may also be modelled using 1D elements.

Modelling hydraulic structures in 2D domains must be carried out with a good understanding of the limitations of different approaches and the different flow regimes possible. The modeller must understand why and where the energy losses occur when assigning form losses to a 2D cell or contraction and expansion losses to a 1D element (Syme 2001b).

It is important to note that contraction and expansion losses associated with structures are modelled very differently in 1D and 2D schemes. 1D schemes rely on applying form loss coefficients, as they cannot simulate the horizontal or vertical changes in velocity direction and speed. 2D schemes model these horizontal changes and, therefore, do not require the introduction of form losses to the same extent as that required for 1D schemes. However, 2D schemes do not model losses in the vertical or fine-scale horizontal effects (such as around a bridge pier) and, therefore, may require the introduction of additional form losses. See Syme 2001 for further details.

It is strongly recommended that the losses through a structure be validated through:

- **Calibration to recorded information (if available).**
- **Cross-checked using desktop calculations based on theory and/or standard publications (eg. Hydraulics of Bridge Waterways, US FHA 1973).**
- **Crosschecked with results using other hydraulic software.**

To validate structure flows and energy losses:

- Specify time-series output (PO) lines of flow (Q) and flow area (QA) across the structure (see Section 4.8). Upstream and downstream water levels may also be specified or taken from the map (SMS) output.
- Using the upstream and downstream water levels, determine whether flow is upstream or downstream controlled and estimate the flow using theoretical equations or other method.
- Using publications such as Hydraulics of Bridge Waterways (US FHA 1973), determine the energy loss coefficient and compare this with the total energy loss calculated in the model. The total energy loss (ζ_{tot}) is the upstream head minus the downstream head ($h_1 - h_2$) divided by the dynamic head based on the depth and width averaged velocity (V) (ie.

Q_-/QA) as given below. Clearly, any energy losses associated with bed resistance (eg. Manning's equation) need to be allowed for by taking this amount out of the $(h_1 - h_2)$ term.

$$\zeta_{tot} = (h_1 - h_2) \frac{2g}{V^2}$$

- Using other software (eg. HEC-RAS), create a check model using the flow and downstream water level as boundaries and compare the calculated upstream water levels.

Table 4.18 Hydraulic Structure Modelling Approaches

Structure	1D Approach	2D Approach
Box Culvert (For culverts with a steep slope, use a 1D element)	OK	OK
Circular Culvert	OK	N/A
Bridge	OK	OK
Weirs	OK	OK

1D Approach Preferred approach where the total structure width is less than one or two 2D cell widths.

Entry and/or exit losses may need to be reduced where the structure width is significant compared with the cell size (Syme 2001b).

SX Link: Momentum is not transferred into or out of the 1D element to/from the 2D domain. “Suppressed” flow patterns in the 2D domain occur at the structure outlet when using 1D elements, especially if the structure width is significant compared with the cell size. The water tends to spread, rather than jet out as there is no inertia across the link. This affect of this is illustrated in Figure 4-4, which shows the effect on flow patterns and the preservation of inertia across 1D/2D links when modelling a structure. The middle image (red velocity arrows) is that using SX links, whilst the top image (green arrows) is that using a fully 2D solution. As can be seen, using a SX link the water tends to spread from the structure outlet, as opposed to forming a jet as in the fully 2D solution which is conserving momentum. When using the SX link, a jet like effect can be created using “wing walls” in the 2D domain at the structure outlet by assigning flood free elevations to the ZU and ZV Zpts either side of where the 1D element discharges into the 2D domain.

HX Link: Momentum is not transferred into or out of the 1D element to/from the 2D domain, however the velocity field across the HX link is assumed to be undisturbed. Provided the HX link is appropriately located (ie. perpendicular to the

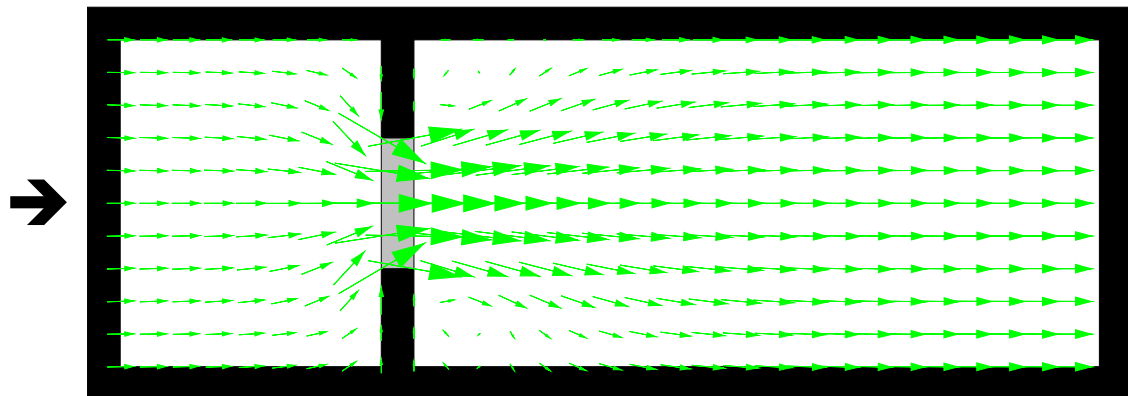
flow field) this produces the effect of preserving momentum as illustrated by the dark blue arrows (bottom image in Figure 4-4). Use of HX links at a structure may require a smaller 1D timestep than that required by a SX link.

2D Approach

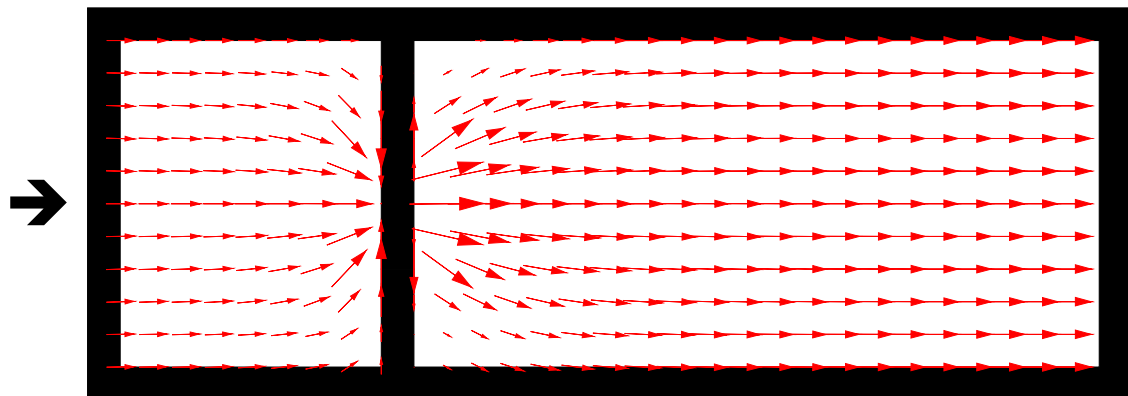
Preferred where the total structure width is greater than one or two 2D cell widths. The flow area must be adequately represented by the 2D Zpts and any adjustments to cell widths (see Section 4.7.2). The head drop across the structure during different flow regimes should be validated against other methods and/or literature.

Some additional form losses are normally required to achieve correct head drop (see Syme 2001b). Where the cell size is less than the depth, use the Smagorinsky Viscosity formulation. Care should be exercised using cell sizes less than 2m (Barton 2001).

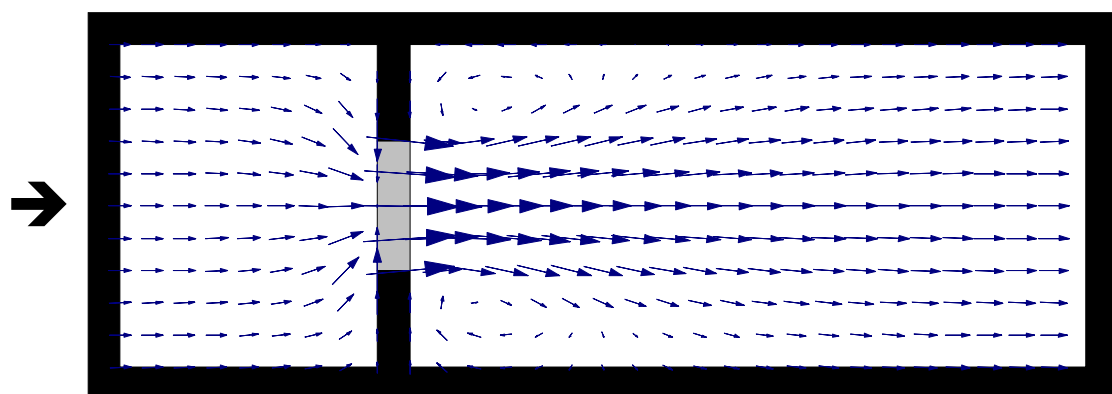
Momentum is transferred through the structure as shown in the top image (green arrows) in Figure 4-4, providing far more realistic flow patterns than using a 1D element with a SX link as illustrated by the middle image (red arrows) in Figure 4-4.



Flow patterns using 2D FC cells (ie. a fully 2D solution)



Flow patterns using a 1D element connected to 2D SX links



Flow patterns using a 1D element connected to 2D HX links

Figure 4-4 Different Flow Patterns from 2D FCs and 1D/2D Links when Modelling a Submerged Culvert

4.7.2 2D Flow Constrictions (FC)

4.7.2.1 2d_fcsh and 2d_fc Layers

Flow constriction (FC) attributes allow the user to constrict the flow across a 2D cell side in a number of ways so as to model large hydraulic structures such as bridges and banks of box culverts. 2D cell sides can be modified in the following ways:

- Place a lid (obvert or soffit) on the cell side.
- Change the flow width of the cell side.
- Add additional form (energy) losses.
- Include side wall friction (“BC” FC_Type only).

As of Build 2008-08-AA, flow constrictions can now be entered using GIS layers using [Read GIS FC Shape](#) in the .tgc file (not the .tcf file). This command is preferred over the previous (and now outdated) [Read GIS FC](#). The advantages of using [Read GIS FC Shape](#) over [Read GIS FC](#) are:

- The GIS layer is cell size independent. Therefore if you change the 2D cell size, this layer does not need to be reworked (as is the case with a [Read GIS FC](#) layer).
- A polygon can be used with [Read GIS FC Shape](#) to modify a group of 2D cell sides, whereas a [Read GIS FC](#) polygon only modified a single cell, that being the cell where the polygon centroid was located.
- [Read GIS FC Shape](#) works directly on the cell mid-sides, so it is possible to have a “thin” FC, whereas [Read GIS FC](#) works on whole cells and assigns the attributes to all four cell sides.
- [Read GIS FC Shape](#) can have a 3D shape representing the obvert or soffit (eg. for a sloping bridge deck or even an arched bridge).
- [Read GIS FC Shape](#) is applied from the .tgc file, which some modellers would prefer over [Read GIS FC](#) which is from the .tcf file.
- To model flow in 2D both under and over bridges, pipelines or other “horizontal” obstructions, use [Read GIS Layered FC Shape](#), which is a more advanced version.

Table 4.19 describes the different 2d_fcsh layer attributes as used by [Read GIS FC Shape](#). Use the 2d_fcsh_uvpt_check and 2d_uvpt_check layers that are output if [Write Check Files](#) is set, to cross-check that the changes to the cell-sides are as intended.

As described in Section 4.4.11, the point objects can be placed in a separate layer for a 2d_fcsh layer. In this case, only the first two attributes are required.

The original [Read GIS FC](#) continues to be supported and its attributes are documented in Table 4.20.

Table 4.19 Flow Constriction Shape (2d_fcsh) Attribute Descriptions

GIS Attribute	Description	Type
Invert	Invert of constriction (m above datum). To leave the Zpt levels unchanged (ie. use the existing Zpt elevations), enter a value of 99999.	Float
Obvert_or_BC_Height	FC_Type = Blank or “BD”: Obvert (soffit) of constriction in m above datum. FC_Type = “BC”: Height of box culvert (m). FC_Type = “FD”: Floating depth (m) of the deck (ie. depth below the water line). Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.	Float
Shape_Width_or_dMax	Point: Not used. Line: See same attribute in Table 4.8 for 2d_zsh layers. Polygon: Not used.	Float
Shape_Options	Point: Not used. Line or Polygon: MAX, RIDGE or RAISE: Only changes a Zpt (Invert) elevation if the Z Shape elevation at the Zpt is higher. MIN, GULLY or LOWER: Only changes a Zpt (Invert) elevation if the Z Shape elevation at the Zpt is lower. Line: TIN: Indicates the line is to only be used for generation of TINs within polygons (only sections of TIN lines that fall within polygons are used).	Char(20)
FC_Type	FC type where: - Blank for general (does not include allowances for any vertical walls or friction from underside of deck. “BC” for Box Culverts (Note: At this stage, BC is only available if Manning’s n bed resistance option is specified.) “BD” for Bridge Deck “FD” for Floating Deck	Char(2)
pBlockage	The percentage blockage of the cells. For example, if 40 is entered (ie. 40%), the cell sides are reduced in flow width by 40%, ie. is set to 0.6 times the full flow width.	Float

GIS Attribute	Description	Type
FLC_below_Obvert	Form loss coefficient to be applied below the FC obvert. Used for modelling fine-scale “micro” contraction/expansion losses not picked up by the change in the 2D domain’s velocity patterns (eg. bridge pier losses, vena-contracta losses, 3rd (vertical) dimension etc). Note: So that this attribute is independent of 2D cell size it has different treatment depending on the object it is attached to as follows: Line: For thin lines, the FLC value is applied to the cell sides unchanged. For thick (whole cell) lines, the FLC value is divided by two (two cell sides in the direction of flow). For wide lines the FLC value is divided by the number of cells across the line (ie. the line’s width divided by the cell size) and applied to all cell-sides. Polygon: FLC is the form loss per metre length (in the predominant direction of flow). FLC values are not dependent on the flow width, but are on the length of travel in the direction of flow. However, if a negative FLC value is specified, the absolute value is taken and applied unadjusted to all cell-sides affected by the shape. Note, this is not cell size independent, therefore if the 2D cell size is changed, this attribute also needs to be changed. Should be used as a calibration parameter. The form loss coefficient is applied as an energy loss based on the dynamic head equation below where ζ_a is the FLC value. $\Delta h = \zeta_a \frac{V^2}{2g}$	Float
FLC_above_Obvert	Form loss coefficient to be applied above the FC obvert. See FLC_below_Obvert attribute above for more information.	Float
Mannings_n	For box culverts (BC), the Manning’s n of the culverts (typically 0.011 to 0.015) should be specified. This value prevails over any other bed resistance values irrespective of where in the .tgc file they occur (the exception is if another FC BC object overrides this one). If set to less than 0.001, a default value of 0.013 is used. For bridge decks (BD), can be used to introduce additional flow resistance once the upstream water level reaches the bridge deck obvert (or soffit). For floating decks (FD) this is always the case as the deck soffit is permanently submerged. The additional flow resistance is modelled as an increase in bed resistance by increasing the wetted perimeter at the cell mid-side by a factor equal to $(2 * \text{Bed\_n}) / \text{FC\_n}$. For example, if the FC Manning’s n and the bed Manning’s n values are the same, the wetted perimeter is doubled, thereby reducing the	Float

GIS Attribute	Description	Type
	conveyance and increasing the resistance to flow. To be used as a calibration parameter to fine-tune the energy losses across a bridge or floating structure. Ignored for blank FC_Type.	
BC_Width	The width of one BC culvert barrel in metres. For example, if there are 10 by 1.8m wide culverts, enter a value of 1.8. Applicable to BC FC_Type only. Not used by other types of FCs.	Float

Table 4.20 Flow Constriction (2d_fc) Attribute Descriptions

Note: 2d_fcsh layers are preferred to 2d_fc layers as of Build 2008-08-AA.

GIS Attribute	Description	Type
type	Secondary flag identifier where: - Blank for general (does not include allowances for any vertical walls or friction from underside of deck. “BC” for Box Culverts (Note: At this stage, BC only available if Manning’s n bed resistance option specified.) “BD” for Bridge Deck “FD” for Floating bridge Deck	T
invert	Invert of constriction (m above datum). Mandatory for box culverts (type = “BC”). If not a box culvert, and you wish to leave the Zpt levels unchanged (ie. use the existing Zpt elevations), enter a value greater than the obvert level (see next attribute).	N
obvert_or_BC_height	type = blank or “BD”: Obvert of constriction (m above datum) type = “BC”: Height of box culvert (m). Values less than 0.01 are set to 0.01. type = “FD”: Floating depth (m) of the deck (ie. depth below the water line). Build 2004-04-AD. Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.	N
u_width_factor	Flow width constriction factor in the X-direction (ie. the flow width perpendicular to the X-direction). For example, a value of 0.6 sets the flow width at the left hand and right hand sides of the cell to 60% of the cell width. Values less than 0.001 are set to 1. Use a value of 1.0 to leave the flow width unchanged. Values greater than 1 can be specified.	N

GIS Attribute	Description	Type
v_width_factor	Width constriction factor in the Y-direction. See description above for u_width_factor.	N
add_form_loss	Form loss coefficient. Used for modelling fine-scale “micro” contraction/expansion losses not picked up by the change in the 2D domain’s velocity patterns (eg. bridge pier losses, vena-contracta losses, 3rd (vertical) dimension etc). Can be used as a calibration parameter. The form loss coefficient is applied as an energy loss based on the dynamic head equation below where ζ_a is the add_form_loss value. The form loss coefficient is applied 50/50 to the right and left sides (u-points) of the cell, and similarly to the v-points. $\Delta h = \zeta_a \frac{V^2}{2g}$	N
Mannings_n	For box culverts (BC), the Manning’s n of the culverts (typically 0.011 to 0.015) should be specified. This overwrites any previously specified Manning’s n values at the cell’s mid-sides. If set to less than 0.001, a default value of 0.013 is used. For bridge decks (BD), can be used to introduce additional flow resistance once the upstream water level reaches the bridge deck obvert or soffit. For floating decks (FD) this is always the case as the deck soffit is permanently submerged. The additional flow resistance is modelled as an increase in bed resistance by increasing the wetted perimeter at the cell’s mid-sides by a factor equal to (2.*Bed_n)/FC_n. For example, if the FC Manning’s n and the bed Manning’s n values are the same, the wetted perimeter is doubled, thereby reducing the conveyance and increasing the resistance to flow. To be used as a calibration parameter to fine-tune the energy losses across a bridge or floating structure. Note that Build 2007-07-AA incorporates a fix to BD and FD FC cells that did not correctly apply the Manning’s n attribute (see Defaults == PRE 2007-07-AA for information on backward compatibility). Ignored for “Blank” type FC.	N
no_walls_or_neg_width	Number of vertical walls per grid cell. If set to zero (between -0.001 and 0.001) one vertical wall per cell is used. A non-integer value can be specified. Alternatively, and more easily, specify the width of one culvert in metres by using a negative value. For example, if the culverts are 1.8m wide, enter a value of -1.8 and the number of vertical walls per cell is automatically calculated. Applicable to Box Culverts only. Not used by other types of FCs.	N
blocked_sides	Indicates whether any of the walls of the constricted cell(s) are blocked off (ie. no flow across/through the side wall). Specify one or more of the following letters in any order with in the field to indicated which wall(s) are blocked: • “R” – block right hand side wall	T

GIS Attribute	Description	Type
	“L” – block left hand side wall “T” – block top side wall “B” – block bottom side wall Note: the quotes should be omitted.	
invert_2	leave blank (not used as yet)	T
obvert_2	leave blank (not used as yet)	T
Comment	General comment or note for own use – not used.	T

4.7.2.2 Applying FC Attributes

When adapting structure loss coefficients from a 1D model or from coefficients that apply across the entire waterway, for example, from *Hydraulics of Bridge Waterways* (FHA 1973), the following should be noted:

- The TUFLOW 2D solution automatically predicts the majority of “macro” losses due to the expansion and contraction of water through a constriction, or round a bend, provided the resolution of the grid is sufficiently fine (see Barton 2001, Syme 2001b).
- Where the 2D model is not of fine enough resolution to simulate the “micro” losses (eg. from bridge piers, vena contracta, losses in the vertical (3rd) dimension), additional form loss coefficients and/or modifications to the cells widths and flow height need to be added. This can be done by using flow constrictions (FC cells). Additional form loss can also be added using [Read GIS FLC](#).
- The additional or “micro” losses, which may be derived from information in publications, such as *Hydraulics of Bridge Waterways*, need to be either:
 - distributed evenly over the FC cells across the waterway by dividing the overall additional loss coefficient by the number of cells; or
 - assigned unevenly (eg. more at the cells with the bridge piers), however, the total of the loss coefficients should be equivalent to the required overall additional loss coefficient.
- The head loss across key structures should be reviewed, and if necessary, benchmarked against other methods (eg. using HEC-RAS or *Hydraulics of Bridge Waterways*). Note that a well-designed 2D model will be more accurate than a 1D model, provided that any “micro” losses are incorporated.
- Ultimately the best approach is to calibrate the structure through adjustment of the additional “micro” losses – but this, of course, requires good calibration data!

An example of how to apply 2D FCs to a bridge structure is shown in Figure 4-5. The loss coefficient quoted in the figure is an example, and is not necessarily applicable to other structures. Every structure is invariably different!

When applying FCs the best approach is to view the structure as a collection of 2D cells representing the whole structure, rather than being too specific about the representation of each individual cell. A good approach is to use a 2d_po layer to extract time histories of the water levels upstream and downstream of the structure and of the flow and flow area upstream, downstream and through the structure (see Section 4.8.1).

Of particular importance is to check that the correct flow area through the structure is being modelled. Digitise a 2d_po QA line through the structure from bank to bank, and use this output to cross-check the flow area of the 2D FC cells is appropriate (the QA line will take into account any adjustments to the 2D cells due to FC obverts and changes to the cell side flow widths). If the overall structure flow area is not correct, then the velocities within the structure will not be correct and therefore the energy losses due to the changes in velocity direction and magnitude and additional form losses will not be well modelled.

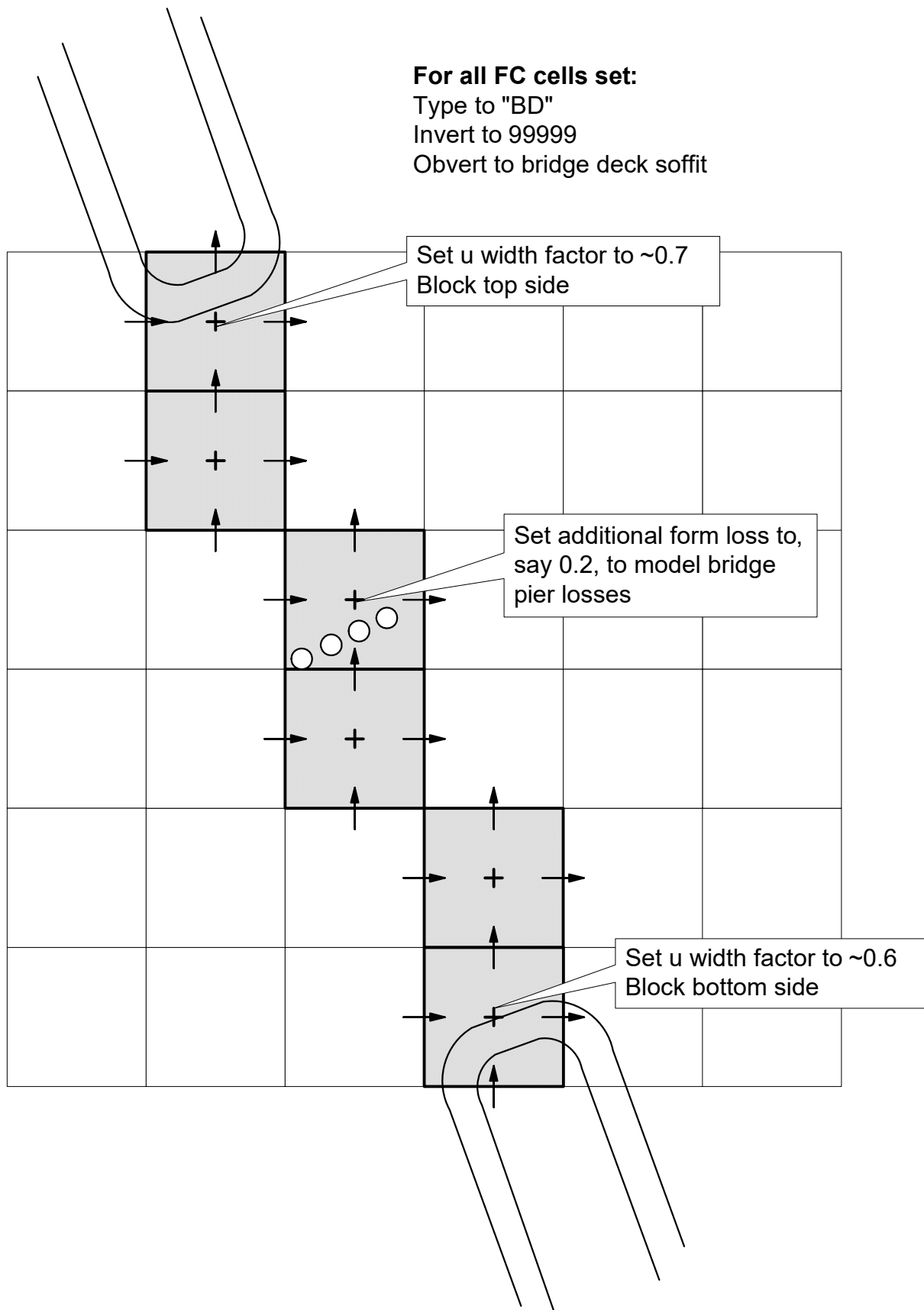


Figure 4-5 Setting FC Parameters for a Bridge Structure

4.7.2.3 Layered Flow Constrictions (2d_lfcsh Layers)

Layered flow constrictions are similar to FCs, but allow the attributes to be varied with water depth. This provides the opportunity to model in 2D the flow under and over a bridge deck, or a pipeline crossing a waterway.

Four layers (not GIS layers!) are represented, with the bottom three layers each having their own attributes. The top, fourth, layer assumes the flow is unimpeded (eg. flow over the top of a bridge). Within the same shape, the invert of the bed, and thickness of each layer can vary in 3D. Each layer is assigned its own percentage blockage and form loss coefficient.

For example, the layers of a bridge structure would be along the following lines.

- **Layer 1:** Beneath the bridge deck. Might be 5% blocked due to the bridge piers and have a small form loss for the energy losses associated with the piers.
- **Layer 2:** The bridge deck. This would be 100% blocked and the form loss coefficient would increase due to the additional energy losses associated with flow surcharging the deck.
- **Layer 3:** The bridge rails. These might be anything from 100% blocked (solid concrete rails) to 10% blocked (very open rails). Some form losses would be specified depending on the type of rails.
- **Layer 4:** Flow over the top of the rails - flow assumed to be unimpeded.

Layered FCs function by adjusting the flow width of the 2D cell so as to represent the combination of blockages of the four layers, and by accumulating the form losses. When the flow is only within Layer 1, only the attributes of Layer 1 are applied. As the water level rises into Layer 2, the influence of the Layer 2 attributes increase as the water continues to rise. Similarly for Layer 3 and Layer 4.

The cell side flow width is calculated by summing the flow areas of each layer (including the effects of layer blockages), and dividing by the water depth.

The form losses are applied as follows:

- If the water is entirely within Layer 1, the Layer 1 FLC is applied.
- If the water level has reached Layer 2, the value applied is the Layer 1 FLC plus a fraction of the Layer 2 FLC based on the depth of water within Layer 2. For example, if Layer 1 FLC is 0.1 and Layer 2 is 0.5, and the water is 40% of the way up Layer 2, the FLC applied is $0.1 + 0.4 \times 0.5 = 0.3$.
- Similarly, if the water level is into Layer 3, the FLC is the Layer 1 FLC plus the Layer 2 FLC plus a fraction of the Layer 3 FLC.
- Once the water level is above Layer 3, ie. is into Layer 4, the FLC is held constant at the sum of the FLCs for Layers 1 to 3.

As described in Section 4.4.11, the point objects can be placed in a separate layer. In this case, four attributes are required as outlined in Table 4.22. Using points allows the three layers to vary in depth and height within the one shape (eg. to model an arch bridge or a sloping deck).

Table 4.21 Layered Flow Constriction Shape (2d_lfcsh) Attribute Descriptions

GIS Attribute	Description	Type
Invert	Invert of constriction (m above datum). To leave the Zpt levels unchanged (ie. use the existing Zpt elevations), enter a value of 99999.	Float
dZ	Performs same function as described for dZ in Table 4.8 and is applied to the Invert elevation values.	Float
Shape_Width_or_dMax	Same as for the same attribute in Table 4.8.	Float
Shape_Options	Point: Not used. Line or Polygon: MAX, RIDGE or RAISE: Only changes a Zpt (Invert) elevation if the Z Shape elevation at the Zpt is higher. MIN, GULLY or LOWER: Only changes a Zpt (Invert) elevation if the Z Shape elevation at the Zpt is lower. Line: NO MERGE: For thin lines (Shape_Width_or_dMax = 0), the final elevations along the line are as specified. If NO MERGE is not specified for a thin line, the final elevations are set to be the same as the current Zpt values plus the dZ value. TIN: Indicates the line is to only be used for generation of TINs within polygons (only sections of TIN lines that fall within polygons are used). Polygon: If none of the options below are specified, the Invert elevations at perimeter vertices that do not have an elevation point snapped to them are merged with the current Zpt values (provided Invert does not equal 99999). MERGE ALL: Ignores invert elevations from any points snapped to the perimeter and merges all perimeter vertices with the current Zpt values. NO MERGE: Does not merge the perimeter elevations with the current Zpt values.	Char(20)
L1_Obvert	The obvert (soffit) of Layer 1.	Float
L1_pBlockage	The percentage blockage of Layer 1. For example, enter 5 for a blockage of 5%.	Float
L1_FLC	Layer 1 form loss coefficient. See notes for FLC values for attribute FLC_below_Obvert in Table 4.19.	Float
L2_Depth	The depth in metres of Layer 2.	Float
L2_pBlockage	The percentage blockage of Layer 2. For example, enter 100 for a solid obstruction such as a bridge deck or pipe.	Float

GIS Attribute	Description	Type
L2_FLC	Layer 2 form loss coefficient. See notes for FLC values for attribute FLC_below_Obvert in Table 4.19.	Float
L3_Depth	The depth in metres of Layer 3.	Float
L3_pBlockage	The percentage blockage of Layer 3.	Float
L3_FLC	Layer 3 form loss coefficient. See notes for FLC values for attribute FLC_below_Obvert in Table 4.19.	Float

Table 4.22 Layered Flow Constriction Shape (2d_lfcsh..._pts) Attribute Descriptions

GIS Attribute	Description	Type
Invert	Invert of constriction (m above datum). To leave the Zpt levels unchanged (ie. use the existing Zpt elevations), enter a value of 99999.	Float
L1_Obvert	The obvert (soffit) of Layer 1.	Float
L2_Depth	The depth in metres of Layer 2.	Float
L3_Depth	The depth in metres of Layer 3.	Float

4.7.3 2D Upstream Controlled Flow (Weirs and Supercritical Flow)

Where flow in the 2D domain becomes upstream controlled, TUFLOW automatically switches between either weir flow and/or upstream controlled friction flow.

If [Supercritical](#) is set to ON (the default as of Build 2002-11-AD) the following rules apply. Note: the bed slope at ZU and ZV points is determined as the slope from the **upstream ZC** point to the ZU or ZV point in the direction of **positive** flow.

- Where the bed slope at a ZU or ZV point is in the same direction as the water surface slope, tests are carried out to determine whether the flow is upstream controlled or downstream controlled. The adopted flow regime is determined by comparing the upstream and downstream controlled regime flows (preference to the lower flow) and whether the Froude No exceeds 1 (unless changed by [Froude Check](#)). The equation used for upstream controlled flow is the Manning equation with the water surface slope set to the bed slope. The Froude No check was introduced at Build 2002-11-AD – models using upstream controlled flow switch prior to this build can use the “PRE 2002-11-AD” switch for [Supercritical](#). It is recommended that the Froude No check be used (which is the default setting from Build 2002-11-AD onwards) as it provides more accurate switching. A further check was incorporated in Build 2003-01-AF that phases out the Froude Check as the water surface approaches the horizontal (otherwise in some situations, the flow would remain in the upstream controlled regime). This check can be disabled for backward compatibility using [Froude Depth Adjustment](#).
- Weir flow only occurs if the bed slope is adverse (different direction) to the water surface slope. Weir flow across 2D cell sides is modelled by first testing whether the flow is upstream or downstream controlled. If upstream controlled, the broad-crested weir flow equation is used to replace the calculations for downstream controlled (sub-critical) flow conditions. Weir flow may be switched off using the [Free Overfall](#) options.

TUFLOW produces an increase in water level at transitions from supercritical flow to subcritical flow as occurs with a hydraulic jump. It does not, however, model the complex 3D flow patterns that occur at a hydraulic jump, as it uses a 2D horizontal plane solution. Results in areas of transition should be interpreted with caution. It is also important to be careful presenting results in areas of supercritical flow as complex flows (such as surcharging against a house) may occur that would yield higher localised water levels – it is good practice to also view the energy levels when providing advice on flood planning levels.

If [Supercritical](#) is set to OFF, and [Free Overfall](#) is set to ON (the default), weir flow may occur on both adverse and normal bed slopes.

The weir flow switch may be varied spatially over the grid by setting a weir factor of zero where there is to be no automatic weir flow using [Read GIS WrF](#). The weir factor also allows calibration or adjustment where the broad-crested weir equation is applied. The broad-crested weir equation is divided by the weir factor. Therefore, a factor of 1.0 represents no adjustment, while a factor greater than one will decrease the flow efficiency. Note: the weir factor is not the broad-crested weir coefficient. For further information, refer to Syme 2001b.

4.7.4 1D Hydraulic Structures

Hydraulic structures in the 1D domains are modelled by replacing the momentum equation with standard equations describing the flow through the structure. The basic structures available are listed below and described in the following sections.

A channel is flagged as a hydraulic structure using the Channel_Type attribute (see Table 4.14) as described in Table 4.13. Except for culverts, a structure has zero length, ie. there is no bed resistance.

4.7.4.1 Adjustment of Contraction and Expansion Losses

As of Build 2005-05-AN, a new feature allows the energy losses, associated with the contraction and expansion of flow lines into and out of a structure, to be automatically adjusted according to the approach and departure velocities in the upstream and downstream channels. This is particularly important where:

- There is no change in velocity magnitude and direction as water flows through a structure. Examples are:
 - A clear spanning bridge over a stormwater channel where there are no losses due to any obstruction to flow until the bridge deck becomes surcharged.
 - Flow from one pipe to another where the pipe size remains unchanged and there is no significant bend or change in grade.
- There is a change in velocity, but the change does not warrant application of the full entrance and exit loss.

For bridges and culverts, if [Structure Losses](#) is set to ADJUST, or “A” has been specified in the 1d_nwk Channel_Type attribute (see Table 4.13), the entrance and exit losses are adjusted according to the equations below.

If [Structure Losses](#) == ADJUST EXCEPT BG TABLES (new option and the default setting for the 2010-10 release), the adjustment of the losses is not carried out for manually specified BG (or LC) tables for bridges. This change is because BG energy loss tables are most often based on coefficients from publications such as Hydraulics of Bridge Waterways (US FHA, 1973) that have already taken into account the effects of approach and departure velocities.

$$C_{entrance\_adjusted} = C_{entrance} \left[1 - \frac{V_{approach}}{V_{structure}} \right]$$

$$C_{exit\_adjusted} = C_{exit} \left[1 - \frac{V_{departure}}{V_{structure}} \right]^2$$

where

V = Velocity

C = Loss Coefficient

As the entrance and exit losses are combined as one coefficient for a bridge, the entrance and exit loss adjustments are proportioned one-third / two-thirds respectively for bridge channels.

As the structure velocity approaches the approach and/or departure velocities, the loss coefficient approaches zero. While, when the approach and/or departure velocity approaches zero (ie. water is leaving/entering a large body of water), the loss coefficients approach their full value.

Tullis and Robinson (2008) provides an excellent proof for the need to adjust losses for different flow regimes using the exit loss equation above. Their paper benchmarks different exit loss equations used within the industry methods against experimental flume test results.

The selection of the upstream and downstream channels on which to base the approach and departure velocities is as follows:

- The upstream channel is determined as the channel, which has a positive flow direction into the structure, that's invert is closest to that of the upstream invert of the structure. If no channel exists, no adjustment of losses is made (this includes structures connected to a 2D domain). Note that the upstream channel must be digitised so that it has the same positive flow direction to that of the structure.
- The downstream channel is selected on a similar basis to that for the upstream channel.
- The selected upstream and downstream channels are listed in the .eof file for cross-checking (search for "Primary Channel").

TUFLOW has no requirement of a minimum loss coefficient value for stability, and therefore allows the adjusted coefficient to approach zero. This feature correctly models the structure losses when the structure causes no disturbance to flow, or when one pipe discharges into another of identical size, grade and alignment.

The adjustment of loss coefficients does not apply to:

- Any bend or additional loss for a culvert entered using the Form_or_Bend_Loss attribute in the 1d_nwk layer. This coefficient can be used to apply additional losses (eg. pit or bend losses) that are not affected by changes in the relativity of the approach/departure and structure velocities.
- Any additional loss coefficient component of a bridge entered using the Form_or_Bend_Loss attribute in the 1d_nwk layer. This coefficient can be used to apply additional losses (eg. pier losses) that are not affected by changes in the relativity of the approach/departure and structure velocities.

If [Structure Losses](#) is set to FIX, or "F" has been specified in the 1d_nwk Channel_Type attribute (see Table 4.13), the loss coefficients are not adjusted.

As of Build 2008-08-AC, the variation in time of the loss coefficients can be viewed using the _TSL output layer (see Section 7.4.4.1).

If there is a manhole at the culvert end, a manhole energy loss approach (see Section 4.5.2) is used instead of the culverts contraction/expansion loss, and the above description does not apply.

4.7.4.2 Bridges

Bridge channels do not require data for length, Manning's n , divergence or bed slope (they are effectively zero-length channels, although the length is used for automatically determining nodal storages – see Section 4.6.2.2). The bridge opening cross section is described in the same manner to a normal channel.

As of Build 2005-05-AN, two bridge solutions are offered using [Bridge Flow](#). Method B is an enhancement on Method A by providing better stability at shallow depths or when wetting and drying. There are also some subtle differences between the methods in how the loss coefficients are applied at the bridge deck as discussed further below.

Bridge structures are modelled using a height varying form loss coefficient rather than fixed contraction and expansion losses. A table (referred to as a BG Table) of backwater or form loss coefficient versus height is required. BG Tables can be entered using .csv files via a 1d_tab (often renamed as 1d_bg) layer (see Section 4.6.3). Alternatively, fixed field formatting can be used anywhere in the .ecf file using [BG Data](#) (see Section F.1 for the fixed field formatting rules). Where the loss coefficient is constant through to the bridge deck (eg. no losses such as a clear spanning bridge, or pier losses only), the BG table can automatically be created by specifying a positive non-zero value for the Form_or_Bend_Loss attribute in the 1d_nwk layer (see Table 4.14). Also note the use of this attribute in applying additional losses to a BG table.

The coefficients may be obtained from publications such as “Hydraulics of Bridge Waterways” (US FHA 1973), through the following procedure. The bridge opening ratio (stream constriction ratio), defined in Equations 1 and 2 of “Hydraulics of Bridge Waterways”, is estimated for various water levels from the local geometry. Alternatively, the bridge opening ratio is estimated with the help of a trial modelling run in which the stream crossed by the bridge is represented by a number of parallel channels, providing a more quantitative basis for estimating the proportion of flow actually obstructed by the bridge abutments. For each level this enables the value of K_b to be obtained from Figure 6 of “Hydraulics of Bridge Waterways”. Additional factors, for piers (K_p from Figure 7), eccentricity (K_e from Figure 8) and for skew (K_s from Figure 10) are obtained. The backwater coefficient input into the table is the sum of the relevant coefficients at each elevation. The velocity through the bridge structure used for determining the head loss is based on the flow area calculated using the water level at the downstream node. Backwater coefficients derived in this manner have usually taken into account the effects of approach and departure velocities (usually via consideration of the upstream and downstream cross-section areas), in which case the losses for the B channel should be fixed (specify “BF” for the 1d_nwk Type attribute, or use [Structure Losses](#) == ADJUST EXCEPT BG TABLES for the 2010-10 release or later – see Section 4.7.4.1)

For Method A, the underside of the bridge deck (the obvert) is taken as the elevation when the flow area stops increasing, or the highest elevation in the bridge's CS data, whichever occurs first. For Method B, the highest (last) elevation in the CS table is always assumed to be the underside of the bridge deck.

For Method A, once the downstream water level is within 10% of the flow depth under the bridge, a bridge deck submergence factor is phased in by applying a correction for submerged decking using a minimum value of 1.5625 (if the specified loss coefficient is greater than 1.5625, this value is applied). Method B does not use the 10% of the flow depth phasing in nor applies a minimum loss coefficient

once the bridge deck is submerged (ie. it applies the value as per the specified loss coefficients (BG) table). Method B relies on the user to provide appropriate values at all flow heights.

A calibration factor is available for bridges. For a given flow the backwater (head increment) of a bridge channel is proportional to the factor. It is normally set to 1.0 by default, and modified if required for calibration purposes. This option is presently only available if using the old fixed field input BG tables – see previous manuals, but is planned to be made available via the `1d_nwk` layer in a future release.

Any wetted perimeter or Manning's *n* inputs in the hydraulic properties table are ignored. If the flow is expected to overtop the bridge, a parallel weir channel should be included to represent the flow over the bridge deck, or a BW channel can be specified.

4.7.4.3 Culverts and Pipes

Culvert channels can be either rectangular, circular (pipe) or irregular (as of Build 2008-08-AA) in shape. A range of different flow regimes is simulated with flow in either direction. Adverse slopes are accounted for and flow may be subcritical or supercritical. Figure 4-6, Figure 4-7 and Table 4.23 present the different flow regimes modelled. The regimes are output to the `.eof` file next to the velocity and flow output values, and to the `_TSF` GIS layer (see Sections 7.4.1 and 7.4.4).

The culvert type (“C” for circular, or “R” for rectangular), dimensions, length, upstream and downstream inverts, Manning's *n*, bend loss (as of Build 2005-05-AN), entrance and exit losses and number of barrels are entered using the `1d_nwk` attributes (see Table 4.14).

For an “I” (Irregular) shaped culvert, the cross-section shape is specified in the same manner as for open channels (ie. using a XZ or HW table - see Section 4.6.1 and 4.6.3, and Table 4.17). For the culvert coefficients and number of barrels use the `1d_nwk` attributes (see Table 4.14). Build 2008-08-AC incorporates major bug fixes for irregular culverts.

The four coefficients are as follows:

- The height contraction coefficient for box culverts, and is usually 0.6 for square edged entrances to 0.8 for rounded edges. This factor is not used for circular culverts.
- The width contraction coefficient for box culverts, with values from 0.9 for sharp edges to 1.0 for rounded edges. This factor is normally set to 1.0 for circular culverts.
- The general entry loss coefficient as specified by the manufacturer. The recommended value is 0.5.
- The exit loss coefficient, normally recommended as 1.0. (Note: This value when entered in fixed field formats in earlier versions of ESTRY is an exit recovery coefficient. To convert the exit recovery coefficient to an exit loss coefficient, multiply it by -1 and add 1, ie. 0 becomes 1 and 1 becomes 0. Its recommended value was zero.)

The calculations of culvert flow and losses are carried out using techniques from “Hydraulic Charts for the Selection of Highway Culverts” and “Capacity Charts for the Hydraulic Design of Highway Culverts”, together with additional information provided in Henderson 1966. The calculations have been compared and shown to be consistent with manufacturer's data provided by both “Rocla” and “Armco”.

Further improvements for calculating culvert flows were incorporated during the 2002-07/08 and 2002-12 builds. The improvements extend the original code to include two new regimes (K and L), regime B for circular culverts, smoother transitioning between flow regimes, better stability and correction of mass errors in rare flow situations. This approach is referred to as Method B, whilst the original approach is Method A. The method is set using [Culvert Flow](#). **As of Build 2002-08-AD, the default method is Method B. Prior to this build the default method was Method A. Note: Method B may still be subject to further enhancements.**

Build 2007-07-AA incorporates [Culvert Flow](#) == Method D, which offers further improvements as discussed below. For backward compatibility options see [Culvert Flow](#), [Culvert Critical H/D](#) and [Culvert Add Dynamic Head](#).

- Culvert entrance/exit velocities are retained for both upstream and downstream controlled flow conditions to assist in the calculations for the next half timestep (previously the velocities for the prevailing regime were only retained). This feature improves numerical stability when transitioning between upstream and downstream controlled flow regimes.
- Much improved convergence for Regime C, along with several tests for non-convergence. If convergence for Regime C is not achieved, inlet control is assumed. This improvement enhances the numerical stability of Regime C and when transitioning into or out of Regime C.
- Regime D included for zero length channels (usually pit channels). Previously only critical and fully submerged conditions applied – testing shows improved results at a pit as surcharging starts.
- Uses the new [Culvert Critical H/D](#) == OFF default setting.
- The previous benefits of [Culvert Add Dynamic Head](#) for [Culvert Flow](#) == Method C no longer apply and [Culvert Add Dynamic Head](#) by default is now set to OFF (as of Build 2007-07-AC).
- Includes a bug fix for Regime E that incorrectly would adjust the exit loss coefficient if [Structure Losses](#) == ADJUST. The bug would result in excessive increases in flow through the culvert whilst flowing in Regime E. Note that using the backward compatibility switch [Defaults](#) == PRE 2007-07-AA, reinstates this bug unless [Culvert Flow](#) == Method D is set.

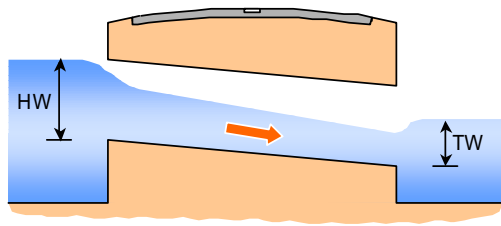
Build 2008-08-AA incorporates minor improvements for transitioning between Regimes A and B, and between inlet and outlet controlled regimes, for circular culverts. [Culvert Flow](#) == Method E is the default as of Build 2008-08-AA. If [Defaults](#) == PRE 2008-08 is specified, Method D is the default.

For benchmarking of culvert flow to the literature, see Huxley, 2004, which is available at:

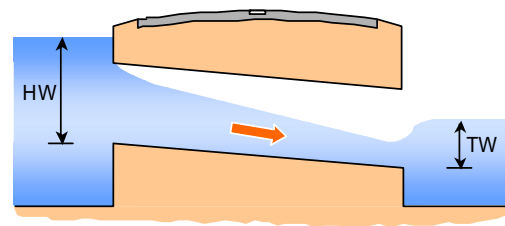
<http://www.tuflow.com/Downloads/TUFLOW%20Validation%20and%20Testing,%20Huxley,%202004.pdf>.

Table 4.23 1D Culvert Flow Regimes

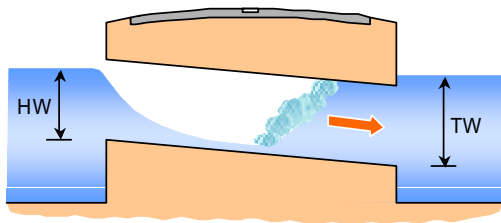
Regime	Description
A	Unsubmerged entrance and exit. Critical flow at entrance. Upstream controlled with the flow control at the inlet.
B	Submerged entrance and unsubmerged exit. Orifice flow at entrance. Upstream controlled with the flow control at the inlet. For circular culverts, not available for Culvert Flow == Method A.
C	Unsubmerged entrance and exit. Critical flow at exit. Upstream controlled with the flow control at the culvert outlet.
D	Unsubmerged entrance and exit. Sub-critical flow at exit. Downstream controlled.
E	Submerged entrance and unsubmerged exit. Full pipe flow. Upstream controlled with the flow control at the culvert outlet.
F	Submerged entrance and exit. Full pipe flow. Downstream controlled.
G	No flow. Dry or flap-gate active.
H	Submerged entrance and unsubmerged exit. Adverse slope. Downstream controlled.
J	Unsubmerged entrance and exit. Adverse slope. Downstream controlled.
K	Unsubmerged entrance and submerged exit. Critical flow at entrance. Upstream controlled with the flow control at the inlet. Hydraulic jump along culvert. Not available for Culvert Flow == Method A.
L	Submerged entrance and exit. Orifice flow at entrance. Upstream controlled with the flow control at the inlet. Hydraulic jump along culvert. Not available for Culvert Flow == Method A.

INLET CONTROL FLOW REGIMES

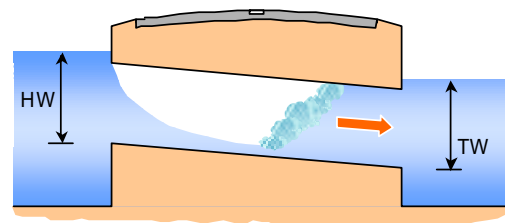
**A: Unsubmerged Entrance,
Supercritical Slope**



**B: Submerged Entrance,
Supercritical Slope**

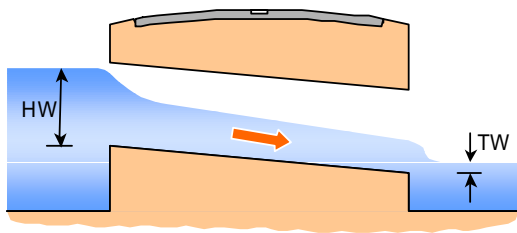
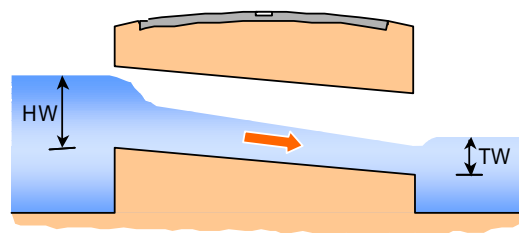
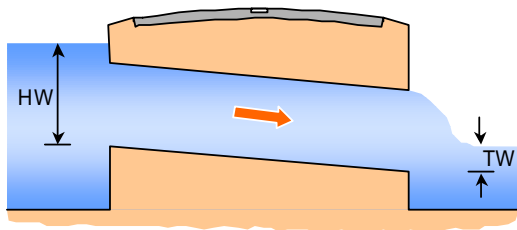
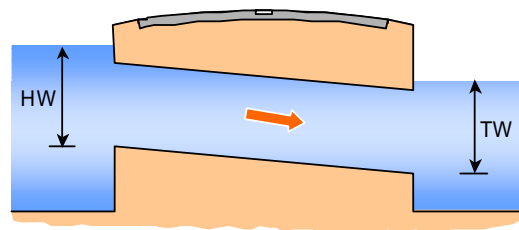
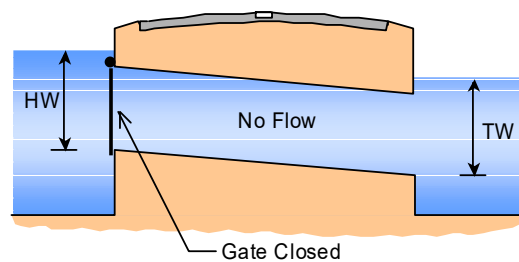
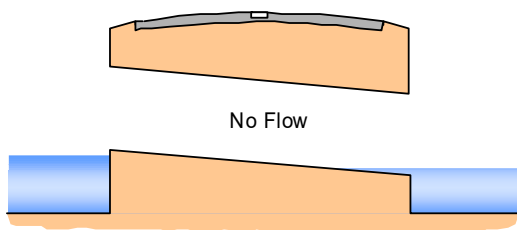
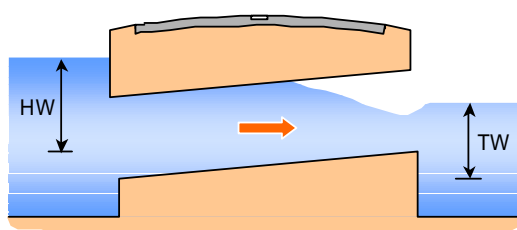
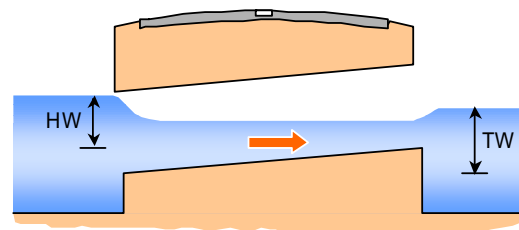


**K: Unsubmerged Entrance,
Submerged Exit
Critical at Entrance**



**L: Submerged Entrance,
Submerged Exit
Orifice Flow at Entrance**

Figure 4-6 1D Inlet Control Culvert Flow Regimes

OUTLET CONTROL FLOW REGIMES**C: Unsubmerged Entrance, Critical Exit****D: Unsubmerged Entrance, Subcritical Exit****E: Submerged Entrance, Unsubmerged Exit****F: Submerged Entrance, Submerged Exit****G: No Flow
Dry or Flap-Gate Closed****H: Adverse Slope, Submerged Entrance****J: Adverse Slope, Unsubmerged Entrance
(Critical or Subcritical at Exit)****Figure 4-7 1D Outlet Control Culvert Flow Regimes**

4.7.4.4 Weirs

Weir channels do not require data for length, Manning's n, divergence or bed slope (they are effectively zero-length channels, although the length is used for automatically determining nodal storages – see Section 4.6.2.2).

For weirs a standard weir flow formula is used as per the equation below. The weir is assumed to be broad-crested. A weir with significantly different characteristics can possibly be modelled by adjusting the weir calibration factor, C_f (see 1d_nwk Height_or_WF attribute in Table 4.14), or using a non-inertial channel with carefully selected section properties. Alternative weir types are planned for the 2011 release.

$$Q_{weir} = \frac{2}{3} C W \sqrt{\frac{2g}{C_f}} H^{3/2}$$

$$V_{approach} = \frac{2}{3} C \sqrt{\frac{2gH}{C_f}}$$

Q_{weir} = Flow over the weir (unsubmerged) m^3/s

$V_{approach}$ = Velocity approaching weir (m/s)

C = Broad-crested Weir Coefficient of 0.57

C_f = Weir calibration factor (default = 1.0) - see 1d_nwk Height_or_WF attribute

W = Flow width (m)

H = Depth of water approaching weir relative to weir invert (m)

The calibration factor, C_f , is available for modifying the flow. For a given approach velocity the backwater (head increment) of the weir channel is proportional to the inverse of the factor. It is normally set to 1.0 by default, and modified if required for calibration or other adjustment. Note, this factor is not the weir coefficient, rather a calibration factor to adjust the standard broad-crested weir equation. The factor can be used to model other types of weirs through adjustment of the broad-crested weir equation.

Transitioning to submerged flow occurs once the downstream depth (relative to the weir sill or invert) exceeds 0.7 times the upstream depth (H), reducing to zero flow when the depth difference is zero.

For benchmarking of unsubmerged and submerged weir flow to the literature, see Huxley, 2004, which is available at:

<http://www.tuflow.com/Downloads/TUFLOW%20Validation%20and%20Testing,%20Huxley,%202004.pdf>.

Weirs have three flow regimes of zero flow (dry), upstream controlled flow (unsubmerged) and downstream controlled flow (submerged). The weirs invert and a calibration factor are entered using the 1d_nwk attributes in Table 4.14.

Note that the velocity output for a weir is the approach velocity, $V_{approach}$, in the above equations, not the velocity at critical depth (when the flow is unsubmerged).

4.7.4.5 Variable Geometry Channels

A channel's cross-section geometry can be varied by setting it as a variable geometry channel (see Channel_Type "V" in Table 4.14). In addition, a VG Table in fixed field format is required. VG Tables are read using [VG Data](#) (see versions of the TUFLOW manual prior to 2007 downloadable from www.tuflow.com for the format).

4.7.4.6 Non-Inertial Channels

In order to bypass the Courant stability condition, a special channel type (N) is included, known as non-inertial channel or a friction-controlled channel. For this channel, the inertial terms are ignored (eliminating inertial effects) and the stability control procedure is automatically applied.

This channel type may be used to model flow areas not allowed for otherwise.

4.7.4.7 M Channels (User Defined Flow Matrix)

M channels were introduced for the 2009-07 release and allow the modeller to define the flow through a channel (usually a structure) based on a flow matrix specified by the user. To set up M channels follow the steps below.

- 1 In the 1d_nwk layer, the following attributes require values (the remaining attributes are not used):
 - (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 the 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 Cell 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.

	A	B	C	D	E	F	G	H	I	J
1	UD Channel Flow Matrix Example									
2	y2				y1					
3	1	0	0.2	0.3	0.4	0.5	0.6	0.7	0.8	
4	0	0	0.1	0.5	0.7	0.9	1.1	1.3	1.5	
5	0.2	-0.1	0	0.2	0.5	0.7	0.9	1.1	1.3	
6	0.3	-0.2	-0.1	0	0.2	0.5	0.7	0.9	1.1	
7	0.4	-0.3	-0.2	-0.1	0	0.2	0.5	0.7	0.9	
8	0.5	-0.4	-0.3	-0.2	-0.1	0	0.2	0.5	0.7	
9	0.6	-0.5	-0.4	-0.3	-0.2	-0.1	0	0.2	0.5	
10	0.7	-0.6	-0.5	-0.4	-0.3	-0.2	-0.1	0	0.2	
11	0.8	-0.7	-0.6	-0.5	-0.4	-0.3	-0.2	-0.1	0	
12										

4.7.4.8 WV Channels (Variable Geometry Weir)

The V option for W (weir) channels allows the modeller to vary the cross-section geometry of a weir over time using a trapezoidal shape. To set up a WV channel follow the steps below. The WV option was introduced in the 2009-07 Release.

- 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.

	A	B	C	D	E
	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					

4.8 Customised Output

4.8.1 Time-Series Output (2d_po, 2d_lp) from 2D Domains

Time-series data output from 2D domains is available for a range of hydraulic parameters as listed in Table 4.24. Output takes the form of time-series hydrographs (referred to as PO – Plot Output) or longitudinal profiles over time (LP). All data types are available for PO and only H₁ (water level) is available for LP.

The locations of PO and LP output must be defined prior to a simulation. This is carried out by creating one or more GIS layers containing points, lines and polylines that define the locations of PO and LP output.

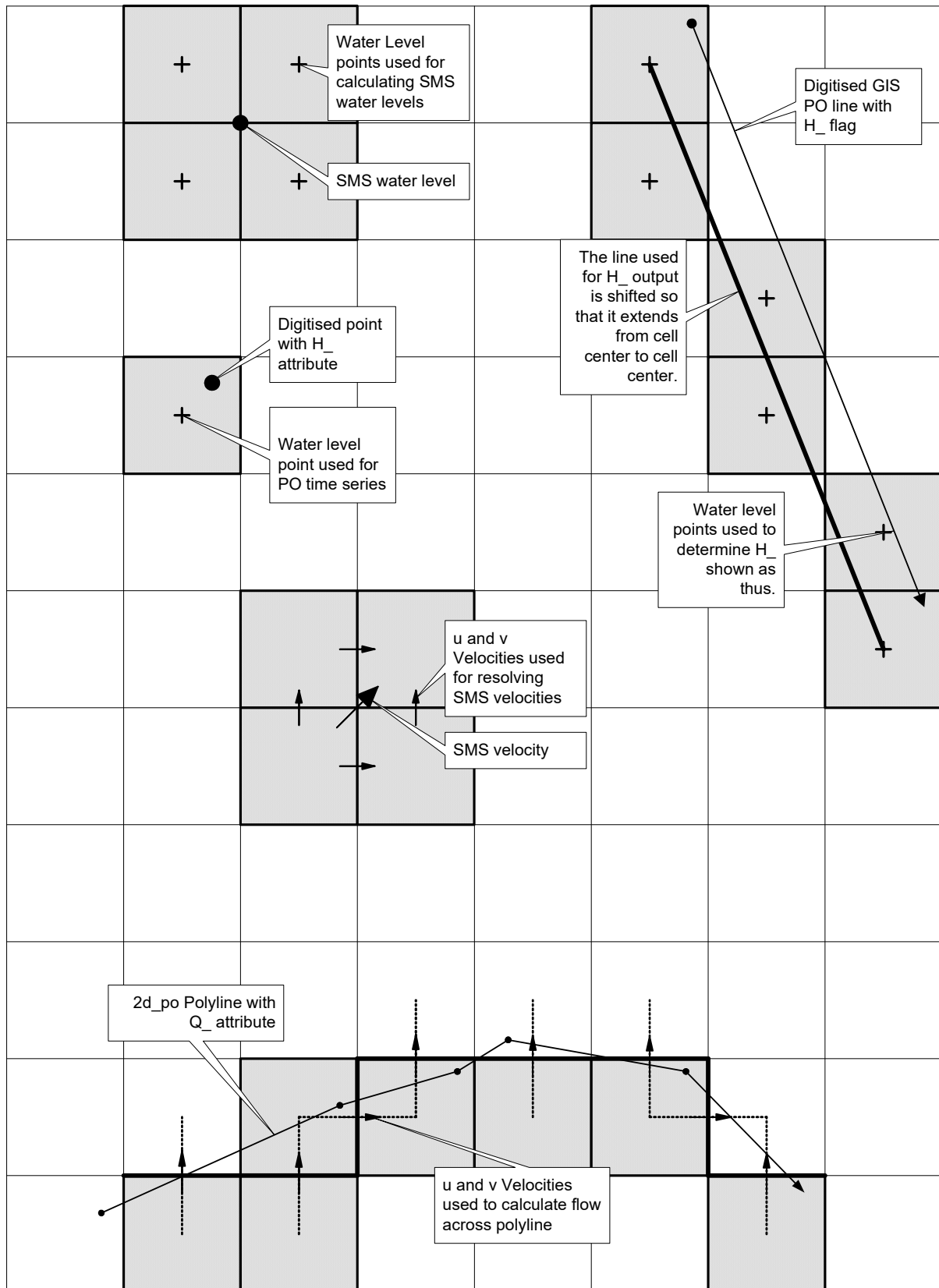


Figure 4-8 illustrates how 2d_po objects are interpreted.

The start time for PO and LP output and the output interval is set separately to map based output using [Start Time Series Output](#) and [Time Series Output Interval](#). If no start time or interval is set, output occurs from the beginning of the simulation at every timestep.

The output is in the form a .csv files and also to the _TS layer (as of Build 2003-06-AA). Refer to Section 7.3.2. As of Build 2003-06-AA, 2D domain time-series (PO) output is synchronised with 1D domain output by default. This allows both 1D and 2D time-series to be placed in the _TS layer. Set [Output Times Same as 2D](#) to OFF in the .ecf file if 1D and 2D time-series data is not to be synchronised. In this case, no 2D PO is written to the _TS layer.

Table 4.25 describes the GIS attributes. Of note is for **flow** flags, TUFLOW sums time-series with the same label (this does not apply if the label is left blank).

For the 2010-10 release the accuracy of the 2d_po output .csv files was increased to four (4) decimal places.

Note: If the total flow across the floodplain is required for a 1D/2D model where the river is in 1D and the floodplain in 2D, it is important to digitise the 2d_po Q_ lines so that they cross the HX line at the point where there is a change in 2D HX cell colour in the 1d_to_2d_check layer. The change in cell colour (usually halfway between the CN lines connecting the 1D nodes to the HX line) indicates a change in the 1D node for transferring mass between 1D nodes and 2D HX cells.

4.8.2 Evacuation Routes (2d_zshr)

Evacuation routes can be specified to provide output on:

- the degree of inundation of routes; and
- for helping define
 - evacuation routes;
 - warning times;
 - risks; and
 - duration that routes are cut off.

The.tgc [Read GIS Z Shape Route](#) is used to define the routes and, by default, also adjust the Zpt elevations along the route using the standard Z line options.

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

- Route_Name Char(40)
- Cut_Off_Type Char(40)
- Cut_Off_Values Char(80)

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.

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) applied at that point in the .tgc file.

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 must 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 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).

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 as described in Section 4.4.9.

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 GIS Z Shape Route](#) commands in the .tgc file by repeat usage of the same commands if the user so desires.

- [Set Route Cut Off Values](#)
- [Set Route Cut Off Type](#)

The _RCP 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:

- Route_Name (ie. the name of the route)
- Cut_Off_Value (ie. the cut off value applied)
- First_Cut_Off_Time (the simulation time in hours the Cut_Off_Value was first exceeded)
- Last_Cut_Off_Time Float (the simulation time in hours when the cut off value was last exceeded)
- Duration_Cut_Off (the 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)

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

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. This allows for a Z Shape Route Line to cross a river without physically blocking it! Because a route can be comprised of more than one line (as long as each line is given the same Route_Name) they will all be regarded as being part of the same route. Therefore to represent a bridge, split the route line either side of the bridge and give all three lines the same Route Name, with the line representing the bridge or river crossing assigned a Shape_Option of "BRIDGE".

Table 4.24 Time-Series (PO) Data Types

Flag	Description	Point	Line (or Polyline)
H_	Water Level (Head)	Water level of the h-point of the nearest cell. If the cell is dry, the ground level (ZC) is output.	The average water level of all wet cells along the line. If all cells are dry, the lowest cell's ground level (ZC) is output. If a polyline is used, the average water level along each line segment is output.

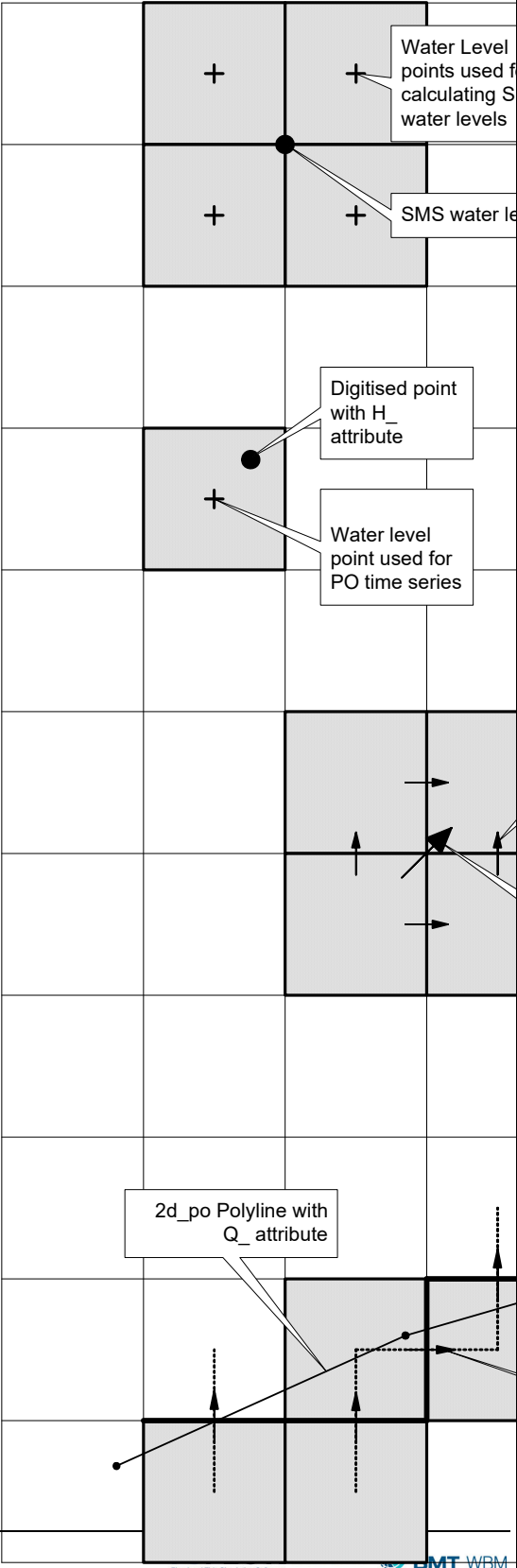
Flag	Description	Point	Line (or Polyline)
Q_	Flow or discharge.	N/A (zero flow results).	The flow crossing the line. For a polyline, the sum of the flows crossing each polyline segment. The flow across a line or polyline segment is determined by summing the flow across cell sides whose perpendiculars' intersect the line (see 

Figure 4-8).

The sign of the flow across a line or polyline segment is positive if the flow is in the same

Flag	Description	Point	Line (or Polyline)
QA	Flow Area (m ²).	N/A (zero area results).	The flow area is calculated using the same cell sides as for Q ₋ . An adjustment for oblique lines is made.
QI	Integral Flow (m ³)	N/A (zero integral flow results).	Integrates the flow (as determined for Q ₋ above) over time (ie. the area under a Q ₋ time-series curve). If Write PO Online is set to ON, the integral flow is not calculated until the simulation is complete.
QX	Flow in X-direction.	N/A (zero flow results).	The X component of Q ₋ (ie. the sum of the flows at the u-points).
QY	Flow in Y-direction.	N/A (zero flow results).	The Y component of Q ₋ (ie. the sum of the flows at the v-points).
V ₋	Velocity (m/s)	The magnitude of the resolved vector based on the two u-points and two v-points of the cell in which the point falls. Exactly which cell is selected may be uncertain if the point falls exactly on a cell's side.	N/A. Do not use lines or polylines for velocity output. At present uses the cell in which the line or each polyline segment starts. Future release plan to calculate Q ₋ and QA, and output the velocity as Q ₋ /QA (ie. the depth and width averaged velocity along the line).
VA	Velocity Angle (degrees relative to east where east is zero, north is 90, etc.).	The angle of V ₋ .	N/A. See comments above for V ₋ .
Vu	u-point velocity (m/s)	The magnitude of the u-point velocity (ie. across the right hand side of the cell).	N/A. See comments above for V ₋ .
Vv	v-point velocity (m/s)	The magnitude of the v-point velocity (ie. across the top side of the cell).	N/A. See comments above for V ₋ .
VX	Velocity in X-direction (m/s).	The magnitude of the average of the u-point velocities (ie. across the left and right hand sides of the cell).	N/A. See comments above for V ₋ .

Flag	Description	Point	Line (or Polyline)
VY	Velocity in Y-direction (m/s).	The magnitude of the average of the v-point velocities (ie. across the bottom and top sides of the cell).	N/A. See comments above for V_.
W_	Wind (undocumented and yet to be tested feature on PC version).		

Table 4.25 Plot Output (2d_po) Attribute Descriptions

GIS Attribute	Description	Cols in Text File	Type
N/A	Flag Identifier “PO”	01-02	T
N/A	N grid coord	11-15	I
N/A	M grid coord	16-20	I
N/A	Second N grid coord (if a line, otherwise blank)	21-25	I
N/A	Second M grid coord (if a line, otherwise blank)	26-30	I
type	Any combination of the two letter flags listed in Column 1 of Table 4.24 (limit of 10 flags per entry). In Version 3 or later, the flags are not case sensitive. Note, in Version 3 or later, “u ” has changed to “uu” and “v ” to “vv”. For example, to output water level and flow time-series for the same line, enter “H_Q_” for the type attribute of the line.	31-50	T
label	Label up to 30 characters defining the name of the time-series. The label appears at the top of the columns of data in the _PO.csv file. Spaces are permitted, but do not use commas. Note: If the same label occurs more than once for a flow output, the time-series are added together as one time-series. This allows a flow line that is discontinuous to be specified as a series of individual lines.	51-80	T

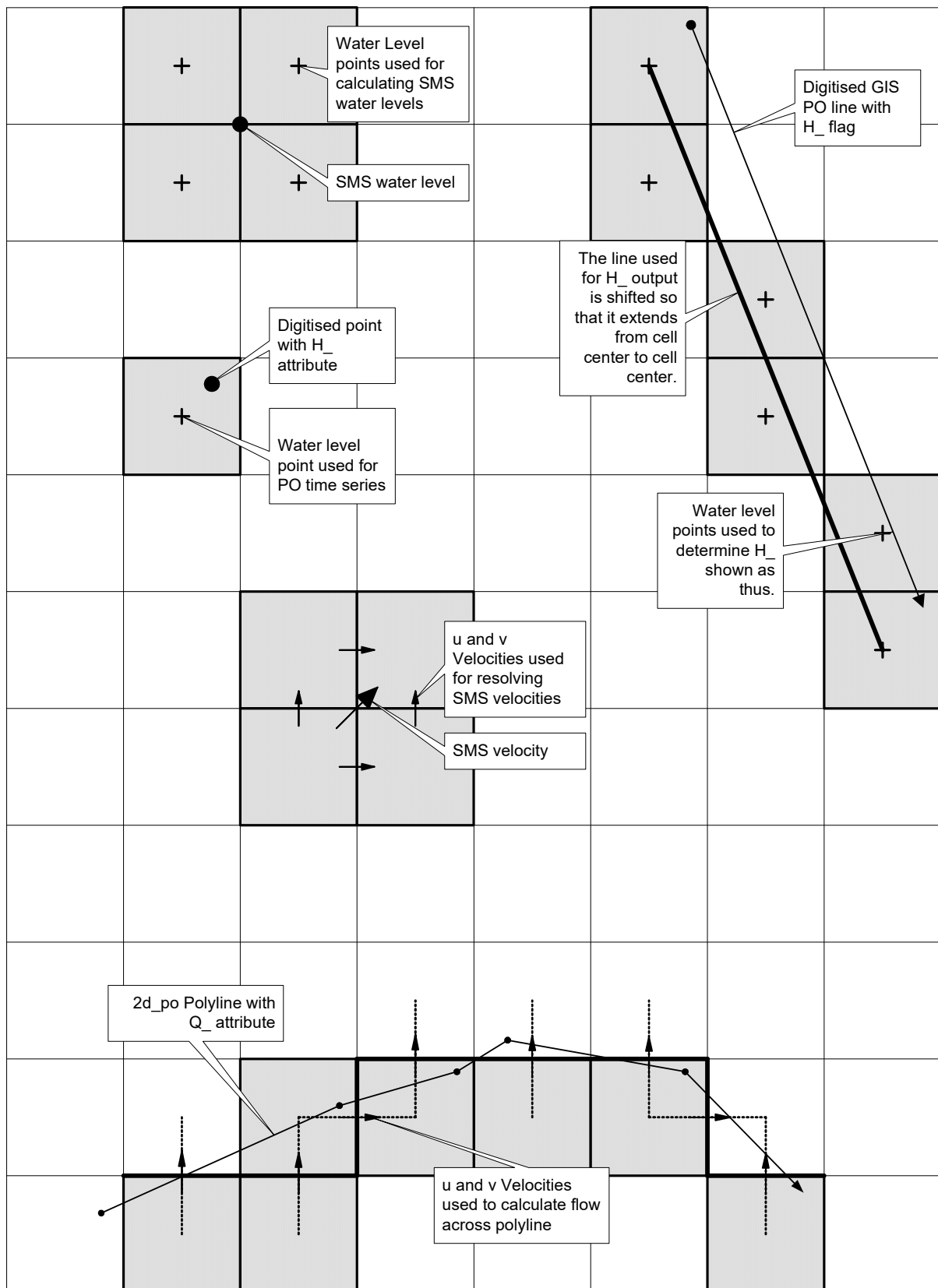


Figure 4-8 Interpretation of PO Objects and SMS Output

4.9 Initial Water Levels (IWL) and Restart Files

4.9.1 2D Domains

Initial water levels (IWL) are set globally as a constant using the [Set IWL](#) (.tcf file) or [Set IWL](#) (.tgc file) command. IWLs can also vary spatially using one or more GIS layers. This is particularly useful for setting initial water levels in lakes, dams, etc.

To set a gradually varying water surface, the best approach is to start the simulation “cold” or “dry” and create a restart file to set initial water levels and flow velocities – see [Write Restart File at Time](#), [Write Restart File Interval](#) and [Read Restart File](#).

The easiest way to set up a GIS IWL layer is to:

- 1 Create a 2d_iwl layer using an empty layer created by [Write Empty MI Files](#).
- 2 Digitise regions, lines or points and assign each object an initial water level value.
- 3 Export the GIS layer as a MIF/MID.
- 4 Use the [Read GIS](#) IWL command to read in the IWL values.

Alternatively, the [Read RowCol](#) command can be used as follows:

- 1 Select the relevant grid cells or ZC points from a 2d_grd or 2d_zpt GIS layer.
- 2 Save the selection as another layer named 2d_iwl_<name>.
- 3 Modify the 2d_iwl attributes (see Table 4.26) so that you:
 - (a) keep the first two columns as the row (n) and column (m) grid references;
 - (b) remove all other columns;
 - (c) add a (third) column as one named “IWL” defined as a float or decimal.
- 4 Using the GIS to set the IWL value(s) as required.
- 5 Other grid cells or ZC points can be copied and pasted into 2d_iwl if required and the IWL value(s) allocated.
- 6 Export the GIS layer as a MIF/MID.
- 7 Use the [Read RowCol](#) IWL command to read in the IWL values.

Any number of IWL layers may be used, noting that if a cell’s IWL occurs more than once, the last occurrence prevails, ie. TUFLOW overwrites any previous IWL already set.

The [Read GIS](#) IWL or [Read RowCol](#) IWL command maybe used in the .tcf and .tgc files, noting that the .tgc file is processed before the .tcf file (ie. any IWL commands in the .tcf file will override those in the .tgc file). Using the IWL command in the .tcf file allows the initial water levels to be set independently of the geometry file. This is useful where several simulations of different events use the same .tgc file but have different initial conditions, thereby removing the need to have separate geometry files for each event. IWL commands are 2D domain dependent. (Note: Prior to Build 2004-03-AA, the Read GIS IWL command was not available for the .tcf file.)

Table 4.26 2d_iwl Attributes

GIS Attribute	Description	Type
Read GIS IWL Command		
IWL	Initial water level of object relative to model datum (m).	Float
Read RowCol IWL Command		
N	Row of cell.	Integer
M	Column of cell.	Integer
IWL	Initial water level at cell relative to model datum (m).	Float

4.9.2 1D Domains

Similarly for 1D domains, initial water levels (IWL) are set globally as a constant using [Set IWL](#). IWLs can also vary spatially using one or more GIS layers. This is particularly useful for setting initial water levels in lakes, dams, etc. The default initial water level at 1D nodes is zero (0).

Note: The restart file feature is only available for 2D/1D models; it has yet to be implemented for 1D only models.

To set up a GIS IWL layer for the 1D domains:

- 1 Create a 1d_iwl layer using an empty layer created by [Write Empty MI Files](#).
- 2 Digitise points snapped to nodes and assign each point an initial water level value.
- 3 Export the GIS layer as a MIF/MID.
- 4 Use the [Read GIS IWL](#) command to read in the IWL values.

Any number of IWL layers may be used, noting that if a node's IWL occurs more than once, the last occurrence prevails, ie. TUFLOW or ESTRY overwrites any previous IWL already set.

At Build 2003-04-AA the initial water levels at nodes connected to a 2D SX link are set to the initial water levels of the 2D SX cells.

Table 4.27 1D Initial Water Level (1d_iwl) Attributes

GIS Attribute	Description	Type
Read GIS IWL Command		
IWL	Initial water level of object relative to model datum (m).	Float

4.10 Boundary Conditions and Linking 2D/1D Models

1D and 2D domains use the same approach to setting up boundary conditions. They both access the same boundary condition database, although separate databases can be set up if desired. They also use the same commands in the text files.

4.10.1 Boundary Condition (BC) Database

A boundary condition (BC) database is set up using spreadsheet software such as Microsoft Excel. Two types of files are required:

- 1 A database or list of BC events including information on where to find the BC data.
- 2 One or more files containing the BC data.

The database file must be .csv (comma delimited) formatted. It must contain a row with the pre-defined keywords `Name` and `Source` as listed in Table 4.28. Other keywords control how data is extracted from the source.

The BC data files can be in a variety of formats as described for the `Source` keyword in Table 4.28. Additional formats can be incorporated upon request. It is strongly recommended that all .csv files originate from one spreadsheet with a worksheet dedicated to each .csv file.

Both 1D and 2D domains can access the same files.

The active BC Database is specified using [BC Database](#) (.tcf file), [BC Database](#) (.ecf file) and/or [BC Database](#) (.tbc file). Note, specifying [BC Database](#) in the .tcf file automatically applies to both 1D and 2D domains (ie. there is no need to specify the command in the .ecf or .tbc files). The active database can be changed at any point by repeating the command in any of these files.

At Build 2008-08-AA, the maximum line length (ie. number of characters including spaces and tabs) in a source file was increased from 10,000 to 100,000 characters.

Table 4.28 BC Database Keyword Descriptions

Keyword	Description	Default Column ¹
Name	The name of a BC data event. The name must be the same name as used in the GIS 1d_bc and 2d_bc layers. It may contain spaces and other characters, but must not contain any commas. It is not case sensitive. As of Build 2005-05-AN, the name of a group of boundaries can be used for RAFTS (.loc and .tot files) and WBNM (via the .ts1 file format) hydrographs. For example, if “N1 Local” is the boundary Name in a 1d_bc or 2d_bc layer, then the group is interpreted as the text to the right of the symbol (ie. Local), and the text to the left is the ID (ie. N1) of the time-series data in the file containing the hydrographs. In this example, TUFLOW: • Searches for an entry “Local” in the Name column of the BC database. • Opens the file in the Source column, say Q100.ts1. • Extracts the hydrograph for node N1 from Q100.ts1. Using this approach the size of the BC Database .csv file can be reduced from several hundred lines for large hydrologic models to a couple of lines.	n/a
Source	The file from which to extract the BC data. Acceptable formats are: • Blank – if left blank, the BC data is assumed constant over time at the value specified under the Column 2 (Value) column (see Column 2 keyword below). • Comma delimited (.csv) files. Must have a .csv extension. • TUFLOW .ts1 time-series boundary data format (incorporated in Build 2003-06-AE). This format is fast to process and should be used for input of large numbers of hydrographs. See the convert_to_ts1.exe utility (Section 11.7) for converting output from RAFTS, RORB, URBS and WBNM to .ts1 format). • RAFTS-XP .tot and .loc files. As of Build 2003-09-AA, both 12 and 16 field output is supported. As of Build 2006-06-AA, 20 character field format recognised. • WBNM _Meta.out files. • ESTRY fixed field file containing boundary condition. Must be a .ecf, .ebc or .ecf extension. • TUFLOW fixed field file containing boundary condition. Must be a .tbc or .tef extension. • XP .int and .ext interface formats. • Other file formats are included upon request. Note, the type of file is determined by the extension, therefore, ensure the file has the correct extension.	n/a
Column 1 or Time	For .csv files, the name of the first column of data (usually time values) in the .csv Source File. Other examples besides Time are Flow for a HQ (stage-discharge) boundary, or Mean Water Level for each wave component in a 2D HS (sinusoidal wave) boundary.	3

Keyword	Description	Default Column ¹
	For all other types of Source entries, leave this field blank.	
Column 2 or Value or ID	For .csv files, the name of the second column of data in the .csv Source File. For example, water levels in a HT boundary. For a Blank Source entry, the constant value to be applied. For ESTRY fixed field boundary files (.eef or .ebc), the Node ID of the BC data. Note, the Node ID is limited to 5 characters. For TUFLOW fixed field boundary files (.tef or .tbc) the BC ID number. For RAFTS-XP (.tot or .loc), WBNM _Meta.out and TUFLOW/ESTRY .ts1 files, the name of the hydrograph location to extract. Note, from Build 2002-10-AJ, it is now NOT possible to combine the Value and ID keywords in the column label, for example “Value or ID” as shown in the example in Section 4.10.2. If they are combined, the default column number of 4 is used.	4
Add Col 1 or TimeAdd	An amount to add to all Column 1 (normally time) values (eg. a time shift) for the BC data event. If left blank or zero, there is no change to the time values. This field is ignored for Blank Source entries.	5
Mult Col 2 or ValueMult	A multiplication factor to apply to the Column 2 values. If left blank or one (1), there is no change to the values. Note, Mult Col 2 is applied before Add Col 2 below. This field is ignored for Blank Source entries.	6
Add Col 2 or ValueAdd	An amount to add to Column 2 values. If left blank or zero, there is no change to the values. Note, Add Col 2 is applied after Mult Col 2. This field is ignored for Blank Source entries.	7
Column 3	For .csv files, the name of the third column of data when a third column of data is required. For example, the phase difference for each wave component in a 2D HS (sinusoidal wave) boundary. For all other types of Source entries, leave this field blank.	8
Column 4	For .csv files, the name of the fourth column of data when a fourth column of data is required. For example, the period for each wave component in a 2D HS (sinusoidal wave) boundary. For all other types of Source entries, leave this field blank.	9

¹ If the keyword is not found in the “Name, Source” line, the default column is used to define the column of data for that keyword.

4.10.2 BC Database Example

The Excel spreadsheet below illustrates a simple example of a BC database setup in a worksheet that is exported as a .csv file for use by TUFLOW and/or ESTRY.

	A	B	C	D	E	F	G	H	I
	Name	Source	Column 1	Column 2	Add Col 1	Mult Col 2	Add Col 2	Column 3	Column 4
2	h=2	heads.csv	Time	h=2					
3	h=2 (alternative)			2					
4	River Inflow	flows.csv	Time 1	River Flow					
5	Creek Inflow	flows.csv	Time 1	Creek Flow					
6	Base Flow	flows.csv	Time 2	Base Flow					
7	RAFTS Inflow	rafts.tot		IN					
8									

TUFLOW or ESTRY search through the file until a row is found with the two keywords Name and Source. Name and Source do not have to be located in Columns 1 and 2.

Table 4.28 describes the purpose of each keyword and the default column where applicable. At present a range of formats are accepted, and other formats can be incorporated upon request.

The example above is interpreted as follows:

- A BC data event named “h=2” is located in the file heads.csv. The time values are located under a column called “Time” and the BC values are located under a column “h=2”.
- As an alternative to “h=2” above, a BC data event “h=2 (alternative)” is set a constant value of 2.
- “River Inflow” is located in flows.csv using time column “Time 1” and BC values from column “River Flow”. Similarly, “Creek Inflow” and “Base Flow” are also located in flows.csv.
- A BC data event named “RAFTS Inflow” extracts the hydrograph from a RAFTS-XP .tot file named “rafts.tot” for RAFTS node “IN”.

The heads.csv and flows.csv files are created by saving the worksheets “heads.csv” and “flows.csv” as .csv files (see below).

Microsoft Excel - bc dbase.xls

File Edit View Insert Format Tools Data Window Help

A8 =

	A	B	C	D	E	F	G	H	I
1	This worksheet contains water level boundary values								
2	Time	h=2							
3	0	2							
4	1	2							
5									
6									
7									
8									
9									

bc dbase.csv heads.csv flows.csv

Ready

Microsoft Excel - bc dbase.xls

File Edit View Insert Format Tools Data Window Help

E4 =

	A	B	C	D	E	F	G	H
1	Time 1	River Flow	Creek Flow		Time 2	Base Flow		
2	0	0	0		0	10		
3	3	50	10		1	10		
4	6	100	30					
5	7	110	35					
6	9	90	30					
7	12	60	20					
8	20	0	0					
9								

bc dbase.csv heads.csv flows.csv

Ready

4.10.3 Using the BC Event Name Command

As of the 2010-10 release, the ability to model multiple events using an [Event File](#) .tcf file offers even more power and flexibility to using [BC Event Text](#) and [BC Event Name](#) as discussed below. See Section 4.12.1 for details.

The [BC Event Text](#) and [BC Event Name](#) commands (.tcf file) minimise data repetition by removing the need to create a separate BC database .csv file for each BC event. These commands are also available in the .ecf file for 1D only models ([BC Event Text](#) and [BC Event Name](#)) and .tbc file ([BC Event Text](#) and [BC Event Name](#)).

How the commands work is illustrated in the example below.

A BC database file worksheet is created as illustrated below and the following lines occur in all .tcf files. (**Tip:** Specify these lines in a separate file and use the [Read File](#) command in all the .tcf files to read these commands. This saves repeating these lines, and other commands common to all .tcf files.)

```
BC Database == ..\bc_dbase\PR_bc_dbase.csv
BC Event Text == __event__
```

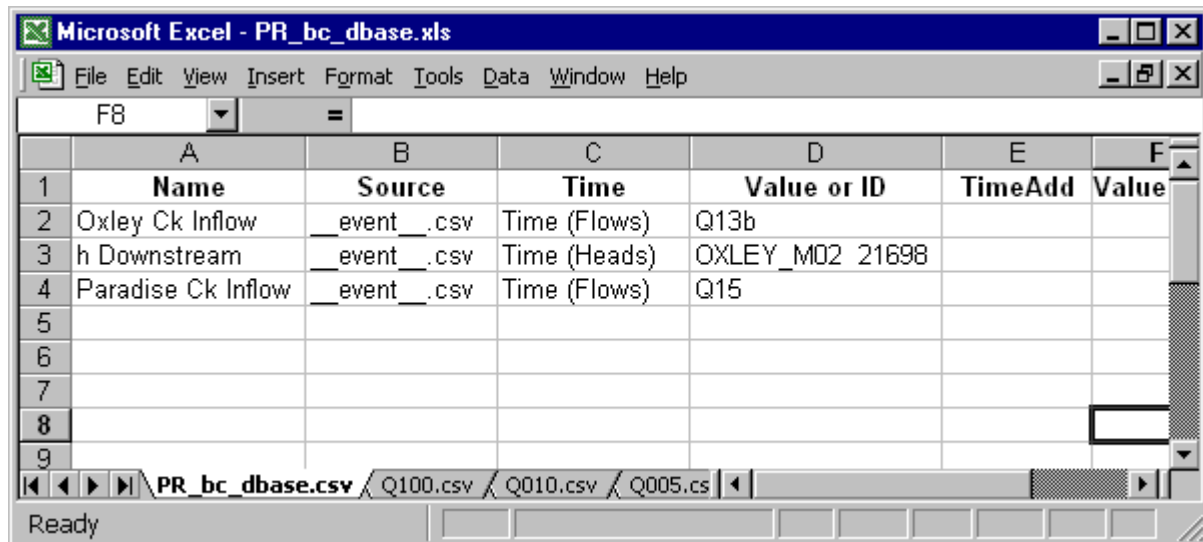
The above commands set the active BC Database for TUFLOW and ESTRY to ..\bc_dbase\PR_bc_dbase.csv, and defines the text “__event__” as that which defines the BC event name as discussed below.

In the .tcf file for the 100 year flood simulation, the following command occurs:

```
BC Event Name == Q100
```

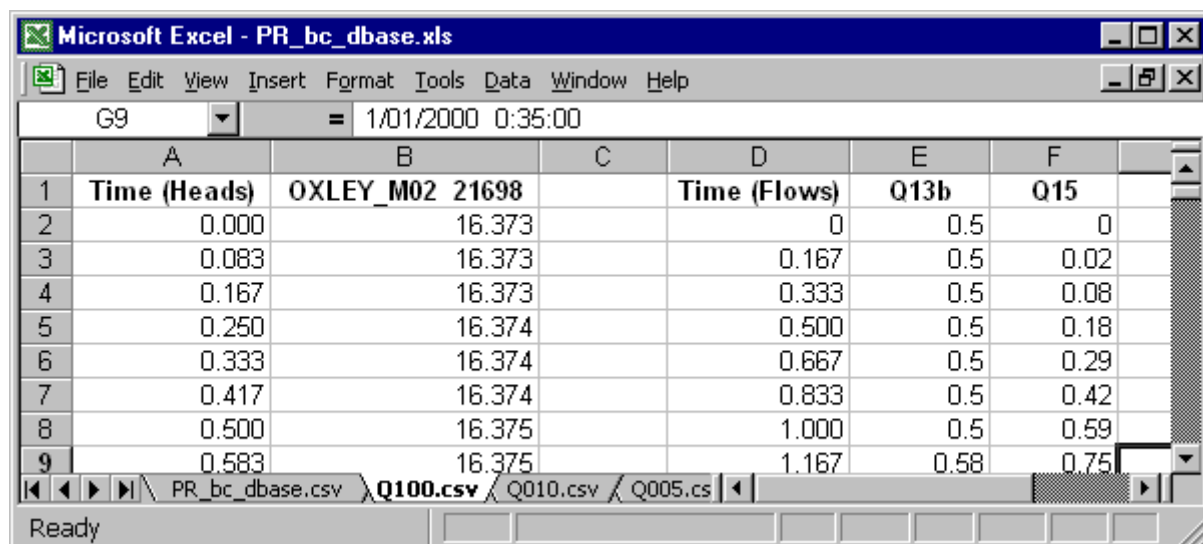
As of the 2010-10 release, TUFLOW replaces **every** the occurrence of “__event__” with “Q100” in each line of the BC database. (Note: Prior to the 2010-10 release, only the **first** occurrence was changed.) If “__event__” does not occur the line remains unchanged. In the example below, the following occurs:

- For the BC event “Oxley Ck Inflow”, the BC data is read from file “Q100.csv” rather than “__event__.csv” as indicated in the spreadsheet.
- Similarly, the same applies for “h Downstream” and “Paradise Ck Inflow” BC events.



	A	B	C	D	E	F
	Name	Source	Time	Value or ID	TimeAdd	Value
1	Oxley Ck Inflow	_event_.csv	Time (Flows)	Q13b		
2	h Downstream	_event_.csv	Time (Heads)	OXLEY_M02 21698		
3	Paradise Ck Inflow	_event_.csv	Time (Flows)	Q15		
4						
5						
6						
7						
8						
9						

The file Q100.csv is created from the worksheet “Q100.csv” as shown below.



	A	B	C	D	E	F
	Time (Heads)	OXLEY_M02 21698		Time (Flows)	Q13b	Q15
1	0.000	16.373		0	0.5	0
2	0.083	16.373		0.167	0.5	0.02
3	0.167	16.373		0.333	0.5	0.08
4	0.250	16.374		0.500	0.5	0.18
5	0.333	16.374		0.667	0.5	0.29
6	0.417	16.374		0.833	0.5	0.42
7	0.500	16.375		1.000	0.5	0.59
8	0.583	16.375		1.167	0.58	0.75
9						

To set up other simulations, for example a five year flood simulation, it is simply a process of creating the Q005.csv file, copying the Q100.tcf file to a Q005.tcf file and changing [BC Event Name](#) to:

BC Event Name == Q005

There is no need to create another BC database file.

4.10.4 Recommended BC Arrangements

Hydraulic models typically have water level boundaries at the downstream end and flow boundaries at the upstream ends.

Water level boundaries representing the ocean or a lake are specified using 1D or 2D HS and/or HT boundaries. HS and HT boundaries can be combined to represent the different components of the boundary (eg. HS for a sinusoidal tide and a HT for the storm surge component, the combination giving a storm tide). For flood models, occasionally, an upstream water level boundary is used in the absence of reliable river flow estimates. Where the downstream boundary is not at a well-defined water level (eg. ocean), a stage-discharge relationship may be specified using a 1D or 2D HQ boundary. In some situations, a hydraulic structure that is inlet controlled acts as the downstream control, in which case, the water level specified downstream of the structure has no influence on the results.

As of Build 2006-06-AA, use the 2D QT boundary for applying inflows to the 2D domains. Prior to Build 2006-06-AA, the recommended approach for 2D flow boundaries was to dynamically link a 1D node to a 2D HX boundary and apply the flow to the 1D node (Syme 1991). The inflow to the 1D node, generates a flow into the 2D domain across the 2D HX boundary. This combination benefits from the stability, wetting and drying performance and the oblique boundary flexibility of water level boundaries. The velocity distribution and direction across the 2D HX boundary is automatically determined by the flow regime that develops in the 2D domain.

2D water level (HQ, HS and HT) and flow (QT) boundaries should be digitised so that they are approximately perpendicular to the flow direction. There are no issues in digitising the boundary at any orientation to the 2D grid (ie. they do not have to be parallel to the grid), and the boundary polyline can “bend”, ie. it does not have to be a straight line.

2D inflows may also be via sinks and sources such as 2D RF, SA and ST boundaries.

Table 4.29 provides an overview of the type of boundaries typically used for different locations.

Table 4.29 Recommended BC Arrangements

Type of Boundary	1D	2D
Ocean or Estuary	HT or HS	HT or HS
Lake	HT	HT
River/Creek Outflow	HQ or HT	HQ or HT
River/Creek Inflow	QT as point on inflow node.	QT preferred or SA
Local Catchment Inflows around the edge of model or within model.	QT as regions (flow is distributed between nodes within a region)	SA. Use SA with PITS option for directing inflows directly to gully traps.
Direct Rainfall (No Hydrology Inflows)	No option at present	RF or possibly SA with RF option.
Dambreak Hydrograph	QT	SA or QT. SA may offer greater stability and better mass error if mass errors occur with QT.
Pumps	QH	SH or ST
Infiltration	No option at present other than specifying a negative QT.	RF – use negative values.

4.10.5 Linking 1D and 2D Domains

4.10.5.1 Linking ESTRY (TUFLOW) 1D Domains

The first command required to link 1D domains into 2D domains is [ESTRY Control File](#) in the .tcf file. This provides a link between the .tcf file and the .ecf file. It is strongly recommended that [ESTRY Control File Auto](#) be used to force the .ecf file to have the same name as the .tcf file. Both .tcf and .ecf files should be in the same (runs) folder (see Section 2.2.2).

To link 1D and 2D domains use the 2D HX and 2D SX boundary types connected to 1D nodes (1d_nwk layer) using CN lines or points in the 2d_bc layer as described in Table 4.32 and Table 4.33 in Section 4.10.7. The recommended approach for when to use HX and SX connections is as follows:

- 2D HX boundaries are preferred for transitioning between 1D domains and 2D domains, or when carving a 1D network through a 2D domain (see Figure 4-9). They can also be successfully used for connecting to 1D structures embedded into a 2D domain, especially where preservation of momentum from 2D to 1D to 2D is important (see Figure 4-4).
- 2D SX boundaries are preferred for inserting 1D channels inside a 2D domain. For example, a 1D culvert underneath a road embankment, or for modelling connections to underground pipe drainage systems.

4.10.5.2 Linking ISIS 1D Domains

TUFLOW 2D domains can also be dynamically linked to ISIS 1D domains. The ISIS software must be installed and configured to support the linking to TUFLOW.

The linking approach is very similar to linking to ESTRY. A 1d_x1d GIS layer showing the locations of the ISIS 1D nodes must be created. This layer requires only one attribute, namely a string 12 characters long that contains the unique IDs of the ISIS 1D nodes. Any other attributes are presently ignored. Creation of this layer may be possible through exporting from ISIS a text or csv file containing the Node ID and XY coordinates of the nodes provided the coordinates are in the same projection as the 2D domain(s).

Note: As of Build 2007-07-AA, the IDs are case sensitive (because ISIS is case sensitive), therefore, the ID in ISIS and the IDs in the 1d_x1d layer(s) must be identical and case sensitive.

In the .tcf file, specify the following command to read the 1d_x1d layer:

[Read GIS ISIS Nodes](#) == ..\model\mi\1d_x1d_isis_nodes.mif
(alternatively the [Read GIS ISIS Network](#) may also be used)

Connections (CN) in the 2d_bc layer are snapped to the nodes in the 1d_x1d layer in the same manner as if connecting to a 1d_nwk layer. The 1d_x1d layer can contain ISIS nodes that are not connected to a 2D domain.

There are presently limitations when connecting to ISIS nodes as follows:

- HX lines should only be connected to ISIS RIVER units. This requires that the ISIS unit between consecutive connections along a HX line is always a RIVER unit. If this is not the

case, an “ERROR - 2D HX cell has been assigned to a non-RIVER unit” occurs. Where a non-RIVER unit occurs (eg. at a structure), the HX line needs to be broken.

- Along a HX line, all ISIS RIVER unit nodes between the upstream and downstream ends of the HX line must be connected (unlike ESTRY where a node can be intentionally or accidentally omitted). If an ISIS node is not connected an “ERROR - Cannot determine upstream ISIS unit node for 2D HX cell” occurs.
- Special ISIS units maybe required for some HX and SX connections (refer to the ISIS documentation).

The same checks that the ZC elevation of a HX cell lies above the bed of the 1D nodes and that the ZC elevation of a SX cell lies below the 1D node bed are made, and if not, an error occurs.

Use the new 2d_x1d_check file to crosscheck the ISIS nodes that were read and associated information passed from ISIS. Also use the 1d_to_2d_check file to crosscheck which HX and SX cells are connected to which ISIS nodes.

Note that the start and end simulation times and the timestep are controlled by the ISIS inputs fields, therefore, any [Start Time](#), [End Time](#) and [Timestep](#) commands are ignored in the .tcf file.

No .ecf file is required and the [ESTRY Control File](#) command should not be specified unless there are also ESTRY 1D domains in addition to the ISIS 1D domains.

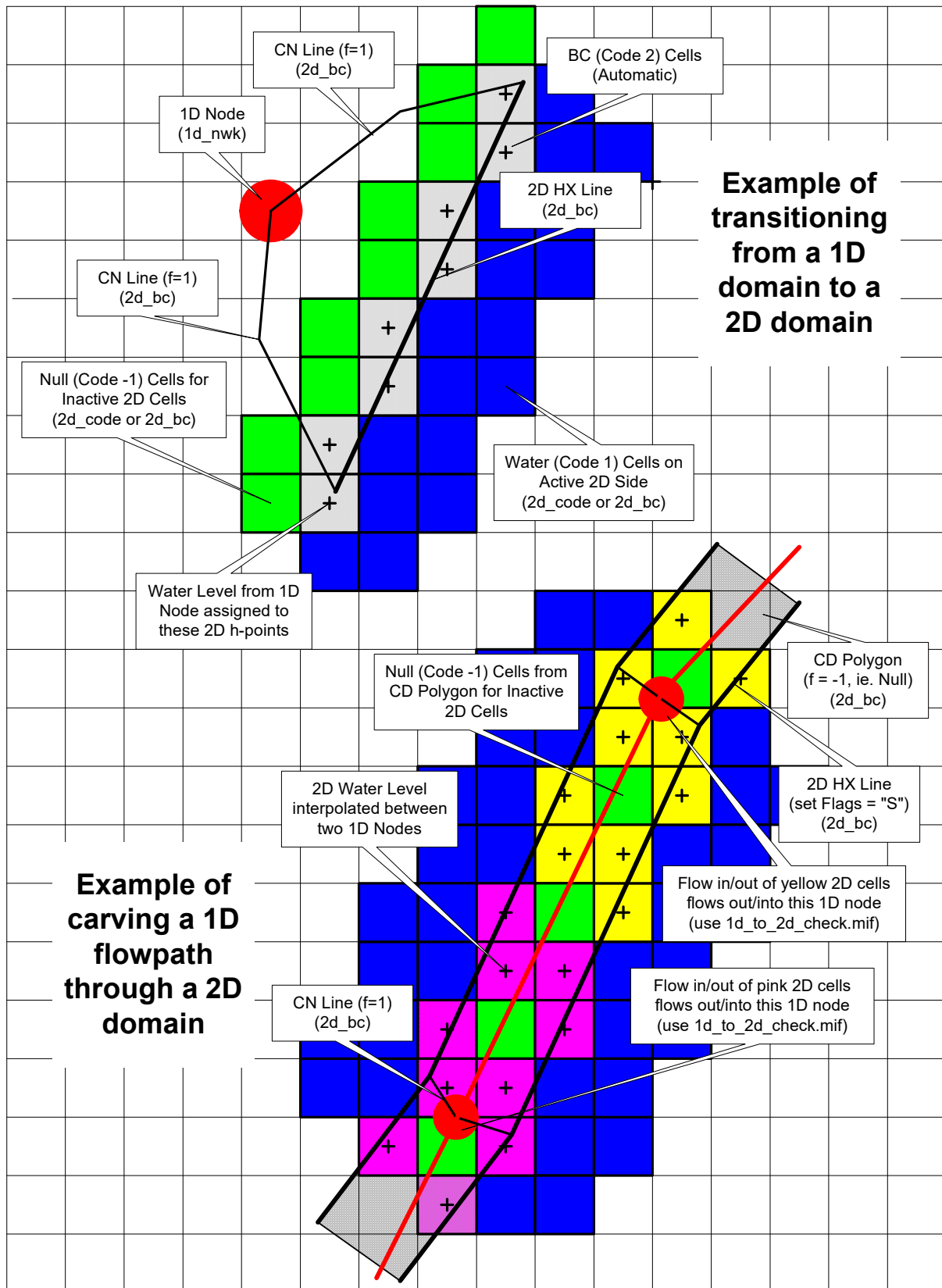


Figure 4-9 Examples of 2D HX Links to 1D Nodes

4.10.6 1d_bc Layers

Boundary conditions for 1D domains are defined using one or more 1d_bc GIS layers. Fixed field text inputs are also supported for backward compatibility. The different types of boundaries and links are described in Table 4.30. Note, links to 2D domains are automatically created from any links specified in the 2d_bc layer (see Section 4.10).

GIS 1d_bc layer(s) contain points that are snapped to the 1D node in a 1d_nwk layer. Each point has several attributes as described in Table 4.32.

As of Build 2005-05-AN, regions can be specified for 1D QT boundaries. If a region is used, the QT hydrograph is equally distributed to all nodes falling within the region that are not a H boundary (this includes nodes connected to the 2D domain via a 2D SX). If no suitable nodes are found an ERROR occurs.

Table 4.30 1D Boundary Condition and Link Types

Type	Description	Comments
Water Level Boundaries		
HS	Sinusoidal (Tidal) Water Level (m)	A sinusoidal wave based on any number of constituents. At present, HS boundaries must be entered using the fixed field approach (see manuals prior to 2007 for formats).
HQ	Water Level (Head) versus Flow (m)	Assigns a water level to the node based on the flow entering the node. This boundary is normally applied at the downstream end of a model.
HT	Water Level (Head) versus Time (m)	Assigns a water level to the node based on a water level versus time curve. If other HT or HS boundaries are applied to the node the water level is set to the sum of the water level boundaries.
HX	Water Level (Head) from an eXternal Source (ie. a 2D domain)	Not required anymore. Was previously used to indicate that a 2D SX boundary is linked to the 1D HX boundary node. This is now determined automatically from 2D SX boundaries.
Treatment		Nodes can be wet or dry. If the water level is below the bed, the bed level is assigned as the water level to the node. As the water level in the node is defined by the boundary, the node's storage has no bearing on the results.

Type	Description	Comments
Combinations		Any number of water level boundaries can be assigned to the same node. The water level used is the sum of the water levels assigned. For example, a storm tide may be specified as a combination of a tidal HS boundary, a HT boundary of the storm surge and another HT boundary of the wave set up. Clearly the HS boundary would be water elevations and the two HT boundaries water depths. The exception is that a HX boundary, being a dynamically linked one, cannot be summed with another H boundary. If you have a 2D SX boundary connected to a node (this automatically applies a 1D HX boundary to the node) and also have a HT and/or HS boundary at the same node, the 1D HX boundary prevails and no warning is given.
Flow Boundaries		
QC	Constant Flow (m ³ /s)	A constant flow boundary. At present, QC boundaries must be entered using the fixed field approach (see manuals prior to 2007 for format). Alternatively, specify a QT boundary and leave the Source column blank and enter in a constant value for the Column 2 value (see Table 4.28).
QH	Flow vs Water Level (Head) (m ³ /s)	Assigns a flow to the node based on the water level of the node at the previous half timestep.
QT	Flow versus Time (m ³ /s)	Assigns a flow into the node based on a flow versus time curve. A negative flow extracts water from the node. As of Build 2005-05-AN, a region (polygon) can be used to equally distribute the flow hydrograph to all nodes that fall within the region. Any nodes that are a H boundary or are connected to a 2D SX cell are not included. As of Build 2008-08-AA, the limit on the number of 1D QT regions assigned to a 1D node is ten (10) - the limit previously was one (1).
QX	Flow from an eXternal Source (ie. a 2D domain)	Not required anymore. Was previously used to indicate that a 2D HX boundary is linked to the 1D QX boundary node. This is now determined automatically from 2D HX boundaries.
Treatment		The node can be wet or dry. The storage of the node influences the results. If the node storage is made excessively large, the flow hydrograph is attenuated, while if it is under-sized the node is likely to be unstable.
Combinations		Any number of flow boundaries can be assigned to the same node. The final flow is the sum of the flows assigned. A connection to a 2D HX boundary (automatically set as a QX boundary at the node) can be applied in conjunction with other Q boundaries.

Table 4.31 1D Boundary Conditions (1d_bc) Attribute Descriptions

GIS Attribute	Description	Type
Type	The type of BC using one of the two letter flags described in Table 4.30.	Char(2)
Flags	Available flags are: S Apply a cubic spline fit to the boundary values (HT, QT, HQ and QH only). Useful for simulating tidal HT boundaries. F Prior to Build 2003-05-AE, was required to set the logical “Ignore” field to false indicating to apply the boundary condition. Now not required. T For backward compatibility can enter a “T” to ignore the boundary condition. As of Build 2008-08-AA, the flags below are available for QT regions. Any combination of C, O and P can be used. If C, O or P are not specified, the flow is distributed to all non-H boundaries (1D H boundaries are any boundary starting with H or any 2D SX connections). Note, specifying CO is the same as specifying no flags. QT regions only: C Only direct flow into closed nodes (closed nodes are those that only have C, R, I and pit channels connected). O Only direct flow into nodes that have at least one open channel. P Only direct flow into the bottom of pits channels. At Build 2003-05-AE attribute was changed from “Ignore” to “Flags”. Previous values of F for false (F) to apply the boundary condition and T for true to ignore are still supported. Recommended that the attribute name is changed from “Ignore” to “Flags” and the type from “Logical” to “Char(6)”, and clear all “F” values, for existing models.	Char(6)
Name	The name of the BC in the BC Database (see Section 4.10.1). If no name is specified, this indicates that a boundary will be provided using EB Data to read fixed field boundary condition table formats.	Char(50)
Description	Optional field for entering comments. Not used.	Char(100)

4.10.7 2d_bc, 2d_sa and 2d_rf Layers

Boundary conditions and links to 1D domains for 2D domains are defined using one or more 2d_bc GIS layers. Fixed field text inputs are also supported for backward compatibility. The different types of boundaries and links are described in Table 4.32.

The GIS layers may contain points, lines, polylines and regions, noting that for regions only the centroid is used. Each object has several attributes as described in:

- Table 4.33 for 2d_bc layers referenced by [Read GIS BC](#).
- Table 4.34 for 2d_sa layers referenced by [Read GIS SA](#).
- Table 4.35 for 2d_rf layers referenced by [Read GIS RF](#).

If using direct rainfall, Build 2008-08-AA or later should be used (without any backward compatibility switches to builds before Build 2008-08-AA). In most cases the double precision version (see Section 5.2) is needed to provide sufficient accuracy.

As of Build 2006-06-AA, the default method for selecting boundary or link cells is to use the “cross-hair” approach. For boundaries or links digitised as a line, cells are selected only if the line intersects imaginary “cross-hairs” that extend from the cell’s mid-sides to mid-sides. This means that if a boundary or link line starts inside a cell, and does not intersect with the cross-hairs, that cell will not be selected. Consequently, the selection of boundary and link cells maybe slightly different if upgrading a model to Build 2006-06-AA or later. The new approach does NOT affect:

- Point objects (which will select the cell that the point lies within – avoid snapping to cell sides as the cell selected maybe unpredictable).
- Polygon objects (which select cells where the cell centroid lies within the polygon).

The new approach has also built in more stringent conditions where a cell can only be assigned one boundary from a single GIS layer (except for: pit SXs; sink/source points or lines with two vertices only; and polygon boundaries such as SA and RF). **If other boundaries are subsequently assigned (eg. to apply a storm surge on top of an ocean tide), these must be in separate GIS layers.** Note that some boundary types cannot be assigned more than once to the same cell regardless as documented in Table 4.33 (for example, HX 1D/2D interface boundaries and the new 2D QT boundaries).

Additional error checks have also been incorporated during the input of 2D BCs provided [Boundary Cell Selection](#) == Method C or [Line Cell Selection](#) == Method D (the new default) is set. TUFLOW stops with an error if a cell is:

- assigned a HT or HS and is already a Q, S or HX cell
- assigned a 2D or HX and is already any other boundary
- assigned a Q or V and is already a H or S
- assigned a ST, RF or SX and is already a H or Q

A useful tip to vary a 2D water level boundary both temporally and spatially is as follows. Digitise a 2d_bc HX line along the boundary. At the ends of the boundary, and at any vertices in between,

digitise 1d_bc HT (or HS) boundaries (ensure they are snapped to the 2d_bc HX line). At each 1D HT boundary specify the water level versus time hydrograph at that location (or use a HS curve). The water levels along the 2D HX line are based on a linear interpolation of the 1D HT/HS hydrographs. This feature is particularly useful for coastal models where the tidal boundary varies in height and phase along the boundary.

4.10.7.1 2d_sa Trigger Option

The 2009-07 release has a new option added to the .tbc [Read GIS SA](#) 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 reaches the reservoir rather than at fixed point in time.

An example of the new option is:

[Read GIS SA Trigger](#) == m\2d_satr_dambreaks.mif

The 2d_satr layer is a 2d_sa layer (with one attribute, Name) to which the three new attributes below are added and are described in Table 4.34.

- Trigger_Type
- Trigger_Location
- Trigger_Value

Every timestep during the simulation, the flow or water level of the 2d_po object referenced by Trigger_Location is monitored and once Trigger_Value exceeds the 2d_po value the SA hydrograph commences.

If applying the hydrograph as a dambreak, when digitising the SA trigger polygon, it should preferably be on the downstream side of the dam wall, and covering the width of the dam wall possibly several cells thick (in the direction of flow). If there are stability issues, enlarging the SA polygon would help, however applications have indicated the feature performs well with the SA polygon a few cells thick.

The 2d_po line (flow) or point (water level) referred to by Trigger_Location can be located anywhere in the model. For cascade reservoir failure modelling it 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 just upstream of the dam.

Note: 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).

Table 4.32 2D Boundary Condition and Link Types

Type	Description	Comments
Water Level Boundaries and Links		
2D	Links two 2D domains	“Stitches” two 2D domains together by a series of water level control points. Momentum across the link is preserved provided the Zpt elevations along the selected cells in both 2D domains are the same or similar.
HS	Sinusoidal (Tidal) Water Level (m)	A sinusoidal wave based on any number of constituents. Four columns of data are required in the source file if using .csv files. The four columns in order are the mean water level (m), amplitude (m), phase difference (°) and period (h). Each row of data represents the harmonics of one wave. Any number of harmonics can be specified within the one HS boundary. Prior to Build 2003-05-AC, HS boundaries had to be entered using the fixed field approach (see manuals prior to 2007 for formatting).
HQ	Water Level (Head) versus Flow (Q)	Assigns a water level to the cell(s) based on a water level versus flow (stage-discharge) curve. Alternatively, a slope can be specified (see Table 4.33) using the b attribute, and TUFLOW automatically generates the HQ curve. As of Build 2007-07-AE, the variation in Manning's n with depth feature is taken into account if automatically calculating the stage-discharge curve. Previously, the Manning's n at the lower depth was used at all depths. This enhancement only applies if Defaults == PRE 2007-07-AA or PRE 2006-06-AA is not specified.
HT	Water Level (Head) versus Time	Assigns a water level to the cell(s) based on a water level versus time curve.
HX	Water Level (Head) from an eXternal Source (ie. a 1D model)	One or two 1D nodes provide a water level every half timestep. Automatically creates 1D QX boundaries at the node(s) (see Table 4.30), which receive a flow from the 2D domain every half timestep. 2D HX boundaries are linked to 1D nodes using CN connections (see below). Tip: A common cause for instabilities at the start of a simulation is that the starting water level in the 1D node is different to those in the adjacent 2D cells.
Treatment		Cell(s) can be wet or dry. It is <u>not</u> a requirement that at least one cell is wet. HT lines can be oblique to the X-Y axes, in which case, Oblique Boundary Method should be set to “ON” (this is the default). The water level can vary in height along a line of cells. Tip: A common cause of instabilities at or near head boundaries at the start of a simulation is the initial water level specified at the adjacent cells is different to the head value. If your model immediately goes unstable at the boundary, check your initial water levels. If it is a 2D HX boundary the water levels in the 1D node and the 2D cells should be similar.

Type	Description	Comments
Combinations		Any number of water level boundaries can be assigned to the same cell(s) except for HQ boundaries, and 2D and HX links. The water level used is the sum of the water levels assigned. For example, a storm tide may be specified as a combination of a tidal HS boundary, a HT boundary of the storm surge and another HT boundary of the wave set up. The HS boundary would be water elevations and the two HT boundaries water depths. The exception is that a 2D 2D or HX boundary, being a dynamically linked one, cannot be summed with another H boundary. In earlier versions of TUFLOW, if you accidentally specify a 2D HX boundary and a 2D HT or HS boundary at the same cell, the 2D HX boundary prevails and no warning is given.
Flows (2D Flows With A Direction Component)		
QC	Constant Flow (m ³ /s)	A constant flow boundary. At present, QC boundaries must be entered using the fixed field approach (see manuals prior to 2007 for formatting). The velocity is determined from the flow value and the model water levels. The direction of flow is required. Note, this boundary is not included in the mass balance calculations. Alternatively, specify a QT boundary and leave the Source column blank and enter in a constant value for the Column 2 value (see Table 4.28).
QT	Flow versus Time (m ³ /s)	Distributes flow in quantity and direction across the cell(s) based on their topography, bed roughness and whether upstream or downstream controlled flow. The limiting assumption is that the water level along the line is constant, therefore, the line must be digitised roughly perpendicular to the flow and should avoid areas where significant superelevation or other similar effects occur. This option was introduced for Build 2006-06-AA and replaces the previous QT boundary, which is now a QT (with “A” Flag) as described below.
QT (A Flag)	Flow versus Time (m ³ /s)	Assigns a velocity and a flow direction to the sides of the cell(s) based on a flow versus time curve. The velocity is determined from the flow value and the water depths. The direction of flow is required. Note, this boundary is not included in the mass balance calculations.
VC	Constant Velocity (m/s)	Same as for a constant flow boundary (see QC above) except a velocity is specified. Note, this boundary is not included in the mass balance calculations.
VT	Velocity versus Time (m/s)	Same as for a QT boundary (see above) except a velocity is specified. Note, this boundary is not included in the mass balance calculations.

Type	Description	Comments
Treatment		For the new QT option in Build 2006-06-AA, cells can wet and dry, the line can be oblique to the grid. This is the recommended boundary for applying a flow hydrograph directly to a 2D domain. For the other types, cell(s) can be wet or dry, however, it is recommended that cells remain wet, otherwise the quantity of flow is dependent on the number of wet cell(s) along the boundary. QT (A Flag) lines should be specified along lines parallel or 45° to the X-Y axes. Except for the QT option, these boundaries are rarely used. Tip for Builds prior to 2006-06-AA: It is strongly recommended to use a 1D node linked to a 2D HX boundary (see above) in preference to using a flow boundary, especially in flood models where there is major wetting and drying. This arrangement is far more practical, stable and flexible (the boundary can wet and dry, can lie oblique to the grid, and the velocity distribution and flow direction across the boundary is automatically determined).
Combinations		Any number of flow and velocity boundaries can be assigned to the same cell(s), except for QT boundaries. The final velocity is the sum of the velocities assigned.

Sources (2D Flows With No Direction Component)

RF	Rainfall versus Time (mm) Infiltration versus Time (mm)	As of Build 2006-05-AN, a rainfall hyetograph can be applied. The rainfall time-series data must be in mm versus hours, and is converted to a hydrograph to smooth the transition from one rainfall period to another (the converted hydrograph appears in the .tlf file for cross-checking). Note: the input curve is not mm/h versus h, but mm vs h, ie. each rainfall value is the amount of rain that fell in mm between the previous time and the current time. The first and second rainfall values value in the hyetograph should be set to zero to ensure that the hyetograph when converted to a flow hydrograph has an initial value of zero. The start time of the simulation should be set to the first time value in the hyetograph. The approach applies a rainfall depth to every active cell (ie. Code 1 cells) within each region, and essentially replaces the need to use a hydrological model. Initial and continuing losses are applied on a material-by-material basis (see Read Materials File). Build 2008-08-AA offers substantial improvements for direct rainfall modelling over previous builds, offering much improved stability for shallow sheet flow, and much reduced mass errors when the cells first become wet. The double precision version (see Section 5.2) of Build 2008-08-AA or later is recommended for direct rainfall models. Builds prior to Build 2008-08-AA should no longer be used for direct rainfall modelling other than for established models that perform satisfactorily using prior builds. RF boundaries have their own command, Read GIS RF and Read RowCol RF, and own GIS layer (see Table 4.35). As of Build 2007-07-AA (and 2006-06-BF), if a negative rainfall is specified

Type	Description	Comments
		this is treated as a loss (ie. an infiltration into the ground and/or evaporation). This is particularly useful if there is a loss of water into the ground or into the atmosphere due to evaporation. Negative rainfall is only applied to wet cells (ie. it does not apply to dry cells), whereas a positive rainfall is applied to all active cells whether wet or dry.
SA	Flow versus Time (m^3/s) over an area, or Rainfall versus Time (mm)	Applies the flow directly onto the cells within the polygon as a source. Negative values remove water directly from the cell(s). Most commonly used to model rainfall-runoff directly onto 2D domains with each polygon representing the sub-catchment of a hydrology model. SA boundaries have their own command, Read GIS SA, and own GIS layer (see Table 4.34). If Read GIS SA is used without any options the flow hydrograph is applied as follows. Within each SA catchment (region), if all the 2D cells are dry, the flow is directed to the lowest cell based on the ZC elevations. If one or more cells are wet the total flow is distributed over the wet cells. If using the ALL or PITS options, the flow is equally distributed to all cells (see Read GIS SA). The ALL option is similar to the direct rainfall approach and may require the use of the double precision version (see Section 5.2). As of Build 2005-06-AA, a rainfall hyetograph can be applied (see Read GIS SA RF). The rainfall time-series data must be in mm versus hours, and is converted to a hydrograph to smooth the transition from one rainfall period to another. Initial and continuing losses are entered as attributes to the 2d_sa layer (see Table 4.34). Note: If two or more SA inflows of the same name cover the same cell, only the first inflow is used. Recommendation is to have a unique name for each SA polygon and/or do not overlap polygons. Also, two SA polygons of the same name are treated as one polygon in terms of setting the lowest cell for inflows if all cells are dry.
SH	Flow versus Head (m^3/s)	Extracts the flow directly from the cells based on the water level of the cell. Used for modelling pumps or other water extraction. Flow values must not be negative. SH boundaries can be connected to another 2D cell or a 1D node, to model, for example, the discharge of a pump from one location in a model to another. The connection is made using a “SC” line (see below). In the boundary database, the Column 1 data would be head or water level values and the Column 2 data would be flow. The flow value is the rate per 2D cell. If the 2D cell becomes dry, no flow occurs. Feature incorporated in Build 2003-03-AD.
ST	Flow versus Time (m^3/s)	Applies the flow directly to the cells as a source. Negative values remove water directly from the cell(s). Can be used to model concentrated inflows, pumps, springs, evaporation, etc. The flow specified in the boundary file is applied to each cell to which the boundary is connected. For example, if the boundary file specifies $2 \text{ m}^3/\text{s}$ and the ST is applied over four cells, then the total flow applied to the model would be $8 \text{ m}^3/\text{s}$. If the total flow required is $2 \text{ m}^3/\text{s}$, then a f

Type	Description	Comments
		attribute of 0.25 could be applied so that only 0.5 m ³ /s is applied to each cell.
SX	Source of flow from a 1D model.	2D SX cell(s) are connected to a 1D node using a single CN connection (see below). The net flow into/out of the 1D node is applied as a source to the 2D cells. For example, a 1D pipe in the 2D domain “sucks” water out of the upstream cell(s) and “pours” water back out at the downstream cell(s) using 2D SX boundaries. 2D SX boundaries can also be used to model pumps – see “U” flag in Table 4.33. As of Build 2003-08-AE, if an SX cell falls on an inactive cell (Code -1 or 0), the cell is set as active (Code 1).
Treatment		Sources are applied to all the specified cell(s) whether they are wet or dry, except for SA and SX, which apply only to wet cells, or the lowest dry cell if all the SA or SX cells are dry.
Combinations		Any number of source boundaries can be assigned to the same cell(s) whether they are SA, SH, ST or SX. The source rate applied is the sum of the individual sources.
Connections		
CN or EC	Connection of 2D HX and 2D SX boundaries to 1D nodes	Used in GIS 2d_bc layers to connect 2D HX and 2D SX boundaries to 1D nodes. A line or polyline is digitised that snaps the 2D HX or SX object to the 1D node. The 1D node would be in a 1d_nwk layer. Note that if the 2D SX object is snapped to the 1D node, no CN object is required. However, 2D HX objects always require a CN object to connect to the 1D node. Alternatively a CN point object can also be used instead of a line. As of Build 2003-06-AA, an ERROR occurs if a CN object is not snapped to a 2D HX or 2D SX object, or is redundant (ie. not needed). For backward compatibility, use Unused HX and SX Connections (.tcf file) or Unused HX and SX Connections (.tbc file) to change the ERROR to a WARNING. Note that for connections to 2D SX objects only one (1) CN object is required. Whereas 2D HX objects must have a minimum of two (2) CN objects – one at each end – with intermediate CN objects as needed to connect to any 1D nodes.
SC	Connection of 2D SH boundaries	Used for connecting 2D SH boundaries to another 2D cell or 1D node (eg. modelling the pumping of water from one location to another).

Type	Description	Comments
Wind Stresses		
WT	Modelling Wind Stresses as a force on the water column	Time varying wind stresses are available with the 2009-07 release. The undocumented .tcf “Apply Wind Stresses ==” command has been disabled and should not be used. 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. The wind time-series curve is entered as three columns of data (using Column 1, 2 and 3 labels – see Table 4.28). The three columns are: • Time (h) • Wind (m/s) • Direction (degrees relative to East, ie. East = 0°, North = 90°, etc).
Variable Geometry		
VG	Modelling of breaches, etc	Used for varying cell elevations over time. Each cell, or line of cells, needs to be assigned a time series of elevations in the same manner that other boundaries are applied. Also see Section 4.4.10 on Variable Z Shape layers as an alternative. Note: If varying the elevations of a HX cell, the elevation must not fall below the 1D bed value (see the attributes of the 1d_to_2d_check file for that cell). No run-time checks are made in this regard. Also see VG Z Adjustment.
Other		
CD	Code Polygon	Objects in a GIS 2d_bc layer used to define the grid’s cell codes using Read GIS Code BC as an alternative to Read GIS Code. The code value is set using the f attribute (see Table 4.33). The boundary lines are snapped to “CD” regions so that if the boundary location is adjusted, the boundary line and code region can move together. See Read GIS Code [{} BC].
IG	Ignore	An object in a GIS 2d_bc layer can be elected to be ignored by using the “IG” type.
ZP	Elevation Point	Point objects that are used for setting elevations along HX lines. Only used by Read GIS Z HX Line . Incorporated into Build 2008-08-AA.

Table 4.33 2D Boundary Conditions (2d_bc) Attribute Descriptions ([Read GIS BC](#))

GIS Attribute	Description	Type
Type	The type of BC using one of the two letter flags described in Table 4.32.	Char(2)
Flags	Optional flags as follows: “R” – Repeat previously specified boundary (fixed field input only) HT, QT, SH, VG, VT: “S” – Fit a cubic spline curve to the data. HX: As of Build 2006-03-AB, the “S” flag is not recommended unless Adjust Head at Estry Interface is set to ON (see below). “A” – As of Build 2008-08-AC, used to override the RIDGE, MAX or RAISE option in Read GIS Z HX Line. “L” – sets the ZU and ZV elevations to be the same as the ZC value only if they are lower. This can improve instabilities where the ZU and ZV values are significantly lower in elevation, and can cause a sudden increase in transfer of water to/from the cell when the cell wets. “2” – can offer improved performance along HX lines by attempting to more smartly allocate water levels along a HX line when there is a dry section (“hump”) between 1D nodes. “R” – applies the RIDGE option when setting elevations along HX lines using Read GIS Z HX Line. “S” – Prior to Build 2006-03-AB, was recommended to override Adjust Head at Estry Interface, which, by default, was set to ON (as of Build 2006-03-AB, the default is OFF). The S flag sets the 1D and 2D water levels along the HX line to be the same. This suppresses the adjustment of the 1D water level by the average dynamic head (see Syme 1991), and it was needed to use the S flag for 2D HX lines alongside 1D channels flowing through a 2D domain. “V” – reserved flag for possible future releases – do not use. “Z” – Adjust the ZC elevation at each cell at/along the 2D HX object to slightly above the 1D node bed elevation where the 2D HX ZC value is lower. The ZC elevation is set to 1mm above the interpolated 1D node bed less the Cell Wet/Dry Depth. An error occurs if the 2D cell ZC elevation is not above the interpolated 1D node bed. Also see HX ZC Check. It is not recommended to use the Z flag without first checking that the reason for the discrepancy in elevations between 1D and 2D domains is appropriate. QT: “A” – As of Build 2006-06-AA, applies the original QT boundary, which requires an angle of flow, etc. SX: “Z” – Adjust the ZC elevation at each cell at/along the 2D SX object to below the 1D node bed elevation where ZC is higher. The ZC elevation is set to the Cell Wet/Dry Depth below the 1D node bed. Note: As of Build 2002-08-AG, an error occurs if the minimum ZC elevation plus the Cell Wet/Dry Depth at/along a SX object is not below the connected 1D node bed. Also see SX ZC Check. It is not recommended to use the Z flag without first checking that the reason for the discrepancy in elevations between 1D and 2D domains is	Char(3)

GIS Attribute	Description	Type
	appropriate. SX: “U” – indicates that the 2D domain receives the flow (in or out) from the 1D domain, but does not set the water level at the 1D node. This allows pumps (modelled as a “QH” or other Q boundary at the 1D node) to discharge into or extract water from the 2D domain. CD, CN, HS, QC, RF, VC, WT, ZP: Not used.	
Name	HS, HT, QT, RF, SH, ST, VG, VT, WT: The name of the BC in the BC Database (see Section 4.10.1). HS was incorporated in Build 2003-05-AC, SH in 2003-03-AD. HQ: As per above, however, if using the HQ automatic stage-discharge curve generation (ie. b attribute is greater than zero), this attribute is ignored. HQ was incorporated in Build 2006-06-AA. CD, CN, HX, QC, SX, VC, ZP: Not used.	Char(100)
f	HT, QT, RF, ST, VG, VT: Multiplication factor applied to the boundary values. f is assigned a value of one (1) (which has no effect) if the absolute value is less than 0.0001. The values may also be factored using the ValueMult keyword (see Table 4.28). HS: Multiplication factor applied to the amplitude. A value of one (1) is applied if the field is left blank. CN: When used in conjunction (snapped) with a 2D HX object, sets the proportion or weighting to be applied in distributing the water level from the 1D node to the 2D cell. One or two 1D nodes can be connected to the same point on a 2D HX object. Checks are made that the sum of all CN f values connected to a 2D HX point or 2D HX line/polyline node equals one (1). If only one 1D node is connected, set f to one (1). As of Build 2004-07-AC, an f value of zero (less than 0.001) is set to one (1). CD: The code value to be assigned to cells falling on or within the object. SX: n offset for determining cell invert levels for distributing flows. ZP: As of Build 2008-08-AA, used to set the elevation of the 2D cells selected by the HX line. To adjust the height of the HX cells (usually to represent the levee crest or overtopping height) use ZP points, or have a separate layer of elevation points (see Section 4.4.11), snapped to the vertices of the HX line. If there are no points snapped to the HX line a CHECK is issued and the Zpt elevations of the HX cells are unchanged. ZP points are only used by Read GIS Z HX Line. HX, QC, VC, HQ: Not used.	Float

GIS Attribute	Description	Type
d	2D: The minimum distance between 2D2D water level control points between vertices along the 2D line. If set to zero, only the vertices along the 2D polyline are used. This value should not be less than the larger of the two 2D domains' cell sizes. As of Build 2006-06-AA, this feature is under development and further testing, and maybe subject to change. HT, QT, RF, ST, VG, VT: Amount added to the boundary values after the multiplication factor f above. Values may also be adjusted using the ValueAdd keyword (see Table 4.28). HX, ZP: As of Build 2008-08-AA, used to adjust the elevations up or down by the amount of the d attribute. Performs a similar function to the dZ attribute for 2d_zsh layers (see Table 4.8). For example, to raise the 2D HX cells by 0.2 metres set d to 0.2 (note this only applies if ZP points are snapped to the HX line). Only used by Read GIS Z HX Line. QC, VC: The value of constant velocity or flow. SX: m offset for determining cell invert levels for distributing flows. CD, CN: Not used.	Float
td	HT, QT, RF, ST, VG, VT: Incremental amount added per cell to the boundary's time values. Time values may also be adjusted using the TimeAdd keyword (see Table 4.28). HS: Incremental phase lead or lag in degrees per cell along the boundary. SX: Reserved – set to zero. CD, CN, HX, QC, VC, ZP: Not used.	Float

GIS Attribute	Description	Type
a	2D: Increasing this value from the default of 2 may improve stability, but may unacceptably attenuate results. HT, VG: Incremental adjustment of the multiplication factor per cell along the boundary. For the n^{th} cell along the boundary the water level or cell elevation (h_n) is adjusted according to: $h_n = h_1 (1 + a(n - 1)) + b(n - 1)$ where h_1 is the water level at the first cell. HS: Incremental adjustment of the amplitude per cell along the boundary. For the n^{th} cell along the boundary the amplitude (A) is adjusted according to: $A_n = A_{n-1} + a$ HX: Used to add FLC value to all HX cells along the HX line (Build 2009-07-AE or later). This can be useful for 1D/2D models where additional energy losses are needed to model the flow between a river (1D) and the floodplain (2D). Set to zero for backward compatibility. Alternatively Read GIS FLC can be used. QT: As of Build 2006-06-AA, a can be used to stabilise the boundary if needed by adding more “storage”. The default value is 5. Note: increasing this number by excessive amounts can unacceptably attenuate the hydrograph. QC, QT (with A Flag), VC, VT: Angle of flow direction in degrees relative to the X-axis, ie. X-axis (left to right) is zero, Y-axis is 90°, etc. CD, CN, RF ST, SX, ZP: Not used.	Float
b	HT, QT (with A Flag as of Build 2006-06-AA), RF, ST, VG, VT: Incremental amount added per cell to the boundary values after any incremental multiplication factor. Values may also be adjusted using the ValueAdd keyword (see Table 4.28). HQ: Water surface slope in m/m for automatic calculation of the stage-discharge relationship. If b is greater than zero, the automatic approach is adopted irrespective of whether the Name attribute is blank or not. HS: Incremental amount added per cell to the mean water level. CD, CN, HX, QC, SX, VC, ZP: Not used.	Float

Table 4.34 2D Source over Area (2d_sa) Attribute Descriptions ([Read GIS SA](#))

GIS Attribute	Description	Type
Name	The name of the BC in the BC Database (see Section 4.10.1). Note: If two or more SA inflows of the same name cover the same cell, only the first inflow is used. Recommendation is to have a unique name for each polygon and/or do not overlap polygons. If two SA polygons have the same name the polygons are “merged” and treated as one.	Char(100)
Additional Attributes for the RF Option:		
Catchment_Area (m ²)	The contributing catchment area. Note: This attribute is required as the polygon area is not used as the sub-catchment area because the rainfall area may be different to the polygon extent. A value of zero will cause cause zero rainfall to be applied.	Float
Rain_Gauge_Factor	A multiplier that allows for adjusting the rainfall due to spatial variations in the total rainfall. A value of zero will cause cause zero rainfall to be applied.	Float
IL (mm)	The Initial Loss amount in mm.	Float
CL (mm/h)	The Continuing Loss rate in mm/hr.	Float
Additional Attributes for the Trigger Option:		
Trigger_Type	Trigger_Type 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.	Char(40)
Trigger_Location	Trigger_Location is the PO Label in a 2d_po layer (see Section 4.8.1). The 2d_po Type attribute must also be compatible with the Trigger_Type (ie. it must include Q_ or H_ as appropriate).	Char(40)
Trigger_Value	Trigger_Value is the flow or water level value that triggers the start of the SA hydrograph.	Float

Table 4.35 2D Direct Rainfall¹ over Area (2d_rf) Attribute Descriptions ([Read GIS RF](#))

GIS Attribute	Description	Type	FF Cols
Name	The name of the rainfall ¹ BC in the BC Database (see Section 4.10.1). Note: If two or more RF inflows of the <u>same Name</u> cover the same cell, only the first RF inflow is used. Recommendation is to have a unique Name for each polygon and/or do not overlap polygons.	Char(100)	N/A
f1	A multiplier that allows for adjusting the rainfall due to spatial variations in the total rainfall. To vary the rainfall spatially, either apply different f1 and/or f2 attribute values to each polygon, or for a gradually varying rainfall use Read RowCol RF . A value of zero will cause cause zero rainfall to be applied.	Float	N/A
f2	A second multiplier that allows for adjusting the rainfall spatially. A value of zero will cause cause zero rainfall to be applied.	Float	N/A

¹ Initial and continuing losses can be applied on a material-by-material basis (see [Read Materials File](#)). Initial and continuing losses are not applied to negative rainfall values (to model infiltration/evaporation).

4.11 Presenting 1D Domains in 2D Output (1d_wll)

Output from 1D domain(s), whether they be ESTRY, ISIS or XP-SWMM, can be combined with the 2D map output viewed using SMS or a GIS. This allows easier viewing of results and the ability to animate the 1D results in combination with the 2D results.

1d_wll GIS layer(s) are used to define and control the 1D map output. The layer(s) contain lines (called WLLs or Water Level Lines) that cross 1D channels and/or nodes. A WLL is essentially a line of horizontal water level, and should be digitised on this basis (ie. perpendicular to the flow direction).

The direction of digitising the WLLs is important. They must be from left to right looking in the positive direction of the channel.

As of Build 2003-10-AE, two methods are available, although both methods have been modified and improved up until Build 2004-06-AB. Use [WLL Approach](#) to set the preferred method. The two methods are described below. Method B allows more advanced and accurate mapping of 1D results in map formats and became the default and preferred method as of Build 2006-03-AB.

4.11.1 WLL Method A

Note 1: WLL Method B is the recommended approach, and is the only option for ISIS and XP-SWMM 1D Schemes – See Section 4.11.2.

Note 2: There is a known bug in Method A that sometimes causes 1D velocity arrows to appear in the wrong direction. As Method A is no longer supported, there is no plan at present to correct this.

In Method A, each WLL can only have 2 or 3 vertices. To pick up the water level exactly at a node, use a 3 vertex line with the middle vertex snapped to the 1D node. If you use a 3 vertex line across a channel, the channel "thalweg" is taken at the middle vertex, otherwise, for 2 vertex lines the mid-point is used. At this stage, no attributes are used.

Use [Read GIS WLL](#) in the .ecf file to specify the 1d_wll layer and automatically create 1D map output in the SMS files. Several 1d_wll layers can be specified if required. Triangular elements are created in the SMS .2dm mesh file – view these in SMS to check they have been created correctly.

The default is to just use the end vertices and the middle vertex to create triangular elements between WLLs. Additional points along the WLLs can be created using [WLL Additional Points](#) in the .ecf file. If, for example, two additional points are specified, then two extra points are created on each side of the WLL giving a total of 7 points.

The elevations at points along WLLs are presently based on the channel cross-section hydraulic properties table (as reproduced in the .eof file). The elevations are set as constant increments from the bottom elevation of the table to the top elevation. The width at each elevation determines the location of the point along the WLL. The method of locating additional points is controlled by [WLL Adjust XS Width](#). The default method is to adjust the flow width from the hydraulic properties table proportionally along the length of each WLL side. The alternative approach uses the true flow width as would be determined from the hydraulic properties table.

If no additional points are specified, the end points will have an elevation equal to the top elevation in the table and the middle point to the bottom elevation. For fancier looking animations, specify additional points.

If a WLL crosses two or more channels, the channel closest to the middle vertice (3 point line) or half-way point (2 point line) is used.

If a WLL middle vertice snaps to a node with, say two or more channels on the upstream side, the channel it uses to determine flow values is that channel that is closest in angle to the WLL's perpendicular based on the WLL's two end points.

The 1D velocity vectors vary in magnitude across the WLL. At present this is to more easily view the results, with the calculated velocity being the largest one, ie. that shown at the middle vertex along the WLL. Alternative velocity determination based on the relative roughness and depth across the cross-section using Method B below allows more accurate velocity and flood hazard mapping of the 1D results.

4.11.2 WLL Method B

4.11.2.1 Water Level Lines (WLL)

Method B allows elevations and optionally material (Manning's n) values to be assigned to points along a WLL. A more accurate representation of velocity and flood hazard from the 1D domain can be mapped. The velocity at a point on the WLL is estimated by carrying out a parallel channel analysis along the WLL using the flow in the channel the WLL is connected to. The analysis estimates the water surface slope at the WLL based on the conveyance of the profile along the WLL. Implicit in this analysis is that the cross-section selected for the channel produces an average water surface slope representative of that along the length of the channel. The water level at the WLL still remains the linearly interpolated water level between the upstream and downstream nodes.

Method B can have 2 and 3 vertex WLLs as discussed for Method A, as well as WLLs with no limit on the number of vertices. For WLLs with 2 and 3 vertices, the rules discussed above for Method A apply. For 4 or more vertices, one of the vertices (except for the two end vertices) must snap to a vertex on the channel line.

There is one attribute required for Method B (none are required for Method A) as described in Table 4.36. The attribute, dMin, is the minimum distance in metres along which to generate elevation points for that line. If dMin is zero, only elevations at the mid and end points on the WLL are generated.

For TUFLOW 1D (ESTRY), use [Read GIS WLL](#) in the .ecf file to read the 1d_wll layer(s).

For ISIS or XP-SWMM, use [Read GIS ISIS WLL](#) or [Read GIS XP WLL](#) in the .tcf file to read the 1d_wll layer(s). The ISIS units or XP_SWMM links will also need to be built into a GIS layer and read using [Read GIS ISIS Network](#) or [Read GIS XP Network](#) from the .tcf file.

Table 4.36 1D WLL (1d_wll) Attributes

GIS Attribute	Description	Type
Read GIS WLL Command		
dMin	The minimum distance interval along the WLL to generate elevation and material sampling points (WLL Points). These points form the corners of the triangulation. If set to zero, no additional points are generated (ie. only the existing vertices along the WLL are used).	Float

The default approach for setting the elevations at each point is to use the processed cross-section data (CS tables) for the cross-section allocated to the channel. For ISIS and XP-SWMM, this data is automatically fed through to TUFLOW and is based on the cross-section information entered into the ISIS/XP-SWMM model.

As of Build 2007-03-AB, if a WLL is snapped to a node, the processed data used for setting any bed elevations is now that from the higher channel unless it is a bridge in which case it uses the bridge processed data.

[WLL Automatic](#) == CULVERTS is particularly useful for automatically generating WLLs for pipe network systems. [WLL No Weirs](#) is useful for not assigning WLLs to weir channels where they are in parallel to another channel (this is particularly useful for BW, CW and RW channels).

4.11.2.2 Water Level Line Points (WLLp)

If [Write Check Files](#) in the .tcf file is specified, two GIS check layers are created. These are labelled 1d_WLLo and 1d_WLLp. 1d_WLLo reproduces the WLLs with attributes containing information on which channel and nodes the WLL was allocated to. For ISIS and XP-SWMM, the layers are essentially the same, but are named using xWLLo and xWLLp, and will be prefixed the same as for the 2D check files.

1d_WLLp or 2d_xWLLp layers contain all of the elevation points generated based on the dMin attribute. This layer can then be used to allocate elevations (first attribute) to each point from a DTM (in the same manner the 2D Zpts are assigned elevations).

A second attribute, RR, contains the relative resistance of the each point (which will have a value of 1 when first generated). The RR attribute can be replaced by the material value at each point by using a GIS to assign values from material polygons. The material value must exist in the .tmf file.

The attributes of a 1d_WLLp or 2d_xWLLp layer created by [Write Check Files](#) and used in [Read GIS WLL Points](#), [Read GIS ISIS WLL Points](#) or [Read GIS XP WLL Points](#) are listed in Table 4.37.

Note: If using [Read GIS WLL Points](#), this layer must be a copy of the 1d_WLLp layer produced by [Write Check Files](#). Points from this layer can be deleted, but not added. At deleted points, the default of estimating an elevation from the channel's processed data is used. If the 1d_WLL layer is modified or any of the dMin attribute values changed, the 1d_WLLp layer needs to be regenerated again.

For culvert channels (R and C channel type), only the end and mid vertices are used along the WLL, and the elevations are set to the culvert invert irrespective of the number of points along the WLL or the dMin value. This only applies to Method B.

Table 4.37 1D WLL Point (1d_wllp) Attributes

GIS Attribute	Description	Type
Read GIS WLL Points Command		
Z	Ground elevation of the point. Automatically generated from the channel cross-section processed data or point inspected from a DTM.	Float
RR or Material	In the 1d_WLLp check file, the relative resistance of the point. If the elevation was estimated from the channel's processed data a value of 1 is assigned. If the elevation was provided through a point using Read GIS WLL Points, RR is the material Manning's n value divided by the channel's n value. In a 1d_WLLp layer being used in Read GIS WLL Points, this column should either be set to: • zero (0) to force a relative resistance of 1; or • a material value (normally sourced from a GIS layer of material polygons) – the material value must exist in the .tmf file (see Read Materials File).	Float

As of Build 2007-07-AA, elevation values along WLLs for bridge channels are always based on the processed data (ie. any WLLp Z values are overridden) to ensure that the bridge deck underside is correctly represented. This has benefits when using the post-processing utility TUFLOW_to_GIS.exe (see Section 11.2) when extracting obverts of structures for longitudinal profiles.

A useful tip at a junction of 1D channels is to use a connector (Channel_Type = "X" – see Table 4.14). Separate WLLs can then be allocated to the side channel and main channel removing the confusion that sometimes occurs in generating the triangulation between WLLs at junctions.

4.12 Managing Multiple Events and Scenarios

The 2010-10 Release introduces new and powerful functionality to manage the simulation of multiple events and different scenarios. The flexibility offered allows the user the ability to initiate all simulations using the one set of control files. Rather than create a new .tcf file for every simulation and a new .tgc and/or .ecf file for every different model scenario, it is possible to just have one of each of these files, or at least markedly reduce the number of these files.

4.12.1 Modelling Multiple Events Using an Event File

Most hydraulic modelling studies require the simulation of different probability events. For example, a flood study may have to consider a 2, 5, 10, 20, 50, 100, 200 and 500 year event, and one or two extreme floods. Each or most of these may have to be simulated using different duration rainfalls. A downstream boundary such as the ocean may need to be varied to model different probability storm tides or climate change scenarios. The final number of simulations can easily be in the hundreds.

To set your files up to handle multiple events follow these steps and work through the examples shown further below.

- 1 Create a TUFLOW Event File (recommended extension .tef) containing .tcf and .ecf commands that are particular to a specific event. The event specific commands are contained between [Define Event](#) and [End Define](#) commands. Global .tcf and .ecf commands can also be used by placing these commands outside the [Define Event](#) blocks (normally at the top).
- 2 Reference the .tef file in the .tcf file using [Event File](#).
- 3 Up to nine (9) different types of events can be specified for a simulation, although in most cases only one or two is all that is needed. To automatically insert the name of the event(s) into the output filenames so as to make the output names unique, place “~e~” or “~eX~” where X is from 1 to 9 into the .tcf filename.
- 4 To run the multiple events from the same .tcf file there are two options:
 - (a) Setup a .bat file using the new -e or -eX options (see Table 5.2) to run the multiple events using the same .tcf file – refer to the examples below.
 - (b) Using [Model Events](#), change the events manually before each simulation within the .tcf file. Note that if using the -e option, the -e events will override any events defined using [Model Events](#).

The examples below show two approaches for running simulations for a 20 and 50 year flood for the 1 and 2 hour duration storms. Example 1 just uses one type of event. Example 2 achieves the same outcome as Example 1, but by using two types of events (~e1~ and ~e2~) there is no need to repeat commands as is required for Example 1.

In the examples below, note how the same command, in this case [End Time](#), is used within the event blocks to assign a different simulation end time depending on the duration of the event.

Also note the use of [BC Event Source](#), which is a much more powerful alternative to using [BC Event Text](#) and [BC Event Name](#) in that it can be used to define up to 100 different event text/name

combinations instead of being limited to one is using [BC Event Text](#) and [BC Event Name](#). The linkages with the [BC Database](#) file key texts are highlighted in yellow for the “~ARI~” BC event text and cyan for the “~durn~” event text.

Note: The location of [Event File](#) within the .tcf file and the location of the commands within the .tef file is important. If, for example in the examples below, [End Time](#) occurs in the .tcf file after where [Event File](#) occurs, this latter occurrence of [End Time](#) will prevail over those specified in the .tef file. Essentially, the commands from the .tef file (for the events specified using the –e run option or [Model Events](#)) are inserted into the .tcf at the location of the [Event File](#) command, and are processed as if they are embedded into the .tcf file at that location.

Note: To distinguish between 1D and 2D commands in a .tef file, prefix .ecf commands by “1D” followed by a space (see [1D](#)). In the examples below, the .ecf [Output Folder](#) command is used to set different folders for the 1D results depending on the event.

Example 1:

BC Database file:

```
Name,Source,Column 1,Column 2,Add Col 1,Mult Col 2,Add Col 2,Column 3,Column 4
C001,..\Inflows\Nile_~ARI~ ~durn~.csv,Time,C001
C002,..\Inflows\Nile_~ARI~ ~durn~.csv,Time,C002
...
```

Include “~e~” in the .tcf file, for example “Nile_~e~.tcf”, and add the following line:

```
Event File == Nile_Events.tef
```

The “Nile_Events.tef” file would contain lines such as the below:

```
!
! Global Default Settings
Start Time == 0 ! Unless Start Time is subsequently repeated all start at time 0
!
! 20 year event definitions
Define Event == 020y_01h
  BC Event Source == ~ARI~ | 020y
  BC Event Source == ~durn~ | 01h
  End Time == 3
  Output Folder == ..\Results\2d\020y
  1D Output Folder == ..\Results\1d\020y
End Define

Define Event == 020y_02h
  BC Event Source == ~ARI~ | 020y
  BC Event Source == ~durn~ | 02h
  End Time == 4
  Output Folder == ..\Results\2d\020y
  1D Output Folder == ..\Results\1d\020y
End Define
```

```
!
! 50 year event definitions
Define Event == 050y_01h
  BC Event Source == ~ARI~ | 050y
  BC Event Source == ~durn~ | 01h
  End Time == 3
  Output Folder == ..\Results\2d\050y
  1D Output Folder == ..\Results\1d\050y
End Define

Define Event == 050y_02h
  BC Event Source == ~ARI~ | 050y
  BC Event Source == ~durn~ | 02h
  End Time == 4
  Output Folder == ..\Results\2d\050y
  1D Output Folder == ..\Results\1d\050y
End Define
```

To run multiple events from the one .tcf file use a .bat file something like:

```
set TUFLOWEXE=C:\Program Files\TUFLOW\Releases\2010-10\iSP-w64\TUFLOW_iSP_w64.exe
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b
%RUN% -e 020y_01h Nile_~e~.tcf
%RUN% -e 020y_02h Nile_~e~.tcf
%RUN% -e 050y_01h Nile_~e~.tcf
%RUN% -e 050y_02h Nile_~e~.tcf
```

Example 2:

BC Database file:

```
Name,Source,Column 1,Column 2,Add Col 1,Mult Col 2,Add Col 2,Column 3,Column 4
C001,..\Inflows\Nile_~ARI~ ~durn~.csv,Time,C001
C002,..\Inflows\Nile_~ARI~ ~durn~.csv,Time,C002
...
```

Include “~e~” in the .tcf file, for example “Nile_~e~.tcf”, and add the following line:

```
Event File == Nile_Events.tef
```

The “Nile_Events.tef” file would contain lines such as the below:

```
!
! Global Default Settings
Start Time == 0 ! Unless Start Time is subsequently repeated all start at time 0

!
! ARI definitions
Define Event == 020y
  BC Event Source == ~ARI~ | 020y
  Output Folder == ..\Results\2d\020y
  1D Output Folder == ..\Results\1d\020y
End Define

Define Event == 050y
  BC Event Source == ~ARI~ | 050y
  Output Folder == ..\Results\2d\050y
  1D Output Folder == ..\Results\1d\050y
End Define
```

```
!  
! Storm duration definitions  
Define Event == 01h  
    BC Event Source == ~durn~ | 01h  
    End Time == 3  
End Define  
  
Define Event == 02h  
    BC Event Source == ~durn~ | 02h  
    End Time == 4  
End Define
```

To run multiple events from the one .tcf file use a .bat file something like:

```
set TUFLOWEXE=C:\Program Files\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_ISP_w64.exe  
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b  
%RUN% -e1 020y -e2 01h Nile_~e1~_~e2~.tcf  
%RUN% -e1 020y -e2 02h Nile_~e1~_~e2~.tcf  
%RUN% -e1 050y -e2 01h Nile_~e1~_~e2~.tcf  
%RUN% -e1 050y -e2 02h Nile_~e1~_~e2~.tcf
```


4.12.2 Modelling Different Scenarios Using [If Scenario](#)

The 2010-10 Release offers a new feature, [If Scenario](#), that controls which commands are to be applied depending on the scenario or combination of scenarios specified by the user. Using [If Scenario](#) is similar to using If...Else If... Else...End If constructs in a programming or macro language.

Example 1a:

In the example below from a .tgc file,

```
! _____
! Apply materials
Set Mat == 1 ! Set unspecified materials to default of 1
Read GIS Mat == shp\2d_mat_existing.shp ! Apply existing materials
If Scenario == opA
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A changes to existing materials
End If
```

TUFLOW will carry out the following steps:

- 1 Set all material values over the entire 2D domain to a value of 1.
- 2 Process the 2d_mat_existing.shp layer to assign material values from a layer of land-use polygons representing the existing situation.
- 3 One of the following will then occur:
 - (i) If either “-s opA” was specified as a run option, or “[Model Scenarios](#) == opA” was specified in the .tcf file, then the 2d_mat_opA.shp layer is processed to modify the material values where Option A has changed the landuse. If “~e~” or “~e1~” occurs within the .tcf filename, it is replaced by “opA” in the output filenames, otherwise “opA” is appended to the .tcf filename for the output filenames.
 - (b) If opA was not specified, this layer would be ignored as would, for example, be required if modelling the existing scenario.

Example 2a:

Extending Example 1 above to include an Option B, where Option B is a modified version of Option A we could use the following.

```
Set Mat == 1 ! Set unspecified materials to default of 1
Read GIS Mat == shp\2d_mat_existing.shp ! Apply existing materials
If Scenario == opA | opB
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A changes to existing materials
End If
If Scenario == opB
    Read GIS Mat == shp\2d_mat_opB.shp ! Option B changes to Existing and Option A
End If
```

In the above example, either “-s opB” or “-s1 opA -s2 opB” could be used for the run options when modelling the Option B scenario. An advantage of using the latter is the output files would be named “Nile_opA+opB” indicating that the simulation was for Option A plus Option B (which is the case as

Option B builds upon Option A). If using [Model Scenarios](#), the equivalent for “-s1 opA -s2 opB” would be “[Model Scenarios](#) == opA | opB”.

Example 2b:

However, if Option B was not a modification of Option A, but purely a change to the existing case we could use something like:

```
Set Mat == 1 ! Set unspecified materials to default of 1
Read GIS Mat == shp\2d_mat_existing.shp ! Apply existing materials
If Scenario == opA
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A changes to existing materials
End If
If Scenario == opB
    Read GIS Mat == shp\2d_mat_opB.shp ! Option B changes to existing materials
End If
```

or more elegantly the following would give the same result:

```
Set Mat == 1 ! Set unspecified materials to default of 1
Read GIS Mat == shp\2d_mat_existing.shp ! Apply existing materials
If Scenario == opA
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A changes to existing materials
Else If Scenario == opB
    Read GIS Mat == shp\2d_mat_opB.shp ! Option B changes to existing materials
End If
```

In the above, either “-s opB” or “[Model Scenarios](#) == opB” would be used when modelling Option B.

Example 2c:

If the Existing, Option A and Option B scenarios all had their own layer of materials for the whole model you could use something like the below noting the use of the Else option which forces the use of the layer 2d_mat_existing.shp if opA or opB is not specified.

```
Set Mat == 1 ! Set unspecified materials to default of 1
If Scenario == opA
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A materials for whole model
Else If Scenario == opB
    Read GIS Mat == shp\2d_mat_opB.shp ! Option B materials for whole model
Else
    Read GIS Mat == shp\2d_mat_existing.shp ! Existing materials for whole model
End If
```

Or, a more explicit approach to the above would be:

```
Set Mat == 1 ! Set unspecified materials to default of 1
If Scenario == exg
    Read GIS Mat == shp\2d_mat_existing.shp ! Existing materials for whole model
Else If Scenario == opA
    Read GIS Mat == shp\2d_mat_opA.shp ! Option A materials for whole model
Else If Scenario == opB
    Read GIS Mat == shp\2d_mat_opB.shp ! Option B materials for whole model
End If
```

In the example above, to batch run the Existing, Option A and Option B scenarios the .bat file would be something like:

```
set TUFLOWEXE=C:\Program Files\TUFLOW\Releases\2010-10\isp-w64\TUFLOW_isp_w64.exe
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b
%RUN% -s exg Nile_~s~.tcf
%RUN% -s opA Nile_~s~.tcf
%RUN% -b -s opB Nile_~s~.tcf
```

4.13 Data Processing

4.13.1 Hierarchy

The order in which data are read and processed is provided below. The list is not exhaustive, but provides a guide to the data input process.

- 1 Read parameters (variables, options, etc) from .tcf file.
- 2 Read any material data from .tmf file.
- 3 Open .tgc file:
 - (a) Locate and orientate 2D grids.
 - (b) Allocate memory for 2D domains.
- 4 Identify any 2d_fc flow constrictions (FC).
- 5 Identify any 2d_po (PO) time-series output.
- 6 Identify any 2d_lp (LP) time-series output.
- 7 Open .tbc file:
 - (a) Read through 2D boundary conditions.
 - (b) Allocate memory for 2D boundary conditions.
- 8 Return to .tgc file:
 - (a) Process instructions in .tgc file to build 2D domains.
- 9 Finish reading information from .tcf file, including:
 - (a) Read any 2d_fc flow constrictions (FC).
 - (b) Read any 2d_po (PO) time-series output.
 - (c) Read any 2d_lp (LP) time-series output.
- 10 Open .ecf file (if 2D/1D model):
 - (a) Read parameters (variables, options, etc) from .ecf file.
 - (b) Read all 1D node information from all 1d_nwk layers.
 - (c) Read all WLL information from all 1d_wll layers.
 - (d) Read all table link information from all 1d_tab layers.
 - (e) Read all 1D channel information from all 1d_nwk layers.
 - (i) Any channel cross-section information from 1d_tab table links or external sources (eg. MIKE 11 .txt or ISIS .pro files) are processed.
 - (ii) Any bridge loss coefficient tables from 1d_tab table links are processed.
 - (iii) Parameters for any culverts or weirs written to temporary file (_cnch.tmp).
 - (f) Read all table link information for 1D node storages (NA tables) from all 1d_tab layers.
 - (g) Process any initial water level commands (eg. Read 1d_iwl layers).

- (h) Read any fixed field CS tables.
 - (i) Read any fixed field NA tables.
 - (j) Allocate memory for topography.
 - (k) Read any boundary condition data and allocate memory.
 - (l) Read any structure parameters and allocate memory.
 - (m) Repeat above and retain information in memory.
 - (n) Carry out pre-processing and checking tasks.
- 11 Read 2D boundary conditions into memory and write 2d_bc check file.
 - 12 Carry out pre-processing and check tasks:
 - (a) Unit conversions
 - (b) Pre-process linear and cubic spline interpolation tables.
 - (c) Initialise variables and arrays and carry out checks.
 - (d) Set any flow constriction (FC) obverts on 2D cells.
 - (e) Set starting wet/dry flags or read initial conditions from a restart file.
 - 13 Write 2d_grd and 2d_zpt check files.
 - 14 Write .2dm file.
 - 15 Automatically create any 1D HX boundaries from 2D SX boundaries and check.
 - 16 Automatically create any 1D QX boundaries from 2D HX boundaries and check.
 - 17 Start simulation.

4.13.2 XF Files

XF Files are a new feature in the 2010-10 release. They are binary dumps of selected input files automatically created by TUFLOW to vastly speed up the start-up process next time a simulation is carried out. If the original input data does not have a save date later than its corresponding XF file, the XF file is used in preference to the original data.

XF files are always written to a folder named “xf” underneath the folder where the original data resides. If the “xf” folder or a XF file is deleted, the XF files will be recreated next time a simulation is started. Two extensions are used: .xf4 and .xf8. .xf4 refers to iSP (single precision) runs and .xf8 to iDP (double precision) runs.

To globally switch XF files off use [XF Files](#) == OFF. They can also be switched off for individual inputs using “XF OFF” in the command (for example, see [Read GIS Z Shape](#)).

The 2010-10 release has incorporated XF files for [Read RowCol Zpts](#), and for TINs generated using [Create TIN Zpts](#), [Read GIS Z Shape](#) and [Read GIS Variable Z Shape](#). XF files are planned to be added to other commands that read and/or generate large data sets in future builds/releases.

As of Build 2010-10-AB, commands that generate XF files (eg. [Read RowCol Zpts](#)), can directly refer to the .xf file (ie. instead of referring to the GIS layer).

4.14 UltraEdit

UltraEdit (www.ultraedit.com) is recommended as the text editor for TUFLOW text files. UltraEdit has many excellent features, of which a few are noted here or see Section 12.2.

- 1 A file “Wordfile.txt” is provided with TUFLOW in the UltraEdit folder. Replace the equivalent file in the UltraEdit installation folder (typically “C:\Program Files\UltraEdit”) with the one provided with TUFLOW. UltraEdit will now colour code TUFLOW text files. (Note: If you have modified the UltraEdit Wordfile.txt file for your own purposes, you will have to merge the two files.) You can change the colours in UltraEdit via Advanced, Configuration, Syntax Highlighting menus.
- 2 UltraEdit has a very useful feature that allows opening of a file that is specified in the active text file. Place the pointer anywhere over the text of the file you wish to open and click the right mouse button. The top menu item on the pop-up menu will open the file.
- 3 TUFLOW simulations may be initiated from UltraEdit (see Section 5.5).

5 Running TUFLOW

Section Contents

5	RUNNING TUFLOW	5-1
5.1	Installing a Dongle	5-2
5.1.1	BMT WBM Dongles	5-2
5.1.2	Protocols for Accessing Dongles	5-3
5.1.3	"C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File	5-3
5.1.4	WIBU Codemeter (Metal) Dongles (Local and Network)	5-4
5.1.4.1	Installing Codemeter Software	5-4
5.1.4.2	Codemeter Features	5-7
5.1.4.3	Requesting a Licence Change	5-8
5.1.5	SoftLok (Blue) Standalone (Local) Dongle	5-11
5.1.6	SoftLok (Blue) Network Dongle	5-11
5.1.6.1	SoftLok Dongle Security Server	5-12
5.1.6.2	Client Computers	5-14
5.1.7	Dongle Failure During a Simulation	5-15
5.2	TUFLOW.exe and .dll Files	5-16
5.2.1	Using TUFLOW with ISIS or XP-SWMM, or from SMS	5-19
5.3	TUFLOW.exe Options (Switches)	5-20
5.4	via Right Mouse Button in Microsoft Explorer	5-23
5.5	From UltraEdit	5-25
5.6	Batching Simulations using a Batch File	5-28
5.6.1	Simple Example and Switches	5-28
5.6.2	Windows NT/2000/XP/7 Priority Levels	5-29
5.6.3	Advanced .bat Files	5-30
5.7	From a Console (DOS) Window	5-31
5.8	The Console (DOS) Window Does Not Appear!	5-31
5.9	Customising TUFLOW using TUFLOW_USER_DEFINED.dll	5-31
5.10	ESTRY.exe	5-32

5.1 Installing a Dongle

A TUFLOW dongle is required to run TUFLOW, but is not required when using third party software such as a GIS, text editor or SMS.

As of 2010, two brands of dongles are available from BMT WBM, and several third-party dongles are supported that are licensed and maintained by on-sellers of TUFLOW. For the latter, please refer to the vendor's documentation for setting up the dongle.

5.1.1 BMT WBM Dongles

BMT WBM dongles are either:

- A SoftLok (blue) dongle (see Section 5.1.5 for Standalone Dongles and Section 5.1.6 for network dongles). These dongles were stopped being issued in August 2010 as the vendor after several requests could not provide 64-bit support, but they will continue to be supported by future 32-bit versions of TUFLOW. Note, that they only function with 32-bit versions of TUFLOW. SoftLok dongles can be exchanged for a new WIBU (metal) dongle for a nominal fee (please contact sales@tuflow.com).
- A WIBU Codemeter (metal) dongle (see Section 5.1.4). These dongles were introduced in 2010. They offer a range of advantages over the SoftLok dongles such as:
 - support for 64-bit platforms and future options for non-Windows platforms;
 - network licence manager runs as a service (ie. the computer with the network dongle needs to be on, but no one needs to be logged in);
 - much more flexible licensing options (for example, it is now possible to have a Network 10 TUFLOW licence with a Network 5 Multiple 2D Domains Module licence – with the SoftLok dongles a Network 10 Multiple 2D Domains Module would have to have been purchased);
 - multiple WIBU dongles (local and/or network licensed) are accessible together (ie. if all licences from one dongle are taken, licences from other dongles are automatically checked and taken);
 - no network ghost licences (thus far!); and
 - no need for TUFLOW to control limiting of local licences (TUFLOW's run key is not used if the number of CPUs/cores exceeds the local licence limit).

For the 2009-07, 2008-08, 2007-07 and 2006-06 releases, the “DB” builds or later will need to be used to recognise a WIBU Codemeter (metal) dongle.

For 64-bit versions of TUFLOW, a WIBU Codemeter (metal) dongle needs to be used, or if 64-bit supported, a third-party dongle. SoftLok (blue) dongles cannot be used for 64-bit TUFLOW builds, except for Build 2009-07-AF-iDP that used a workaround for standalone dongles only (it is not recommended to use this 64-bit build unless the same results occur using a 32-bit 2009-07 build as this 64-bit build has known bugs).

Two types of licences are provided for both brands:

- Local or Standalone Licence – by default, allows up to four (4) TUFLOW processes to be run from the one computer. Less and more than four simulations can be programmed (contact sales@tuflow.com).
- Network Licence – allows up to a specified limit the number of TUFLOW processes running at any one time across the organisation's LAN (Local Area Network).

5.1.2 Protocols for Accessing Dongles

If more than one type of dongle is available the protocols for taking and checking licences are:

- 1 WIBU Codemeter dongles are searched first, and if a licence is free it is taken.
- 2 If no WIBU licence is available, a licence from a SoftLok dongle is sought.
- 3 If no SoftLok licence is available, you can optionally set via the "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" file described below, for TUFLOW to continue to try and find an available WIBU dongle licence. This is useful if there are no free licences and you wish to start a simulation (the simulation will start once a free licence becomes available).
- 4 Once a simulation is underway, and if the licence is lost, TUFLOW will try to regain a licence, but it cannot switch to a different dongle brand.

5.1.3 "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File

The "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" file can be used to set the retry interval for searching or reinstating a licence and other settings. It has the same form and notation as a TUFLOW .tcf file.

```
! Use this file to set general and WIBU specific dongle parameters
! Use ! or # to comment out commands or make comments
Simulations Log Folder == C:\BMT_WBM\log ! Path or URL to global .log file
WIBU Retry Time == 60 ! seconds. Values less than 3 are set to 3. Default = 60.
WIBU Retry Count == 0 ! Use -1 for indefinitely.
WIBU Dongles Only == OFF ! If ON, searches for WIBU dongles only. Default is OFF.
```

"Simulations Log Folder" sets the folder path or URL to a folder for logging all simulations. If the keywords "DO NOT USE" occur within the folder path or URL, this feature is disabled. The -slp option to set the Simulation Log Folder on a SoftLok dongle (see Table 5.2) is not available for WIBU dongles. Also see [Simulations Log Folder](#).

"WIBU Retry Time" sets the interval in seconds for retrying to take a licence or regain a lost licence. The default is 60 and values less than 3 are set to 3.

"WIBU Retry Count" sets the number of times to retry for a licence at the start of a simulation. If a licence is lost during the simulation, TUFLOW tries indefinitely to regain a licence so as not to lose the simulation.

"WIBU Dongles Only" if set to ON will force TUFLOW to only search for WIBU dongles.

5.1.4 WIBU Codemeter (Metal) Dongles (Local and Network)

5.1.4.1 Installing Codemeter Software

Download the Codemeter install exe using one of the links below, and run the appropriate exe on each computer wishing to access the dongle. Note: check http://www.tuflow.com/Downloads_Misc.htm for later versions than the links to v4.20 below.

Windows 32-bit Platforms:

<http://www.tuflow.com/Downloads/WIBU/CodeMeterRuntime32.v4.20.exe>

Windows 64-bit Platforms:

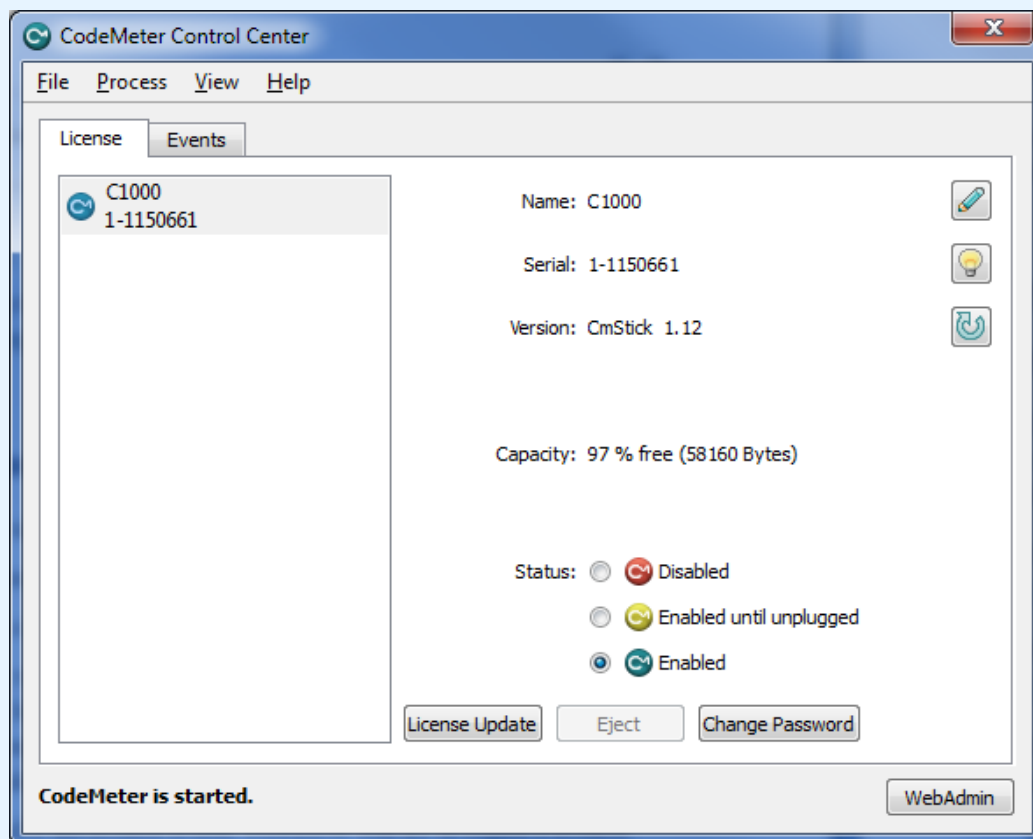
<http://www.tuflow.com/Downloads/WIBU/CodeMeterRuntime64.v4.20.exe>

Once installed, a Codemeter icon  should appear in the system tray. The icon changes colour and appearance depending on the number of dongles attached to the computer as discussed further below.

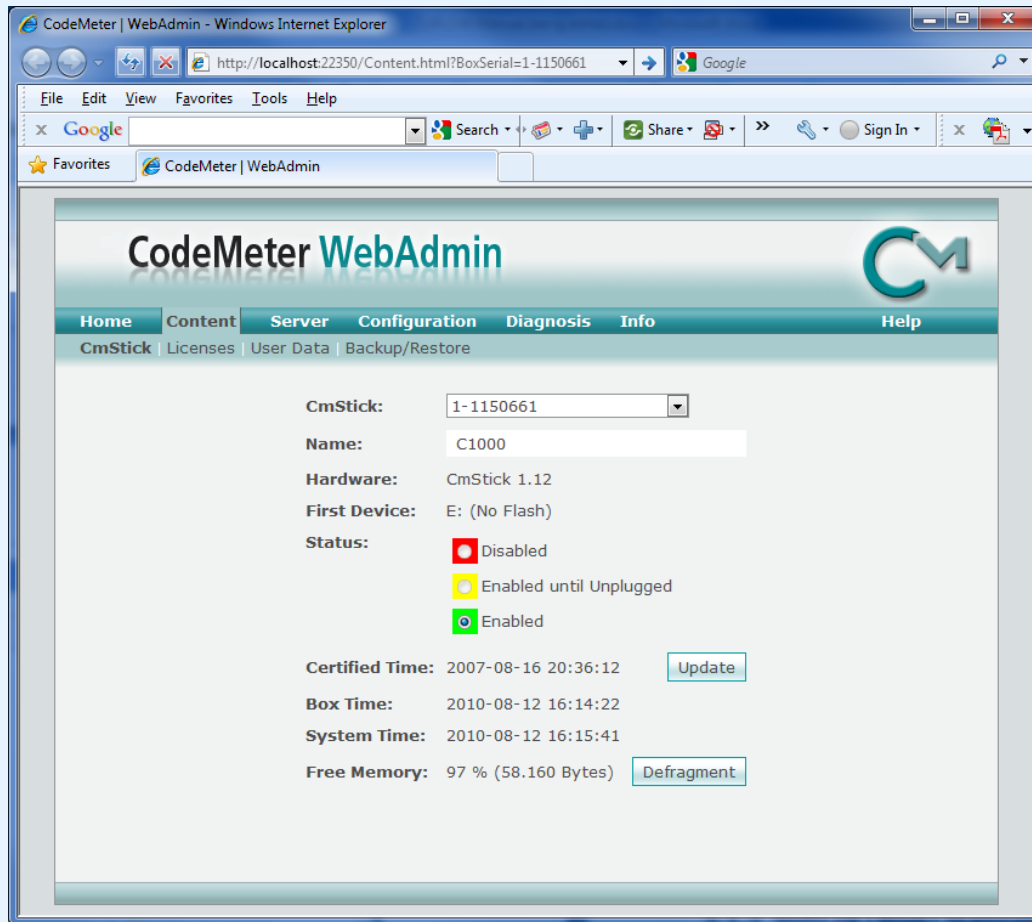
For Local licences, double click on TUFLOW.exe to check that the licensing on the dongle is picked up. Note for 2009-07, 2008-08, 2007-07 and 2006-06 TUFLOW releases, the “DA” builds or later will need to be used to recognise a WIBU Codemeter (metal) dongle.

For Network licences, the Codemeter will need to be setup as a service using the following steps:

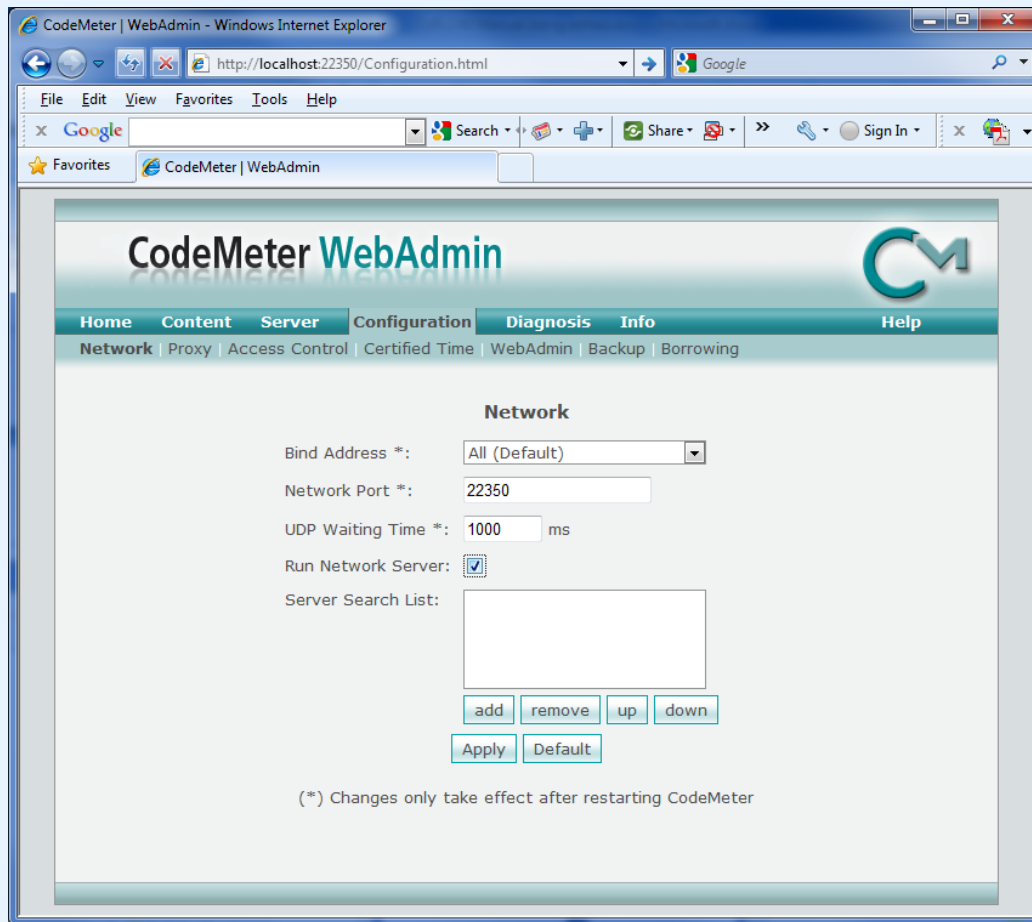
- 1 On the machine with the Network dongle attached, click on the Codemeter icon  in the system tray to bring up the Codemeter Control Center dialog below.



- 2 The dongle should appear under the License tab.
- 3 Click WebAdmin (WebAdmin may also directly be accessed by right clicking on the Codemeter icon in the system tray). This will open your preferred internet browser and appear something like the below.



- 4 Go to the Configuration tab (see below) and tick “Run Network Server”, then click Apply.



- 5 The network dongle should now be accessible by other computers. Note, the above process only needs to be run on the computer with the network dongle attached.

5.1.4.2 Codemeter Features

Microsoft Windows platforms supported include Windows 7, Vista, XP, 2000 and Server 2000/2003/2008. Non-windows platforms supported by Codemeter dongles include Mac OS X, a range of Linux platforms, and Sun Solaris, and are being investigated for use by TUFLOW.

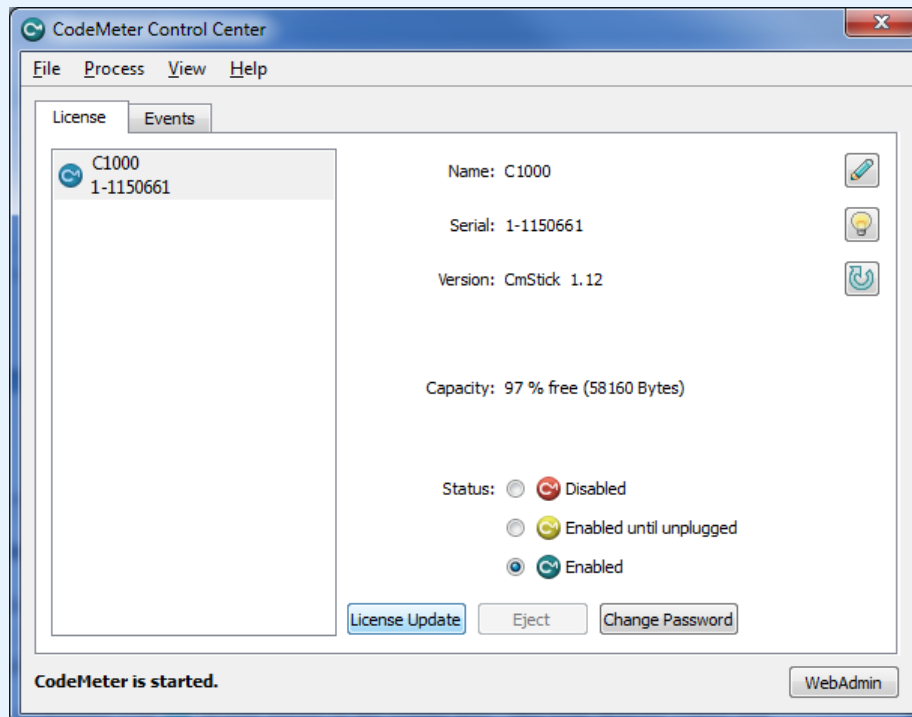
Once installed, a Codemeter icon  should appear in the system tray. The icon changes colour and appearance depending on the number of dongles attached to the computer.

	Green CodeMeter icon		An enabled Codemeter dongle is connected.
	Gray CodeMeter icon		No Codemeter dongles are attached.
	Yellow CodeMeter icon		A Codemeter dongle is connected, which is enabled until it is un-plugged.
	Red CodeMeter icon		A disabled Codemeter dongle is connected.
	Doubled blue CodeMeter icon		Multiple Codemeter dongles are connected.

5.1.4.3 Requesting a Licence Change

To request a change to your WIBU licence please contact sales@tuflow.com. Once the change/upgrade has been agreed upon, follow the steps below and email the update .WibuCmRaC file.

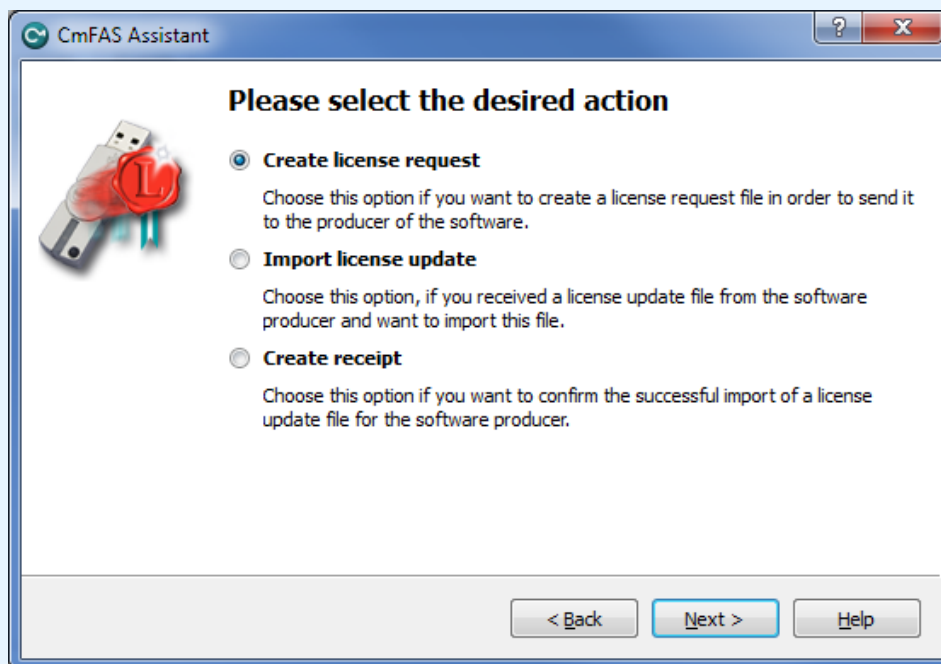
1. Click on the Codemeter icon in the system tray to bring up the dialog below (make sure the dongle is plugged in). Your dongle should appear in the dialog below.



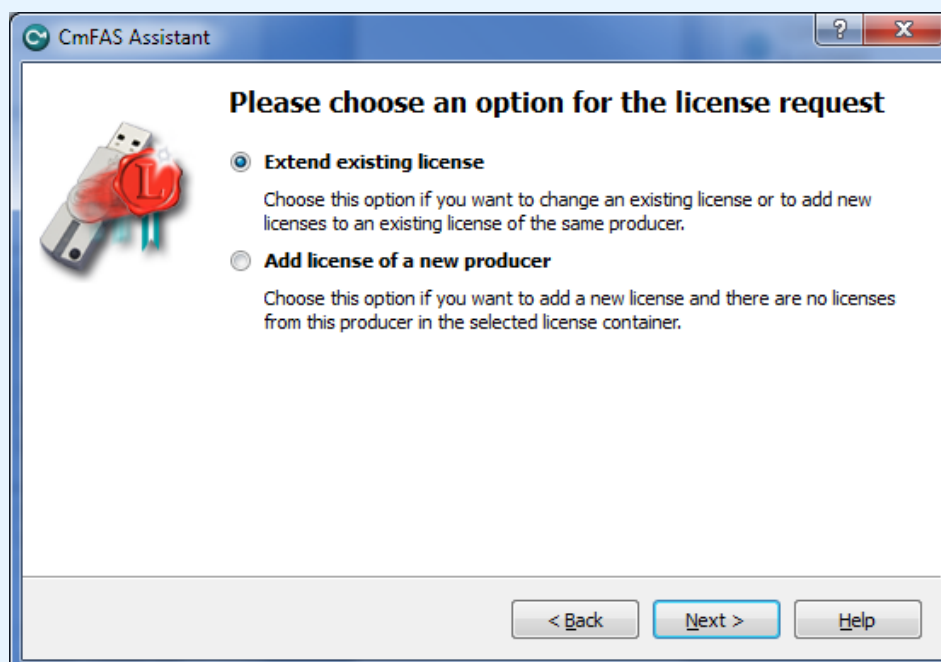
2. Click on "License Update" to bring up the below.



3. Click Next >, then select “Create License Request” in the below.



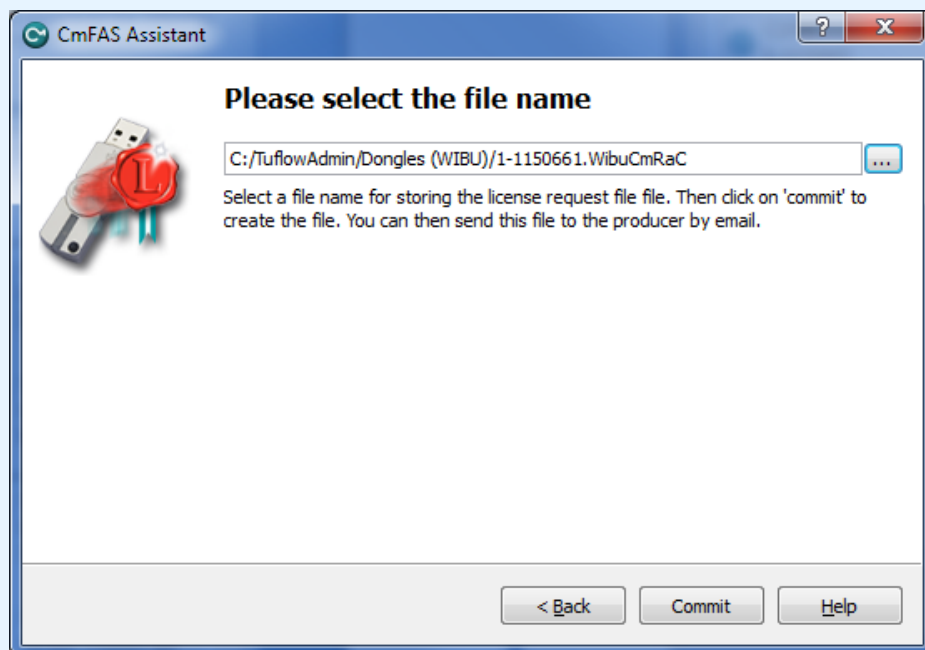
4. Click Next > then select “Extend existing license” in the below.



5. Click Next > and make sure BMT WBM (101139) is selected in the next dialog.



6. Click Next > to bring up the below. Change if you wish to (and remember the location) of the filepath and click Commit.



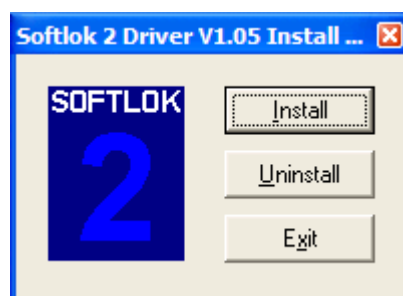
7. Email the .WibuCmRaC file to sales@tufLOW.com. We'll then email back another file for you that will update your dongle for a trial period.

5.1.5 SoftLok (Blue) Standalone (Local) Dongle

To install the SoftLok software drivers that TUFLOW uses to access the dongles, download the latest drivers from http://www.tuflow.com/Downloads_Misc.htm. Make sure that “Dongle Drivers.003.zip” or later is used as there are known problems with earlier drivers. Unzip the file and follow the steps below.

To allow access to a SoftLok dongle the computer’s operating system requires a device driver that interfaces between TUFLOW and the dongle.

Install the device driver by running `sl2inst.exe` found in the `Dongle\Drivers\Install` folder. Depending on your permissions, you may need to be logged in as an Administrator. The following dialogue appears:



Click on “Install” to install the drivers. “Uninstall” is used to remove the drivers if you need to at a later date.

Alternatively, a command line version `drvinst.exe` is also provided. Type “`drvinst install`” from a DOS window to install, or “`drvinst uninstall`” to remove the drivers.

Please note if you move either of these programs to a different folder ensure to copy both of the driver files `windrvr.vxd` and `windrvr.sys` to the same folder.

It is not necessary to reboot the computer – the drivers are available for use immediately.

Insert the TUFLOW SoftLok standalone dongle in the USB port. This allows you to initiate several TUFLOW processes from this computer (subject to the licensing of your Standalone dongle, and computer hardware and processor limitations).

If a TUFLOW SoftLok network dongle is also available across your network, the standalone dongle takes precedence. That is, running TUFLOW from a computer with a standalone dongle does not take up a TUFLOW network dongle user site.

5.1.6 SoftLok (Blue) Network Dongle

Note: For the computer with the network dongle attached and for all computers accessing the network licence, first install the device drivers as described above in Section 5.1.5 SoftLok (Blue) Standalone (Local) Dongle.

Note: If setting up a computer on the network to act as the host or server for the network dongle read Section 5.1.6.1. If setting up a computer that wishes to access the network dongle on the dongle server read Section 5.1.6.2.

5.1.6.1 SoftLok Dongle Security Server

Installation of a network dongle requires one computer on your network to be designated as the dongle security server. This computer does not have to be the network server, but must be accessible by any other computer you plan to run TUFLOW on, and must be running Windows 95, 98, NT or 2000. It must remain on and connected to the network at all times you wish to run TUFLOW, and have a parallel port.

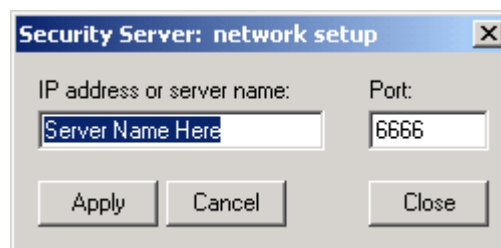
Install the dongle security server software by running `setup.exe` in the `Dongle\Server` folder and follow the on screen prompts. If you are running Windows NT or 2000, you must be logged in as Administrator. The required device drivers as described in the previous section are automatically installed.

Connect the TUFLOW network dongle to the USB port and start the dongle driver by selecting it from the Start menu (Start/Programs/Security Server/Security Server). The following dialog box appears.



The first time the dongle security server software is run the program attempts to start for a few seconds and then fails because it has not yet been set up. Click the Set up button and enter the server name or IP address (IP address is preferred as problems with server names that start with a number have been experienced), usually leave the port as 6666 (see note below) and click Apply and Close.

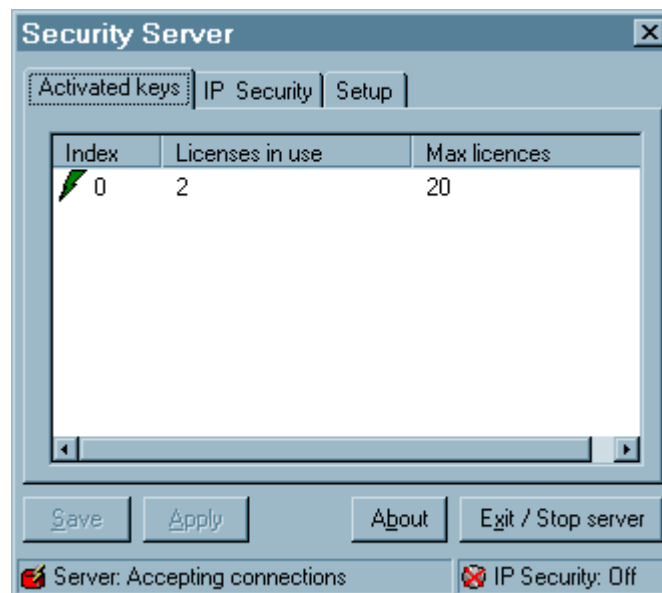
Note: Some System Administrators have discovered that particular firewall and anti-virus software block port 6666 (popular amongst hackers apparently!). If problems occur in accessing a network dongle, try using another port number. The port number may also need to be freed by your organisation's IT department.



Now click the "Retry to start server" button, which should successfully start the server. (Note: there seems to be a problem with starting the server on machines with a name starting with a number. Therefore, avoid these names or use the IP address.) Don't worry about the message "IP Security not

loaded” as this is set up later if required. Now click OK to place the Security server in the system tray (icons at bottom right of taskbar).

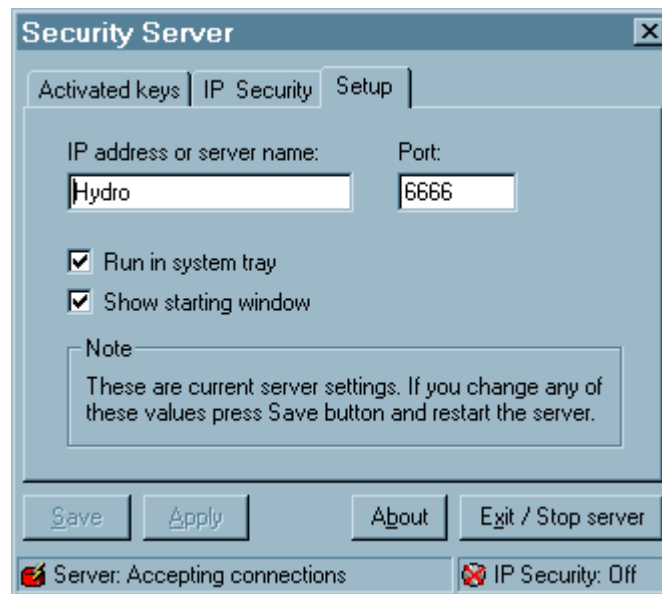
You can at any time double click the Security Server icon to either change the settings or to view the current status as shown below. Note that if the below is opened before any TUFLOW processes have been initiated, the Security Server shows no active keys (ie. the below is blank). In this case, to check if the Security Server is working double click on TUFLOW.exe and one License in use should eventually appear along with the maximum number of licenses available.



To close the dialog box and return it to the system tray click the close dialog button in the top right. To stop the server click the Exit/Stop Server button. The Activated keys tab shows the current activity of the Security Server. The 'Index' column indicates whether the dongle is being accessed. The 'Licenses in use' column indicates how many TUFLOW processes are currently using the network dongle. The 'Max Licences' column shows the maximum number of licenses that can be allocated by the dongle.

As of Build 2007-07-AA, the Security Server can be exited and restarted while there are simulations (of this build or later) underway. This allows any ghost licences to be cleared (ghost licences are unreleased licences that occur on some networks when the network is disconnected from a computer running a TUFLOW simulation). TUFLOW Build 2007-07-AA or later enters a 3 minute holding pattern until a spare network licence becomes available.

By default when first installed the dongle security server accepts TUFLOW processes from any other computer (or IP address). If you wish to control which computers can run TUFLOW this can be set up from the IP Security tab. Further documentation on this feature can be supplied upon request.



The Setup tab allows you to change the IP address or name of the TUFLOW dongle server and also the port. Normally port 6666 is used, but other ports may need to be tried as noted previously. If the server name or port is changed, all computers that run TUFLOW also need to be changed as described in the following section.

Checking the 'Run in system tray' box allows the security server to run in the system tray.

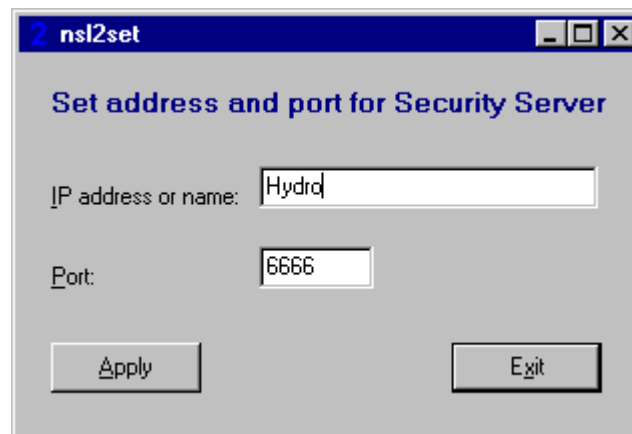
Checking the 'Show starting Window' box displays a dialog box when the server starts. **The usual procedure is to set the dongle security server to start automatically by placing it in the startup folder of the server.** This ensures that the dongle security server is always started in case the server is rebooted. In this case you can clear 'Show starting Window' checkbox to cause it to run in the system tray only.

Click 'Apply' and/or 'Save' for any changes you make on the Setup tab. The Security Server may need to be exited and restarted to effect the changes.

5.1.6.2 Client Computers

For the server or any other computer on the network to access the network dongle, the following steps are required:

- 1 Follow the steps outlined for a Standalone Dongle (if not already done) in Section 5.1.5.
- 2 Run `ns12set.exe` under `Dongle\Drivers\NetSet` to display the dialogue below.



- 3 Enter the name or IP address of the dongle server and the Port (this must be the same as set for the server – default is 6666).
- 4 Click Apply and Exit.

You are now ready to access the network dongle from your computer. To test if the network licence is working, double click on TUFLOW.exe. The console window displays whether or not it can find a Standalone or Network dongle, and the modules valid for the licence.

5.1.7 Dongle Failure During a Simulation

For Build 2007-07-AA or later, if TUFLOW fails to recognise the network dongle during a simulation (eg. the network dongle server computer is down) it enters a holding pattern and retries every three minutes until a license is found. Prior to Build 20078-07-AA, TUFLOW prompts for the Security Server to try and be restarted.

For standalone dongles, TUFLOW prompts with a message that the dongle could not be found. Check the dongle and/or try a different USB port, and press Enter to continue.

Prior to Build 2004-03-AB, a dongle error sometimes occurred if two simulations on the same computer tried to access the dongle at exactly the same time. In this situation, Build 2004-03-AB, rather than report a dongle error, will pause for 6 seconds and retry for up to 100 times. This overcomes the problem of two simulations accessing the same dongle at the same time.

5.2 TUFLOW.exe and .dll Files

The 2010-10 TUFLOW release consists of four different versions as follows, with the acronyms explained in Table 5.1.

- TUFLOW_iSP_w32.exe
- TUFLOW_iSP_w64.exe
- TUFLOW_iDP_w32.exe
- TUFLOW_iDP_w64.exe

Each version has four TUFLOW .dll files, and several system .dll files:

- TUFLOW_LINK_X.dll
- TUFLOW_USER_DEFINED_X.dll
- TUFLOW_AD_X.dll
- TUFLOW_MORPHOLOGY_X.dll
- Several system .dll files. These are supplied as these may not exist on your computer – do not delete or relocate these dlls - keep them with the TUFLOW .exe/.dll files.

For each version all .exe and .dll files must be placed in the same folder and kept together at all times. When replacing with a new build, archive the files by creating a folder of the same name as the Build ID (eg. 2010-10-AA-iSP-w32), and place all files in this folder.

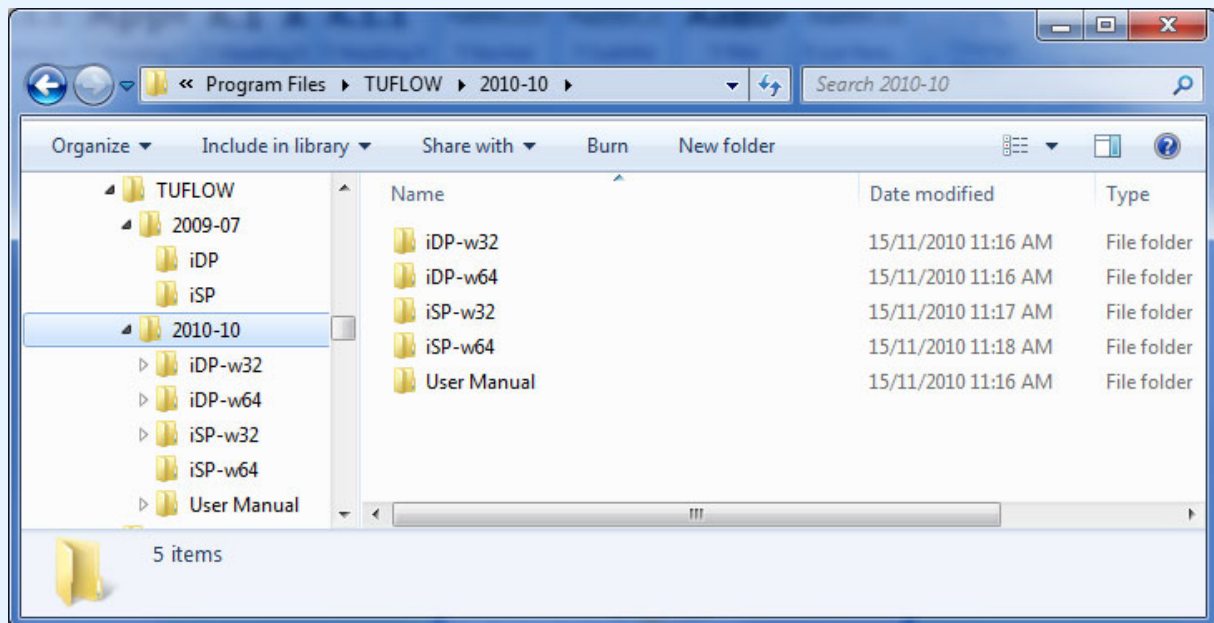
The Build ID includes the acronym and appears in the top bar of the Console window, in the .tlf file and elsewhere. For example, if running the single precision, Windows 64-bit version, the Build ID would be “2010-10-AA-iSP-w64”.

Older builds can always be setup in a similar manner by placing the build in its own folder, if they are needed for running old models.

For the 2010-10 release and also for the DB or later builds of the 200-06, 2007-07, 2008-08 and 2009-07 releases includes the following .tcf commands to ensure a model is run with the correct build.

[Model TUFLOW Build](#) or a combination of [Model TUFLOW Release](#), [Model Precision](#) and [Model Platform](#) can be used to force a simulation to use a specific TUFLOW build, release, iSP or iDP, or w32 or w64. This can be very useful for ensuring the correct TUFLOW build/release/version is used.

It is recommended that for the 2010-10 release that a folder structure such as shown below is used, noting the w32, w64, iSP and iDP folders for each of the four versions supplied. If an update (patch) is issued, archive the old build by moving all files to a separate folder or .zip file, and copy in the new TUFLOW files. This way, UltraEdit or other programs used to initiate TUFLOW will access the latest build.



Alternatively, you can place the iSP and iDP versions in the same folder. Separate folders are still required for w32 and w64 versions as some of the system dlls have the same name for both 32-bit and 64-bit platforms, so they need to be kept separate. In this case the folder structure would look something like:

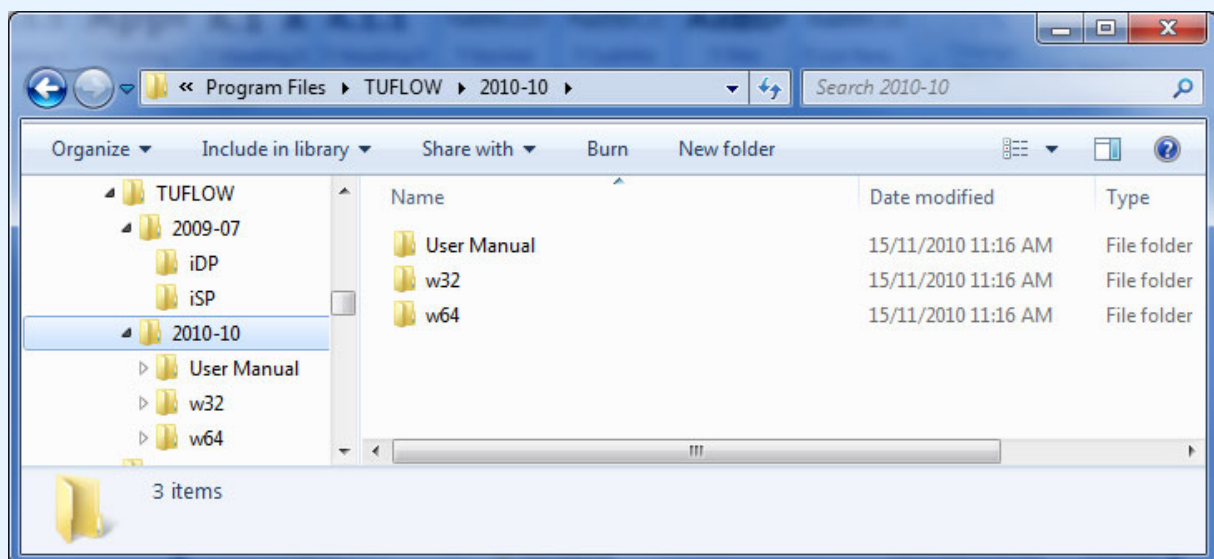


Table 5.1 2010-10 TUFLOW Versions (iSP, iDP, w32, w64)

Version	Description
Intel Compiler Single Precision (iSP)	Compiled using the Intel Fortran Compiler Version 11 using single precision (4-byte) real numbers that typically have around 7 significant figures. This version is recommended for the majority of TUFLOW simulations. Due to switching from Version 10 to Version 11 of the Fortran compiler, very slightly different results (usually fractions of a mm) compared with the 2008-08 and 2009-07 releases can result over the majority of the model if using the same default settings. Whilst this is of no major concern, this effect can cause a cell on the edge of the floodplain to be wet in one version and dry in another, so more pronounced effects may occur around the perimeter of the flooded area.
Intel Compiler Double Precision (iDP)	Compiled using the Intel Fortran Compiler Version 11 using double precision (8-byte) real numbers that typically have around 14 significant figures. This version should be used for models using high ground elevations (greater than 100 to 1,000m), and for most direct rainfall models. If a model falls into these categories, and is experiencing mass errors when using the single precision version, this double precision version should be used. Model Precision can be used to ensure that a simulation is started using the single or double precision version.
Windows 32-bit (w32)	Compiled as a 32-bit process. w32 TUFLOW builds can be run on Windows 32-bit and 64-bit platforms. All builds/releases prior to the 2010-10 release (except for the 2009-07-XX-iDP-64 build) are 32-bit.
Windows 64-bit (w64)	Compiled as a 64-bit process. w64 TUFLOW builds can only be run on Windows 64-bit platforms. w64 2010-10 builds have been found to reduce simulation times by 20 to 50% compared with the 2009-07 (32-bit) release. Slightly different results (mostly fractions of a mm) will occur between w32 and w64 for the same model. Model Platform can be used to force a simulation to use a w32 or w64 version.
Compaq Compiler Single Precision (cSP)	The Compaq Fortran compiled version has been discontinued and is not supplied with the 2010-10 release.

Running TUFLOW is carried out by initiating the TUFLOW_X.exe file using one of the approaches in the following sections.

TUFLOW LINK X.dll allows other schemes such as ISIS and XP-SWMM to dynamically link with TUFLOW. **Note that for ISIS and XP-SWMM versions that were issued using the 2009-07 release, the 2010-10 DLLs are compatible but may need to be renamed without the iSP/iDP and w32/w64 extensions (ie. renamed to be the same name as per the 2009-07 release).**

TUFLOW_USER_DEFINED_X.dll allows users to customise TUFLOW to suit their purposes. See Section 5.9.

TUFLOW_AD_X.dll contains the AD (advection-dispersion) module algorithms.

TUFLOW_MORPHOLOGY_X.dll contains the morphology module algorithms.

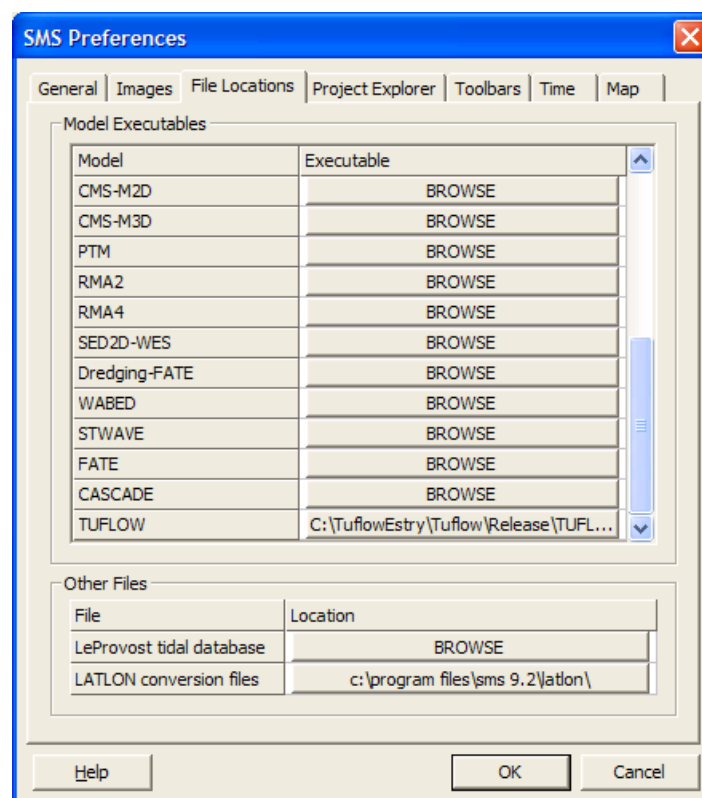
DFORRT.DLL, NSLMS324.DLL and any other supplied .dlls are system DLLs required by TUFLOW.

As a general rule, once a build is over three years old, it is no longer supported. It can still be used, but there is no guarantee that older builds recognise newer dongles, or are updated with bug fixes. A stock of old dongles is kept in supply and available for rental should older unsupported builds need to be used.

5.2.1 Using TUFLOW with ISIS or XP-SWMM, or from SMS

The ISIS-TUFLOW and XP-SWMM executables access the TUFLOW hydraulic computational engine via the TUFLOW_LINK.dll. To utilise a new version of TUFLOW with these engines, all of the .dll files described in the previous section need to be copied to the location where ISIS or XP SWMM access them (usually in the same folder as the ISIS-TUFLOW and XP-SWMM .exe files). Alternatively, future versions of these software may offer a path or environment variable to allow the user to point to the location of the TUFLOW .dll files. They do not access the TUFLOW.exe file, although there are no issues in copying this file as well. Note, it is always wise to keep copies of any old .dll files that are to be replaced.

When running TUFLOW from SMS, SMS by default looks for a TUFLOW.exe in the installation folder. To change this to the location where you have placed the TUFLOW.exe and .dll files, go to the Edit, Preferences, File Locations tab as shown below.



5.3 TUFLOW.exe Options (Switches)

TUFLOW.exe has several switches or options used to control how TUFLOW functions. Options must be placed between TUFLOW.exe and the .tcf filename. Note, for builds prior to the DB builds for 2006-06, 2007-07, 2008-08 and 2009-07 only the “-” (short dash or minus sign) can be used.

Table 5.2 describes the switches. The switches are also displayed to the console if TUFLOW.exe is executed without any arguments (ie. double click on TUFLOW.exe in Windows Explorer).

As of the 2010-10 release, options can now be prefixed by a:

- “-” (short dash or minus sign);
- “_” (long dash); or
- “/” (forward slash)

For example, to copy a model, one of the lines below would be used:

```
"C:\Program Files\TufLOW\TufLOW.exe" -c "My Model.tcf"
"C:\Program Files\TufLOW\TufLOW.exe" -c "My Model.tcf"
"C:\Program Files\TufLOW\TufLOW.exe" /c "My Model.tcf"
```

Table 5.2 TUFLOW.exe Options (Switches)

Switch	Description
-b	Batch mode. Used when running two or more simulations in succession from a .bat file (see Section 5.6). Suppresses display of the TUFLOW simulation has finished dialogue window, or a request to press Enter at the end of a simulation, so that the .bat file can continue onto the next simulation.
-c -ca	Copy a model. As of Build 2007-07-AA a copy of a TUFLOW model can be created as described below. Making a copy of a model is useful for transferring a model to another site or for making an archive of the input data. Note that if the model does not yet complete all input data checks, this option may not copy all files. The -ca option (copy all) was introduced for the 2010-10 release. This option copies all files of the same name (ie. same name, but different extensions) for all input files. This option is particularly useful if the .tab and other associated files of a GIS layer need to be archived or delivered. To copy a model specify -c or -ca on the TUFLOW command line. This creates a folder “<.tcf filename>_copy” or “<.tcf filename>_copy_all” in the same location as the .tcf file. Under the folder all input files are copied (including the full folder structure), and any check files and output folders created. For example, specify: TUFLOW.exe -c “c:\tufLOW_models\my model.tcf” will make a copy of the TUFLOW model based on the file “my model.tcf” in a folder “my model.tcf_copy”, or “my model.tcf_copy_all” if using the -ca option. Notes: a. Make sure there is sufficient disk space (no checks for sufficient disk space are made).

Switch	Description
	b. Use the full path to the .tcf file (this is the default if running from UltraEdit or using the right click approach). c. Output folders are created, and some of the output files are also created but these will be empty. d. Any check folder(s) are created and check files written (these can be deleted if wishing to minimise the size of the folder). e. The full path of the input files is reproduced to provide traceability and also handle inputs from other drives and URLs. Drive letters are replaced, for example, “C:” becomes a folder “C Drive”. URLs (denoted by “\” or “/” at the beginning of the path) are replaced by a folder called “URL”. f. To run the copied .tcf file, it will be necessary to change any non-relative pathnames according to the point above. Alternatively you can share and then map, for example, the “D Drive” folder as “D:”. Also, the Check MI Save Date will need to be set to WARNING or OFF in the .tcf file if the –ca option has not been used as the .tab and other files will not have been copied. g. There is a limit of 1,000 characters (including spaces) on pathnames. As very long pathnames can result due to the above approach, if the number of characters exceeds 1,000, problems may occur. h. If –c or –ca is specified, then -t is automatically invoked (ie. the simulation does not commence, only the input data is tested/checked). i. The -b option still applies if several models wish to be copied using a batch (.bat) file.
-e <name> -e {1-9} <name>	Specifies an event name to be used by Define Event in a .tcf Event File to customise inputs for an event. There must be a space between –e and <name>. <name> may itself contain spaces, but if it does the scenario name must be enclosed in quotes. More than one (maximum of nine) events per simulation maybe specified by placing a number after –e. –e and -e1 are treated the same, so don’t specify both of them otherwise indeterminate results may occur. Also see Section 4.12.1 and Model Events. Examples: -e Q100 Run the Q100 event. -e1 Q100 –e2 02h Run the Q100, 2 hour storm event. -e Q100 –e2 02h Same as above. -e “Q100 2h” Quotes required as there is a space in the event name.
-nmb	No Message Boxes. Suppress use of windows message boxes to prompt the user. All prompts will be via the console window.
-nwk	Force TUFLOW to search for a network dongle (ie. skip the search for a standalone dongle).

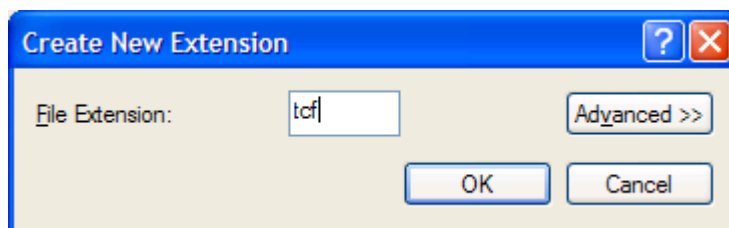
Switch	Description
-s <name> -s {1-9} <name>	Specifies a scenario name to be used by If Scenario blocks to include/exclude commands. There must be a space between -s and <name>. <name> may itself contain spaces, but if it does the scenario name must be enclosed in quotes. More than one (maximum of nine) scenarios per simulation maybe specified by placing a number after -s. -s and -s1 are treated the same, so don't specify both of them otherwise indeterminate results may occur. Also see Section 4.12.2 and Model Scenarios. Examples: -s exg Process all exg scenario commands. -s1 opA -s2 opB Process all opA and opB scenarios. -s opA -s2opB Same as above. -s "Option A" Quotes required as there is a space in the scenario name.
-slp	Note: This feature is only available for SoftLok (blue) dongles. (See Section 5.1.3 for equivalent option for WIBU dongles, and also Simulations Log Folder.) Simulation Log Path. To set the path to a folder on the intranet to log all simulations from the dongle, at a command prompt enter TUFLOW.exe -slp "<url_path_to_global_log_file>". This information is stored on the TUFLOW dongle and should only be carried out by the organisation's administrator of the TUFLOW dongle. Very useful for network dongles being accessed across an intranet so that the administrator can view who's clogging up the network licences! Introduced for Build 2007-07-AA. See Section 7.1.3. At present there is a limit of 64 characters for the path. For example: TUFLOW.exe -slp "\\water\projects\tuflow\log" will prompt to log all simulations started with that dongle to a _TUFLOW Simulations.log file in the folder \\water\projects\tuflow\log.
-t	Test input only. Processes all input data including writing of check files, but does not start the simulation. Useful for checking that simulations in an .bat file all startup OK, prior to carrying out the simulations (especially when the runs will take all night or all weekend and you forgot to export a .mif file!).
-wibu	Search for a WIBU Codemeter dongle only.
-x	eXecute the simulation (the default).

5.4 via Right Mouse Button in Microsoft Explorer

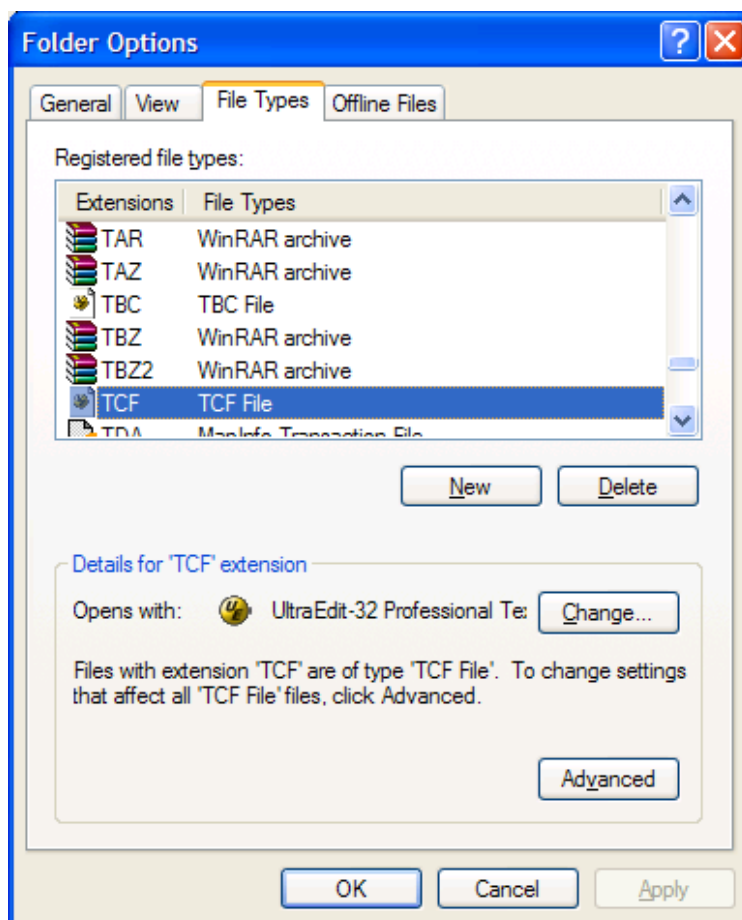
To start a simulation in Microsoft Explorer by using the right mouse button, first follow the following steps to set up a file association:

Windows NT4/2000/XP

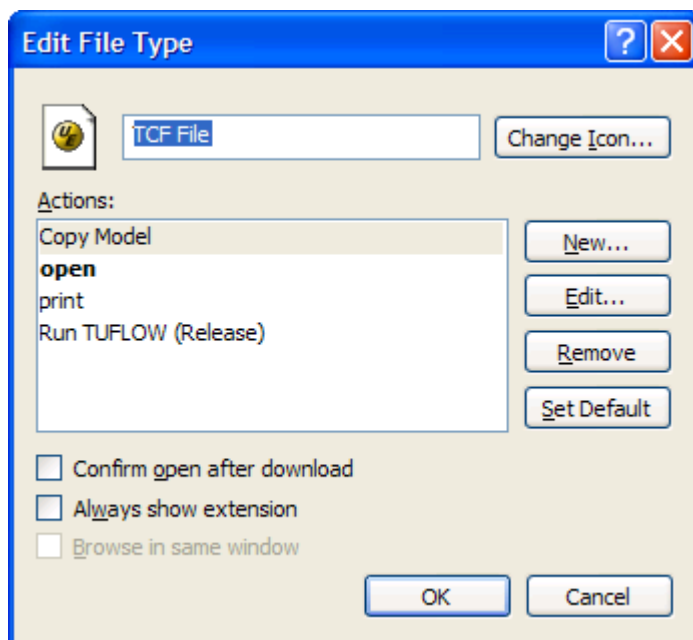
- 1 In Explorer, open the “View”, “Folder Options...” menu and select the “File Types” tab. If .tcf files are not in the “Registered file types:” list box, choose “New Type...”, otherwise select the .tcf file entry under “Registered file types:” as shown in Step 3 below.
- 2 If adding a new type, enter in a description (eg. “TUFLOW Control File”) and “tcf” as the associated extension (see below) and press OK.



- 3 The Folder Options dialog should appear something like the below.

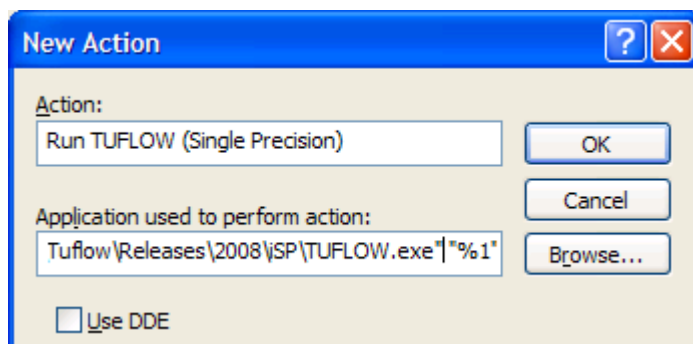


- 4 Click “Advanced” to bring up the dialog below.



- 5 Choose “New...” and enter text to describe the “Action:” (eg. “Run TUFLOW (Single Precision)”) – this text appears on the pop-up menu when you click the right mouse button on a .tcf file in Explorer. Enter or use “Browse...” to specify the path to TUFLOW.exe; note the need for quotes if the path has any spaces. After “TUFLOW.exe”, add a space then “%1” including the quotes, as shown below. Choose “OK”. The “Application used to perform action:” field should be something like:

"C:\Program Files\TufLOW\Releases\2008\iSP\TUFLOW.exe" "%1"



- 6 The action should now appear in the list under “Actions:”. It is not recommended that a “Run TUFLOW” action be set as the default action as it is easy to accidentally start a simulation, which instantly overwrites any existing result files. You may wish to set up other associations at this point, for example, to access older builds or to use the Double Precision build.
- 7 Choose “OK” or “Close”, then “Close” on the “Folder Options” menu.
- 8 Check the file association, by clicking the right mouse button on a .tcf file in Windows Explorer. The “Run TUFLOW (Single Precision)” action should appear in the list.

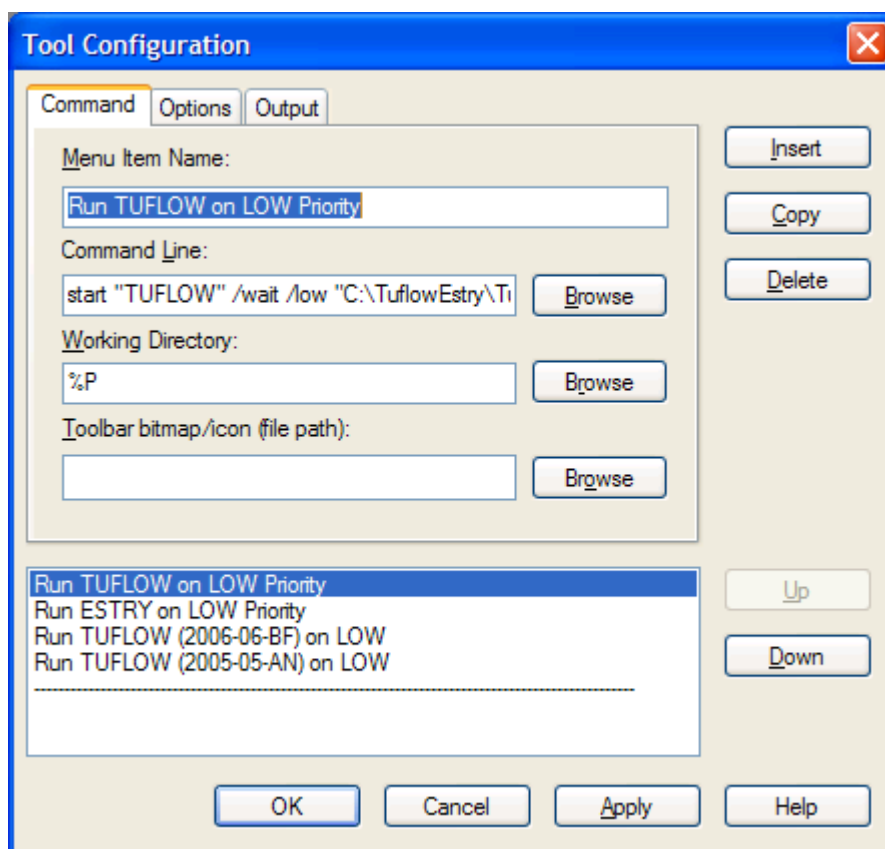
Once the file association is complete, clicking the right mouse button on a .tcf file in Explorer, and selecting the “Run TUFLOW (Single Precision)” action, starts a TUFLOW simulation. A Console Command Window opens and TUFLOW starts.

5.5 From UltraEdit

The majority of users run single TUFLOW simulations from UltraEdit (and use a batch file as described in Section 5.6 for multiple simulations in succession). The benefits of running TUFLOW from UltraEdit is that it provides an environment where the .tcf and other control files can be edited, simulations started and text file output be viewed. There is also no need to close the .tcf file (or other control and output files) to run TUFLOW.

The best way to set up TUFLOW to run from UltraEdit is described below. This will initiate TUFLOW in low priority, therefore the simulation does not dominate the computer’s CPU allowing other processes (eg. MapInfo, SMS, Word, etc) to perform as usual. Low priority will not slow down the TUFLOW simulation whilst no other CPU intensive processes are underway.

To set up UltraEdit to run TUFLOW on low priority go to Advanced, Tool Configuration... to open:



Note: The entry for the Command Line: field would be something like:

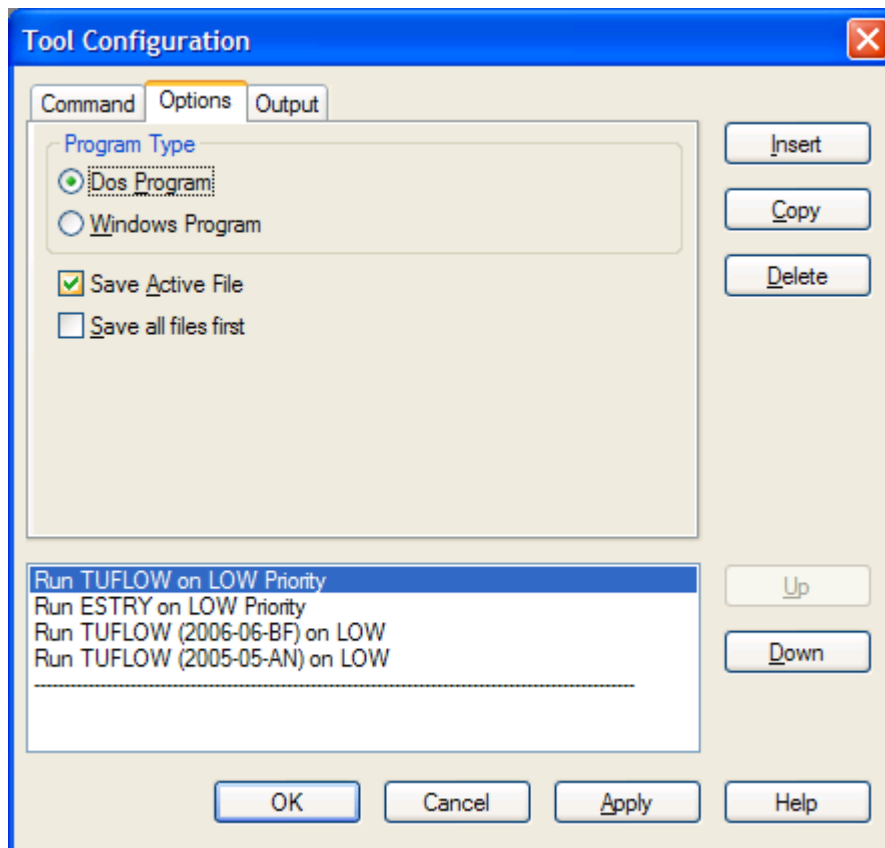
```
start "TUFLOW" /wait /low "C:\Program Files\Tuflow\Releases\2008\iSP\TUFLOW.exe" "%f"
```

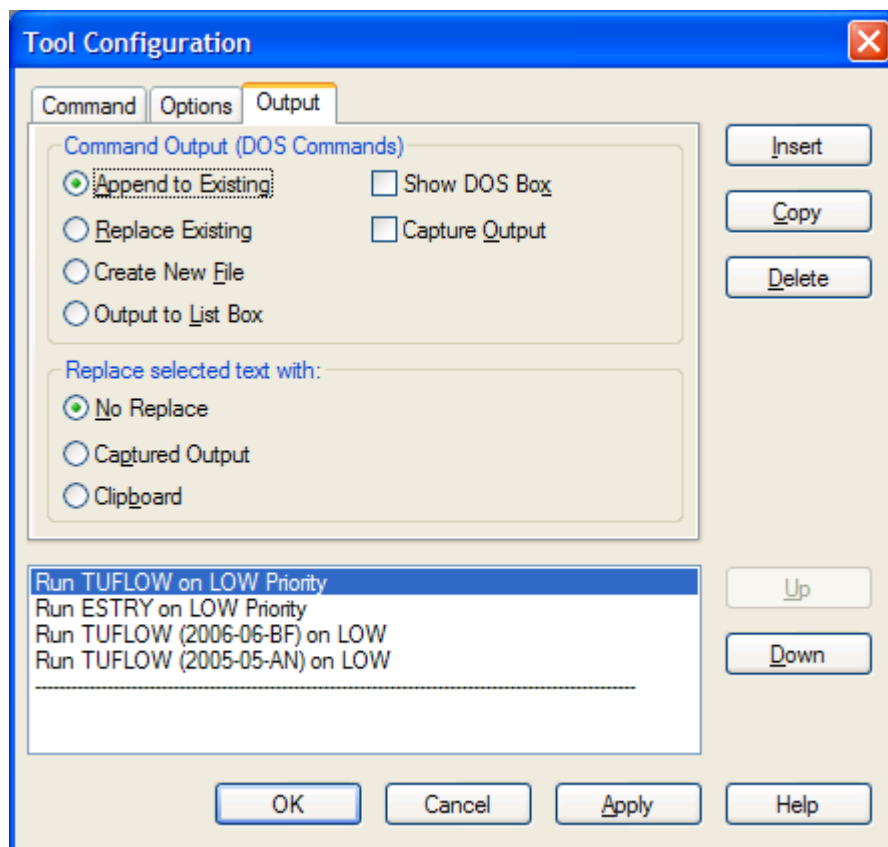
Ensure you specify the double quotes around the pathname to TUFLOW.exe if it contains any spaces, and also the double quotes around the %f in case there are any spaces in the pathname of the .tcf file. The %f indicates the active UltraEdit file.

Next check the various options on the Options and Output tabs are as per the next two dialogues.

The process can be repeated to set up UltraEdit to run ESTRY.exe if required, or to access previous TUFLOW builds as shown in the above dialogue. Use the Insert, Copy and Delete buttons to add and remove configurations.

Note that the dialogues above and below are for UltraEdit Version 13.10. For other versions of UltraEdit the dialogues may appear differently, however, the process will be similar.



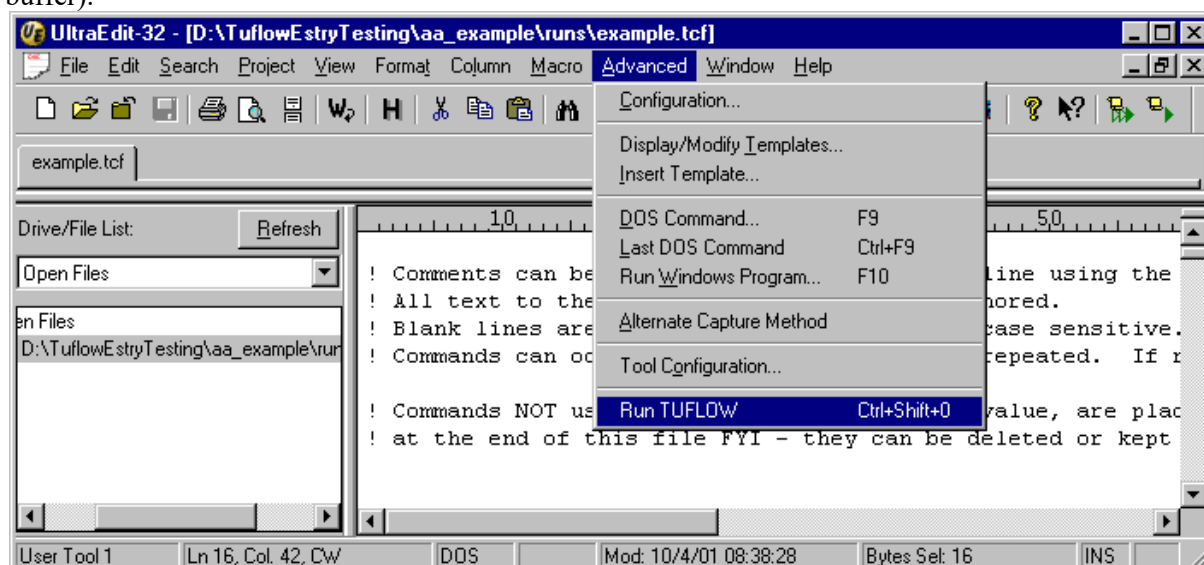


The entered configurations now appear under the Advanced menu as illustrated below. Ensure that the active file in UltraEdit is a .tcf file when starting a TUFLOW simulation.

Note: The “%f” and “%P” are case sensitive. For example, if “%F” is used, this forces filenames to be based on the old DOS eight character file naming convention.

Note: as of Version 9 of UltraEdit, the new “Show DOS Box” checkbox must be checked on.

The “Capture Output” checkbox is useful if the TUFLOW output displayed to the Console window is needed (this particularly useful if running Windows 98 as Windows 98 Console windows have no buffer).



5.6 Batching Simulations using a Batch File

5.6.1 Simple Example and Switches

A batch (.bat) file is the most effective method to run several or many simulations. The simplest format is to specify each simulation one after another as shown below in a .bat file.

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b MR_H99_C25_Q100.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b MR_H99_C25_Q050.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b MR_H99_C25_Q020.tcf
pause
```

The .bat file is run or opened by double clicking on it in Explorer. This opens a Console Window and then executes each line of the .bat file. Note the use of the -b (batch) switch which suppresses the need to press the return key at the end of a simulation. This ensures that one simulation proceeds on to the next without any need for user input. The pause at the end stops the Console window from closing automatically after completion of the last simulation.

The -t (test) switch is very useful for testing the data input without running the simulation. It is good practice to use this switch before carrying out the simulations, as this will tell you whether there are any data input problems. The -t switch runs TUFLOW to just before it starts the hydrodynamic computations.

Using the example above, the recommended approach is to first run the following batch file:

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -t MR_H99_C25_Q100.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -t MR_H99_C25_Q050.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -t MR_H99_C25_Q020.tcf
pause
```

This will indicate any input problems (note some WARNINGS do not require a “press return key”, but they can be located in the .tlf file). Edit the .bat file as follows:

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q100.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q050.tcf
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q020.tcf
pause
```

and carry out the simulations. The -x (execute) switch is optional, but is useful when editing the .bat file to quickly change between -t and -x switches.

Comment lines are specified in a .bat file using “#” in the first column. For example, if you want to re-run only the first simulation in the examples above, edit the file as follows:

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q100.tcf
#C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q050.tcf
#C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q020.tcf
pause
```

For Windows 98 users, using the following command allows the Console Window output to be captured (Windows 98 Console windows do not have a buffer, therefore only the last 20 to 30 lines are

ever viewable). In the example below, the Console output is redirected to a text file named dump.txt, which can be viewed using UltraEdit or other text editor.

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe -b -x MR_H99_C25_Q100.tcf >
dump.txt
```

Consult your Windows on-line help or manual for other features of .bat files and review the following sections for making batch files even more useful.

5.6.2 Windows NT/2000/XP/7 Priority Levels

Windows NT/2000/XP/7 can assign a process a different priority level using the Task Manager. This is very useful for running TUFLOW in the “background” without slowing down other computer work you need to do. Windows NT offers three different priority levels and Windows 2000/XP/7 five.

To initiate TUFLOW simulations from a batch file, precede each of the lines in the above example with “start “TUFLOW” /wait /low” as shown below. This initiates a separate Console Window for each simulation on a low priority. You can also see which simulation is active by viewing the primary Console Window. The /wait option is necessary to force the next simulation not to start until the current one is complete.

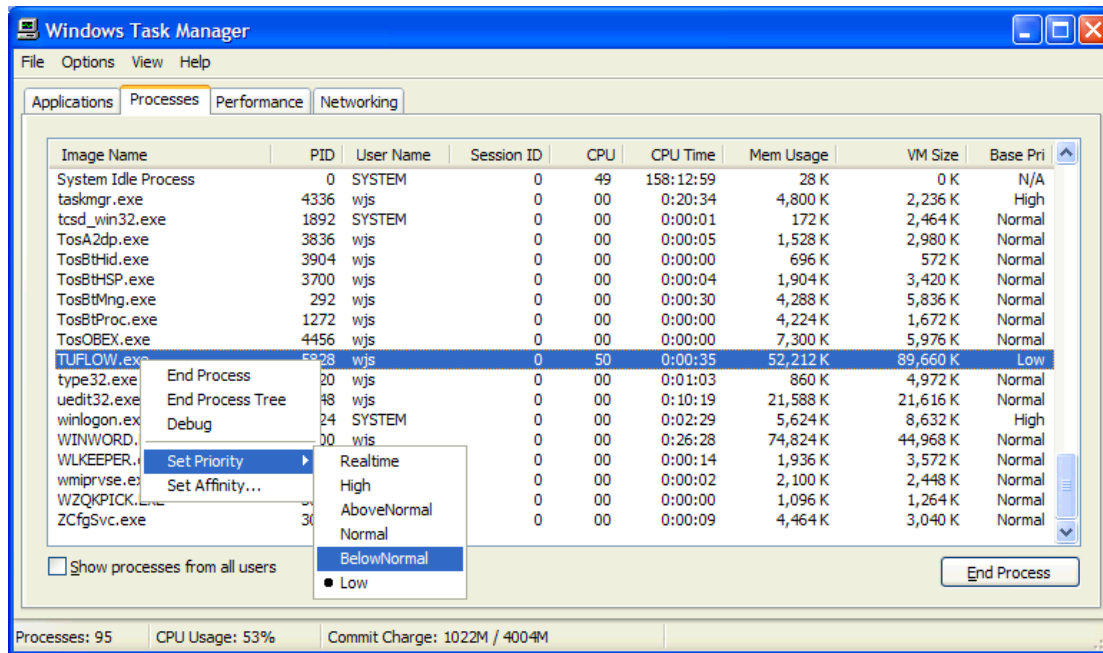
```
start "TUFLOW" /wait /low N:\WBMSoftware\Tuflow.exe -b -x MR_H99_C25_Q100.tcf
start "TUFLOW" /wait /low N:\WBMSoftware\Tuflow.exe -b -x MR_H99_C25_Q050.tcf
start "TUFLOW" /wait /low N:\WBMSoftware\Tuflow.exe -b -x MR_H99_C25_Q020.tcf
pause
```

Other useful switches available are:

- /belownormal and /abovenormal to set these priority levels
- /min to minimise the process once started

The “TUFLOW” in the above is the title that appears in the Console Window.

To change the priority level of simulation manually, open Task Manager (see your System Administrator if you’re not sure how to do this), click on the Processes Tab and find the TUFLOW.exe process you wish to change, right click on TUFLOW.exe, choose Set Priority, then the priority desired as shown in the image below. Note, don’t choose High or Realtime as this will cause the TUFLOW process to take over your CPU and you may not be able to do much until the simulation is finished (of course, this may be a good excuse to not be able to use your computer!).



5.6.3 Advanced .bat Files

Batch files can be easily setup so that they are more generic and easily customised when moving from one project to another. For example, in the below a variable, TUFLOWEXE, is used to define the path to the TUFLOW exe to be used, and a variable RUN is used to incorporate options such as starting the simulation on a low priority and minimise the simulation console window when the process starts (/min option).

```
set TUFLOWEXE=C:\Program Files\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_ISP_w64.exe
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b
%RUN% MR_H99_C25_Q100.tcf
%RUN% MR_H99_C25_Q050.tcf
%RUN% MR_H99_C25_Q020.tcf
```

The advantage of using variables is if the path to the TUFLOW exe changes, or to run a different version of TUFLOW, it is just a simple change in the .bat file. In the above, note the use of quotes around %TUFLOWEXE% in the definition for the RUN variable – quotes are needed around file pathnames whenever they contain a space.

For examples of batch files used for running multiple events and different scenarios from the same .tcf file, see Sections 4.12.1 and 4.12.2.

5.7 From a Console (DOS) Window

A single TUFLOW simulation can be started directly from an open Console Window by entering a line in the same way as entered into a batch file. For example, at the Console prompt enter:

```
C:\TUFLOW\Releases\2010-10\ISP-w64\TUFLOW_isp_w64.exe MR_H99_C25_Q100.tcf
```

You can use the various switches and Windows NT/2000/XP/7 priority settings as discussed in the previous section.

5.8 The Console (DOS) Window Does Not Appear!

Reasons that TUFLOW.exe or ESTRY.exe won't start (ie. no Console Window appears) are:

- When the virtual memory allocation on your computer is congested. For Windows NT/2000/XP/7 check the virtual memory usage using Windows Task Manager. (On Windows 95/98, you don't have the luxury of this facility.) If congested, close some other files or applications and try again. Also check available disk space on your drive from which virtual memory is allocated (normally C: drive) and ensure there is sufficient space. On a PC with 256Mb RAM, it is possible to start a few TUFLOW simulations with little else open. As a rule, particularly if you want other applications open (eg. a GIS with large files open), it is worthwhile investing in larger amounts of RAM.
- Later versions of UltraEdit have issues with starting up a DOS box, and may start TUFLOW as a process with no Console (DOS) Window, but will simulate TUFLOW as a hidden process. To avoid this, set up TUFLOW to run in low priority as described in Section 5.5, ensure the DOS Box checkbox is not ticked (otherwise two DOS windows appear).

For Build 2004-06-AC onwards, make sure you have installed the latest dongle drivers off www.tuflow.com, otherwise TUFLOW will not start up.

If you continue to have problems please contact support@tuflow.com.

5.9 Customising TUFLOW using TUFLOW_USER_DEFINED.dll

TUFLOW_USER_DEFINED.dll allows users to customise TUFLOW to suit their needs. At present, it is confined to the routines that calculate TUFLOW's flood hazard output. This routine can be downloaded from www.tuflow.com, modified by the user (requires some minor knowledge of programming using Fortran or similar) and emailed to support@tuflow.com. TUFLOW_USER_DEFINED.dll will be recompiled and emailed back.

Future releases plan to offer routines that the user can modify, write themselves or have customised for them to perform special tasks during input, simulation and/or output. Examples, are writing output in a special format or inclusion of a special hydraulic structure routine.

5.10 ESTRY.exe

The 1D solution in TUFLOW.exe is available to TUFLOW licensees as its own executable. This may be useful when constructing a 1D only model prior to embedding 2D domain(s) within the 1D domain. The program is called ESTRY.exe and can be downloaded from www.tuflow.com.

ESTRY.exe is generally kept up-to-date with new 1D features in TUFLOW.exe, however, release of new builds of ESTRY.exe may lag those of TUFLOW.exe. The same Build ID is used for both ESTRY.exe and TUFLOW.exe releases.

6 2D/1D Model Development

Section Contents

6	2D/1D MODEL DEVELOPMENT	6-1
6.1	Tutorial Model	6-2
6.2	Setting up a New Model	6-2

6.1 Tutorial Model

A tutorial model is available as of August 2007. At the time of writing, the tutorial is documented for developing models within the MapInfo/Vertical Mapper environment, and for viewing results within a GIS and SMS. Future editions are planned for other GIS environments, and for the SMS TUFLOW Interface and XP-SWMM2D software. The tutorial is also planned to be extended to cover more advanced modelling features such as 2D-2D linking of multiple 2D domains. The data files and documentation are be downloadable from www.tuflow.com as of August 2007.

An update to the Tutorial Model including five or six new modules is planned for release in 2011. The update will include examples of the many new features since 2007. An ArcGIS version is also planned.

As of Build 2007-07-AA, there is no need to have a TUFLOW licence to simulate the tutorial model. Changes to the model's topography, boundaries and other inputs are allowed so that the user can test and try TUFLOW's various features. The command "Tutorial Model == ON" must occur within the .tcf file to simulate the model without needing a licence.

6.2 Setting up a New Model

The steps below describe the process for setting up a new TUFLOW 2D/1D model using MapInfo and Vertical Mapper. These steps should be followed in conjunction with elementary TUFLOW training and as a reminder for later use. They cover the basic elements of setting up a TUFLOW 2D/1D model, with further training and tuition required to use more advanced features.

The steps below assume a sufficient level of GIS skills and use of a text editor. Training in these areas is provided as part of the TUFLOW training. GIS base layers such as a DTM and aerial photos are beneficial to act as a backdrop to the GIS work environment. 3D surface modelling software operating within the GIS is also needed (eg. Vertical Mapper operating within MapInfo).

Set up folders and GIS environment

- 1 Set up TUFLOW model folders as recommended in Table 2.1 in Section 2.2.2 on your computer.
- 2 Start up the GIS and display the GIS base layers on the screen. Develop a clear idea of where your model will be located and its extent (see sections in Chapter 3).

Set the GIS projection & create empty GIS layers

- 3 Create an empty text file and save it with a .tcf extension (eg. my_model.tcf) in the runs folder (see Section 2.2.2 for suggested folder structure). First set the GIS projection using the [MI Projection](#) command. This involves exporting in .mif/.mid format an existing GIS layer (create a "dummy" one if necessary) that is in the projection TUFLOW will use (all layers read by TUFLOW must be in the same GIS projection), and copying the "CoordSys..." line (see [MI Projection](#)).

- 4 Also have TUFLOW create empty GIS layers for later use using the [Write Empty MI Files](#) command. It is recommended to create a folder named “empty” under the “model\mi” folder in which to write the empty GIS layers. The commands in the .tcf file are as follows.

.tcf file:

```
! Set the geographic projection and create empty GIS layers
MI Projection == ..\model\mi\Projection.mif ! Create an empty GIS layer with the
right projection and export it as a .mif file for this command to read
Write Empty MI Files == ..\model\mi\empty
```

- 5 Run TUFLOW using my_model.tcf (see Section 5 for options to run TUFLOW). Empty GIS layers as listed in Table 2.3 should be created in the “..\model\mi\empty” folder and TUFLOW then stops. Remove or comment out (using a “!” or “#”) the [Write Empty MI Files](#) command as this is no longer needed.

Define the location and dimensions of the 2D grid

- 6 In the GIS, import the “2d_loc_empty.mif” layer from the “..\model\mi\empty” folder and save as “2d_loc_my_model.tab” in the “..\model\mi” same folder. Make this layer editable and digitise a straight line (two vertices only) starting at the bottom left corner of the 2D grid and finishing anywhere on the line defining the X-axis (bottom border) of the grid. Save the layer and export as .mif/.mid files.
- 7 Create a .tgc text file (eg. my_model.tgc) in the “..\model” folder. Set the 2D grid origin and orientation using [Read GIS Location](#) and the 2d_loc layer, the cell size using [Cell Size](#), and the grid dimensions using [Grid Size \(N,M\)](#) or [Grid Size \(X,Y\)](#) as follows:

.tgc file:

```
! Set the grid location and dimensions
Read GIS Location == mi\2d_loc_my_model.mif
Cell Size == 10. ! cell size in meters
Grid Size (X,Y) == 200, 100 ! grid dimensions in meters
```

Create the Zpts

- 8 In the .tgc file, enter the line [Set Code](#) == 1 to set all cells to Code 1 (ie. active cells). TUFLOW only exports Zpts at active cells, hence the need to set all cells to active.
- 9 To request TUFLOW to write .mif/.mid files containing the Zpts use [Write MI Zpts](#) then stop the process using [Stop](#) (at this stage we are only interested in writing the Zpt GIS layer and therefore need to stop the simulation at this point). These commands, as shown below, must occur in the .tgc file after setting the code values as discussed above.

add to .tgc file:

```
Set Code == 1
Write MI Zpts == mi\2d_zpt_my_model
Stop
```

- 10 In the .tcf file specify the .tgc file using [Geometry Control File](#).

add to .tcf file:

```
Geometry Control File == ..\model\my_model.tgc
```

- 11 Re-run TUFLOW using the .tcf file. The run should stop after exporting the 2d_zpt layer.

- 12 Import “2d_zpt_my_model.mif” into MapInfo and save using the same name. View the layer noticing the different types of Zpts (these are colour coded).
- 13 The elevations at each Zpt are given a temporary “dummy” value of 99,999, and therefore need to be assigned their correct elevation. Zpt elevations are assigned from the DTM using 3D surface modelling software. For example, by using the “Point Inspection” operation in Vertical Mapper. If the elevations are added as an extra attribute (as is the case with Vertical Mapper), the added attribute must be relocated to be the fourth attribute and the default “Elevation” attribute column must be removed or moved. (In MapInfo use Table, Maintenance, Table Structure... to do this). TUFLOW assumes the elevations are in the fourth attribute column. Additional columns beyond the fourth can exist, but are not used by TUFLOW.
- 14 If some of the Zpts fall outside the DTM, they would have been assigned the “Null” value (eg. Vertical Mapper assigns -9999.) or possibly a zero. These Zpts need to be selected and deleted as they have no meaningful elevation.
- 15 Save and export the 2d_zpt layer (overwrite the .mif/.mid files previously imported).
- 16 Comment out the [Write MI Zpts](#) line in the .tgc file (otherwise the 2d_zpt .mif/.mid files exported in the previous step will be overwritten), and use [Read RowCol Zpts](#) to read the 2d_zpt layer into TUFLOW. **Note the use of MID (not MI) in this command and that the filename must that of the .mid file. This command only reads the .mid file which must contain four attributes, namely cell row (n), cell column (m), Zpt type (Type) and the elevation.**

comment out and add to .tgc file:

```
!Write MI Zpts == mi\2d_zpt_my_model
Read RowCol Zpts == mi\2d_zpt_my_model.mid ! Note use of MID (not MI) and .mid
file
```

Define preliminary active and inactive areas of 2D grid

- 17 To reduce output file sizes and possibly run times, remove permanently dry areas and any other inactive (land) cells from the grid:
 - (a) First, change the [Set Code](#) == 1 command already entered in the .tgc file to [Set Code](#) == 0 to set all cells as inactive (land).
 - (b) In the GIS, import the “2d_code_empty.mif” layer and save as “2d_code_my_model”. Add the layer to your GIS work environment and make it editable. Digitise region(s) to define the active cells. For each of these region(s) set the “Code” attribute to 1.
 - (c) As an alternative to (b) above, the active regions can be placed into a 2d_bc layer that will later contain the 2D boundary locations. This approach is used by advanced modellers to keep the 2D boundaries and code regions together in the same layer. To do this, for each of these region(s) set the “Type” attribute to “CD” and the “f” attribute to “1” – the other attributes are not used (see Table 4.32 and Table 4.33). If you are unsure of exactly where the model boundaries occur, digitise beyond where you think the boundaries will occur and adjust the region later.
- 18 Save then export the 2d_code (or 2d_bc) layer as .mif/.mid files. Use the [Read GIS Code](#) (or [Read GIS Code BC](#)) command in the .tgc file to assign a code 1 (active or water cells) to all cells that fall within the digitised regions. The .tgc commands are shown below for the 2d_code approach.

add to .tgc file:

```
Set Code == 0
Read GIS Code == mi\2d_code_my_model.mif
```

19 Alternatively, if all cells are to be active (water) cells, simply use [Set Code](#) == 1.

To view the 2D mesh

20 A 2d_grd layer can be optionally created using [Write MI Grid](#). This layer will be used to view the model mesh and check the 2D domain. It is not needed as an input to TUFLOW. It is recommended that this layer be placed in the “check” folder as it is mainly used to checking the 2D mesh. When you import this 2d_grd layer into the GIS, it will show you the extent of the mesh’s active cells and the status of the mesh’s attributes at that point in the .tgc file. These attributes can be used to perform thematic mapping and other quality control checks.

add to .tgc file:

```
Write MI Grid == ..\check\2d_grd_check
Stop
```

21 Re-run TUFLOW using the .tcf file.

22 Import “2d_grd_check.mif” in the GIS and save using the same name. View the layer and confirm that the model orientation and extent is correct. Also notice that only cells falling within the water code regions were created.

23 The [Write MI Zpts](#) and [Write MI Grid](#) commands can be used at any point in the .tgc file to produce 2d_zpt and 2d_grd check files of the Zpts and 2D mesh representative of that point in the build process.

24 For the moment, comment out any [Write MI Zpts](#) and [Write MI Grid](#) commands and also comment out the [Stop](#) command.

comment out and add to .tgc file:

```
!Write MI Grid == ..\check\2d_grd_check
!Stop
```

Define the materials (bed resistance categories)

25 Import an empty 2d_mat layer in the GIS and save in the “model\mi” folder as, for example, “2d_mat_my_model”. The layer has no geographic objects and one attribute named Material that must be an integer value.

26 Add the layer to your GIS work environment and make it editable. Digitise regions of different material types, noting that each material is assigned a Manning’s n value. For example: river in-bank; river banks; pasture; roads; lakes; sugar cane; etc. The most common or most difficult to digitise material may be omitted and set as the default material. The different materials to be digitised should be determined in advance of the digitising.

27 Use [Read GIS Mat](#) in the .tgc file to read the 2d_mat layer(s) developed as follows.

add to .tgc file:

```
Read GIS Mat == mi\2d_mat_my_model.mif
```

28 Each material must be assigned a Manning's n value using [Read Materials File](#). This command reads a simple text file, an example of which is shown below. The file is named with a .tmf extension (eg. "my_model.tmf") in the "model" folder.

29 Use [Read Materials File](#) in the .tcf file to read the .tmf file as follows.

```
add to .tcf file:
Read Materials File == ..\model\my_model.tmf
```

Setup up the 1D control file (.ecf file)

30 Create an empty .ecf text file (eg. my_model.ecf) and place it in the runs folder.

31 In the .tcf file, use [ESTRY Control File](#) to specify the .ecf file. It is strongly recommended that the Auto option be specified to force the user to name the .ecf file to be the same as the .tcf file.

```
add to .tcf file:
ESTRY Control File Auto ! looks for a .ecf file with the same name as the .tcf file
```

32 In the .ecf file, progress through and enter the various commands. Those that are mandatory or most commonly used for a 2D/1D model are: [Start Output](#), [Output Interval](#), [Timestep](#), [Output Folder](#), [Read GIS Network](#) and [Read GIS BC](#). Note: it is NOT necessary to use .ecf commands such as [MI Projection](#), [Start Time](#) and [End Time](#) and [Timestep](#) as these are only relevant for a 1D only model.

```
.ecf file:
Start Output == 1 ! start output at 1 hour
Output Interval (s) == 300 ! output 1D results every 5 minutes
Output Folder == ..\results ! output results to the results folder
```

Setup up the 1D network (1d_nwk) GIS layer

33 Import an empty 1d_nwk layer and save in the "model\mi" folder. Add the layer to your GIS work environment and make it editable. Digitise 1D channels ensuring they are snapped at each ends. As they are digitised, enter the channel ID and the other attributes according to the type of channel (see Table 4.14). Note: it is no longer a requirement to digitise nodes (see Section 4.5.1). Setup any links to cross-section data, etc. There are a number of ways to manage cross-sections (eg. using .csv files, MIKE 11 or ISIS cross-section database or the traditional fixed field CS tables).

34 Use [Read GIS Network](#) in the .ecf file to read the 1d_nwk layer(s) developed as follows.

```
add to .ecf file:
Read GIS Network == ..\model\mi\1d_nwk_my_model.mif
```

Setup up the 1D and 2D boundary condition (1d_bc & 2d_bc) GIS layers

35 Import an empty 1d_bc and/or 2d_bc layers and save in the "model\mi" folder. If there are no boundaries in one of the 1D or 2D domains, there is no need to create the corresponding layer.

36 For the 1D domain, digitise points snapped to the ends of channels where inflows and water level boundaries are to be assigned (1D boundaries can only be assigned to nodes). Enter the attributes for each point (see Table 4.30 and Table 4.31) to define the type of boundary and the name of the boundary data to be extracted from the boundary condition database.

37 Use [Read GIS BC](#) in the .ecf file to read the 1d_bc layer(s) developed as follows.

add to .ecf file:

```
Read GIS BC == ..\model\mi\1d_bc_my_model.mif
```

38 For the 2D domain, digitise objects where any boundaries are to be assigned. This includes connections to the 1d_nwk layer (ie. HX, SX and CN 2d_bc objects). Enter the attributes for each object (see Table 4.32 and Table 4.33) to define the type of boundary and the name of the boundary data to be extracted from the boundary condition database, or the connections to the 1D network.

- (a) Note: For 2D boundaries running along the perimeter of the code regions in either the 2d_code or 2d_bc layer, the 2D boundaries should be snapped to the region perimeter to ensure that the boundary is located along the edge of the active cells.
- (b) If using the 2d_bc layer to define the code regions, the same layer can be used for the 2D boundaries.

39 Create an empty .tbc text file (eg. my_model.tbc) and place it in the “model” folder.

40 Use [Read GIS BC](#) in the .tbc file to read the 2d_bc layer(s) developed as follows.

.tbc file:

```
Read GIS BC == mi\2d_bc_my_model.mif
```

41 In the .tcf file, use [BC Control File](#) to set the .tbc file as follows.

add to .tcf file:

```
BC Control File == ..\model\my_model.tbc
```

Setup up the boundary condition database

42 Setup the boundary condition database in the “bc dbase” folder (see Section 4.10).

43 In the .tcf file use [BC Database](#) to set the location of the database as follows. This command may also be used in the .ecf and .tbc files, however, by placing it in the .tcf file it automatically applies to both the 1D and 2D domains – this is preferred so as to keep all boundary conditions in the one database, unless it is desired to separate 1D and 2D boundaries.

add to .tcf file:

```
BC Database == ..\bc dbase\my bc dbase.csv
```

44 If using [BC Event Name](#) and [BC Event Text](#) (strongly recommended where a range of different events occurs – see Section 4.10.3) enter these commands into the .tcf file. For example:

add to .tcf file:

```
BC Event Text == _event_  
BC Event Name == Q100
```

Setup up simulation times and other controls (.tcf file)

45 Determine when the simulation is to start, end, etc. The mandatory and most common commands used are: [Start Time](#), [End Time](#), [Timestep](#), [Map Output Data Types](#), [Start Map Output](#), [Map Output Interval](#), [Output Folder](#), [Store Maximums and Minimums](#) and [Write Check Files](#).

add to .tcf file:

```
Start Time == -10 ! start simulation at -10 hours
End Time == 30 ! end simulation at 30 hours
Timestep == 10 ! use a timestep of 10 seconds
Map Output Data Types == hVqd ! output levels, velocities, unit flows & depths
Start Map Output == 0 ! start map output at zero hours
Map Output Interval == 1800 ! output SMS data every 30 minutes
Output Folder == ..\results ! output results to the results folder
Store Maximums and Minimums == ON MAXIMUMS ONLY ! save peak values only
Write Check Files == ..\check\2d ! write check files and prefix them with "2d"
```

Set initial water levels

46 Set the initial water levels in the 1D and 2D domains using the various commands (see Section 4.9).

Run the model!

47 Run TUFLOW using my_model.tcf (see Section 5). No doubt it will run perfectly!

7 Data Output

Section Contents

7	DATA OUTPUT	7-1
7.1	General	7-2
7.1.1	Console (DOS) Window Display	7-2
7.1.1.1	Console Window Shortcut Keys (Ctrl-C and Ctrl-S)	7-4
7.1.1.2	Customisation of Console Window	7-4
7.1.2	Message Boxes	7-7
7.1.3	_ TUFLOW Simulations.log Files	7-8
7.1.3.1	Local .log File	7-8
7.1.3.2	Global .log File	7-10
7.2	Check Files	7-11
7.2.1	Simulation Log File (.tlf or .elf file)	7-11
7.2.2	ERROR, WARNING and CHECK Messages (in .tlf and _messages.mif Files)	7-11
7.2.3	.wor File	7-12
7.2.4	.eof File	7-12
7.2.5	Using the Write Check Files Command	7-13
7.3	2D Domains	7-18
7.3.1	SMS (Map) Output (.dat or .xmdf Files)	7-18
7.3.2	Time-Series Output	7-27
7.3.3	Conversion to GIS Formats	7-27
7.4	1D Domains	7-28
7.4.1	Output File (.eof file)	7-28
7.4.2	SMS Output	7-29
7.4.3	GIS and Text 1D Domain Check Files	7-30
7.4.4	Time-Series Output	7-30
7.4.4.1	_TS.mif Layers	7-30
7.4.4.2	_ccA.mif Layer	7-32
7.4.5	Maximum/Minimum Output	7-33
7.5	Mass Balance Output	7-34

7.1 General

7.1.1 Console (DOS) Window Display

TUFLOW displays a lot of information to the Console (DOS) Window during the data input stages. If there are data input problems, trace back through the Window buffer (no buffer is available on Windows 98) to establish where in the input data process the problem occurs (noting that it is often more efficient to use the log and check files documented in Section 7.2 to identify problems). See below for setting the size of the Console Window and buffer. Alternatively, and far more efficiently, use the `_messages` and `_check` layers (see Section 7.2.2) or search the `.tlf` file.

For Windows 95/98 users (anyone still using these platforms?), it is not possible to specify a buffer for Console Window output. If you need to view the display output that has vanished of the top of the window, view the `.tlf` log file (see Section 7.2.1) or search this document for “Windows 98” for alternative methods.

Once the simulation has started, the simulation status at each timestep is displayed (use [Screen/Log Display Interval](#) to change the frequency of display). Where different timesteps are used for different domains, the display interval is based on the largest timestep. The Console Window appears as something similar to that shown below. The colours, size and other attributes of the window can be changed as discussed in Section 7.1.1.2.

TUFLOW Build: 2007-07-AA

Simulating: BRI_Q100_exg_001.tcf

12090	6:43:00	-d	0	0	Wet	8454	CS	100	47	GL	10.882	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	67	dU	257
12100	6:43:20	-d	0	0	Wet	8476	CS	99	46	GL	10.883	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	67	dU	256
12110	6:43:40	-d	0	0	Wet	8503	CS	101	47	GL	10.884	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	67	dU	254
12120	6:44:00	-d	0	0	Wet	8511	CS	100	51	GL	10.885	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	67	dU	253
12130	6:44:20	-d	0	0	Wet	8527	CS	98	52	GL	10.886	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	67	dU	252
12140	6:44:40	-d	0	0	Wet	8540	CS	97	52	GL	10.887	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	251
12150	6:45:00	-d	0	0	Wet	8551	CS	98	51	GL	10.887	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	251
Writing Output at: 6:45:00 Clock Time: 0:35:21 CPU Time: 0:19:03																					
12160	6:45:20	-d	0	0	Wet	8564	CS	99	50	GL	10.888	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	250
12170	6:45:40	-d	0	0	Wet	8580	CS	98	50	GL	10.889	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	249
12180	6:46:00	-d	0	0	Wet	8604	CS	102	52	GL	10.890	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	249
12190	6:46:20	-d	0	0	Wet	8628	CS	102	53	GL	10.890	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	248
12200	6:46:40	-d	0	0	Wet	8662	CS	106	54	GL	10.891	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12210	6:47:00	-d	0	0	Wet	8682	CS	110	57	GL	10.892	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	247
12220	6:47:20	-d	0	0	Wet	8698	CS	110	55	GL	10.893	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12230	6:47:40	-d	0	0	Wet	8715	CS	109	56	GL	10.893	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12240	6:48:00	-d	0	0	Wet	8736	CS	110	55	GL	10.894	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12250	6:48:20	-d	0	0	Wet	8756	CS	109	54	GL	10.895	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12260	6:48:40	-d	0	0	Wet	8772	CS	114	55	GL	10.895	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	244
12270	6:49:00	-d	0	0	Wet	8799	CS	116	57	GL	10.896	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	245
12280	6:49:20	-d	0	0	Wet	8813	CS	114	59	GL	10.897	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	245
12290	6:49:40	-d	0	0	Wet	8825	CS	112	58	GL	10.898	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12300	6:50:00	-d	0	0	Wet	8839	CS	111	59	GL	10.898	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	247
12310	6:50:20	-d	0	0	Wet	8846	CS	111	58	GL	10.899	CE	0.0%	0.0%	-0.1%	Qi	80	Qo	68	dU	246
12320	6:50:40	-d	0	0	Wet	8869	CS	110	59	GL	10.900	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	248
12330	6:51:00	-d	0	0	Wet	8885	CS	111	59	GL	10.900	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	248
12340	6:51:20	-d	0	0	Wet	8899	CS	107	58	GL	10.901	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	248
12350	6:51:40	-d	0	0	Wet	8914	CS	111	57	GL	10.902	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	248
12360	6:52:00	-d	0	0	Wet	8933	CS	106	56	GL	10.903	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	249
12370	6:52:20	-d	0	0	Wet	8947	CS	111	55	GL	10.903	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	249
12380	6:52:40	-d	0	0	Wet	8957	CS	107	55	GL	10.904	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	249
12390	6:53:00	-d	0	0	Wet	8972	CS	108	51	GL	10.905	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	249
12400	6:53:20	-d	0	0	Wet	8996	CS	109	53	GL	10.905	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	68	dU	248
12410	6:53:40	-d	0	0	Wet	9009	CS	107	52	GL	10.906	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	249
12420	6:54:00	-d	0	0	Wet	9024	CS	110	53	GL	10.907	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	247
12430	6:54:20	-d	0	0	Wet	9052	CS	110	56	GL	10.908	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	248
12440	6:54:40	-d	0	0	Wet	9064	CS	114	53	GL	10.908	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	247
12450	6:55:00	-d	0	0	Wet	9087	CS	117	53	GL	10.909	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	247
12460	6:55:20	-d	0	0	Wet	9097	CS	115	52	GL	10.910	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	246
12470	6:55:40	-d	0	0	Wet	9124	CS	122	51	GL	10.911	CE	0.0%	0.0%	-0.1%	Qi	81	Qo	69	dU	246

Along each line the following information is shown in order of occurrence for Build 2007-07-AA. For previous builds refer to the previous manuals.

- Number of timesteps completed based on the largest timestep of all 1D and 2D domains.
- Simulation time in hhhh:mm:ss.
- “-d” followed by two numbers:

- The maximum number of 1D nodes per timestep that experienced negative depths below -0.1m since the previous display line.
- The maximum number of 2D cell sides per timestep that experienced negative depths below -0.1m since the previous display line.

The locations of these negative depths are output as warnings in the _messages layer (see Section 7.2.1). Negative depths indicate the model is having difficulty in convergence at that location, which may lead to an instability. See also Section 9.4.

- “Wet” followed by number of wet or active 2D cells.
- If automatic weir switching is active (see [Free Overfall](#)) the next information is “CS” (Cell Sides) followed by two numbers as follows:
 - The number of cell sides where upstream controlled friction flow occurred (see [Supercritical](#)).
 - The number of cell sides where upstream controlled broad-crested weir flow occurred (see [Free Overfall](#)).
- If the free-overfall algorithm is set to ON WITHOUT WEIRS (see [Free Overfall](#)), the next information is “FO” followed by the number of cell sides where the free-overfall algorithm is being applied. Note: this option is now rarely used in lieu of the automatic weir and supercritical flow options.
- If [Display Water Level](#) was specified, the next piece of information is a “GL” (Gauge Level) followed by the water level at the location indicated. This is useful to monitor the rise and fall of the water level at a key location.
- If [Mass Balance Output](#) is set to ON, “CE” (Cumulative Error) followed by three percentages is displayed to show the cumulative mass error as follows:
 - The whole of model % cumulative mass error for all 1D and 2D domains.
 - The % cumulative mass error for all 1D domains.
 - The % cumulative mass error for all 2D domains.
- If [Mass Balance Output](#) is set to ON the following are displayed after the “CE” percentages:
 - “Qi” followed by the total flow into the model (all domains) in m³/s. If the inflow exceeds 999,999m³/s or falls below -99,999m³/s, the flow is expressed in units of 1,000m³/s and a single quote symbol is displayed after the number. A double quote symbol indicates the flow is expressed in units of 1,000,000m³/s.
 - “Qo” followed by the total flow out of the model (all domains) in m³/s. If the outflow exceeds 999,999m³/s or falls below -99,999m³/s, the flow is expressed in units of 1,000m³/s and a single quote symbol is displayed after the number. A double quote symbol indicates the flow is expressed in units of 1,000,000m³/s.
 - “dV” followed by the change in volume in m³ of the model (all domains) since the last display time. If the change in volume exceeds 999,999m³ or falls below -99,999m³, the amount is expressed in units of 1,000m³ (mL) and a single quote symbol is displayed after the number. A double quote symbol indicates the change in volume is expressed in units of 1,000,000m³.

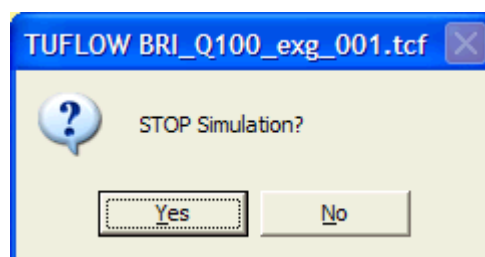
The negative depth numbers, cumulative error percentages, inflow, outflow and change in volume figures are very useful to gauge the health of the model. Frequent negative depths, poor cumulative error (>1%, noting that some models will show a high mass error at the start, which can be acceptable provided it diminishes quickly) and “bouncy” inflow, outflow and change in volume values are all indicators of an unhealthy model. For further discussion see Section 8.1.

Whenever the map output is written (see [Start Map Output](#) and [Map Output Interval](#)), a line “Writing Output at:” is displayed followed by the simulation time, the clock time and the CPU time. If the CPU time is significantly lower than the clock time, then either the simulation was paused for awhile (see next section), the CPU is overloaded or the CPU is not being fully utilised (having Hyper Threading switched on can cause this to occur – see you system administrator!).

7.1.1.1 Console Window Shortcut Keys (Ctrl-C and Ctrl-S)

Useful shortcut keys available in Console Window are:

- Ctrl+S to pause the simulation and freeze the Console Window display. Repeat Ctrl+S to restart. This is useful if you are running a simulation on a notebook computer that you are taking elsewhere. Pause the TUFLOW simulation by pressing Ctrl+S, suspend or hibernate your notebook, then when the computer is restarted, press Ctrl+S again on the TUFLOW window to continue with the simulation.
- Ctrl+C when pressed the first time on a TUFLOW window displays the dialogue below asking whether to stop the simulation. Clicking on Yes will finish the simulation, write all output files off neatly and release any network dongle licence. The simulation is logged as being INTERRUPTED in the .tlf and .log files. Clicking on No will continue the simulation. As of Build 2008-08-AA, the problem of pressing Ctrl+C a second time and unfortunately killing the process without neatly writing out the output files as been resolved, although you may have to press Ctrl C twice to get it to register!



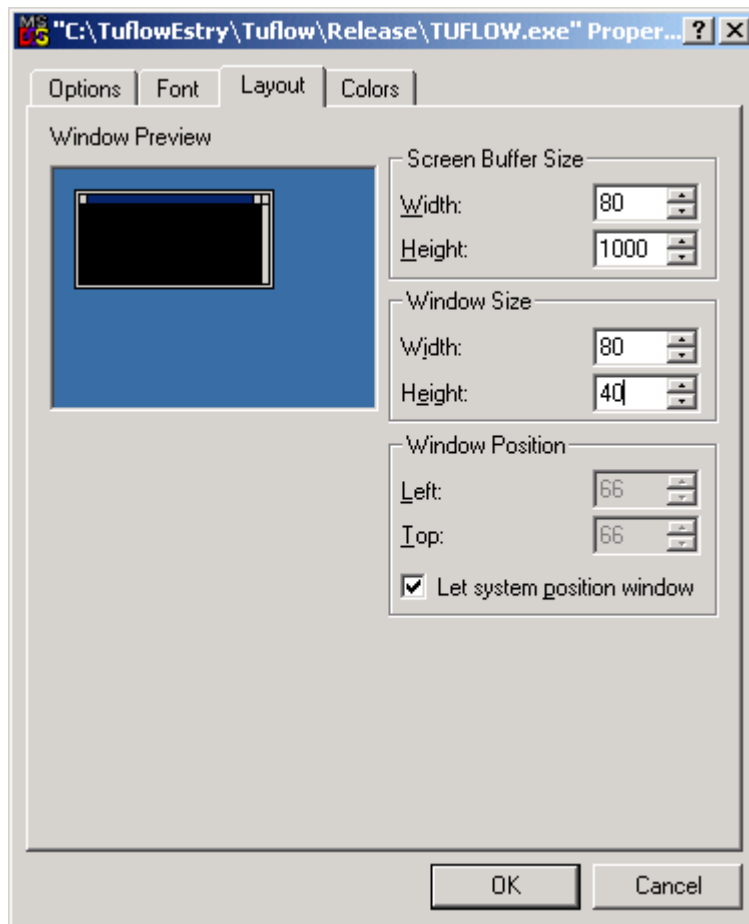
7.1.1.2 Customisation of Console Window

As of Build 2007-07-AA, the windows and buffer sizes are by default set by TUFLOW. During the model input stages the window is set to 122 characters wide and 30 lines high. During the hydraulic calculations the width varies depending on the length of the output to the window and the height is set to 40 lines.

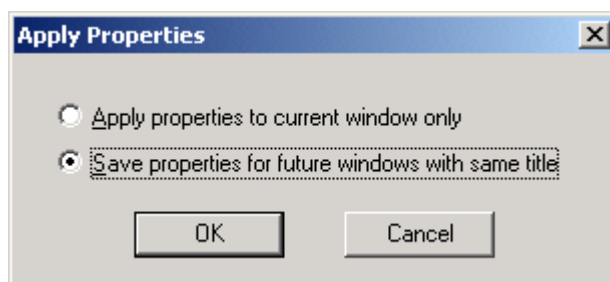
To manually set the Console Window buffer in Windows NT/2000/XP follow the following steps:

- 1 After starting a TUFLOW simulation and before exiting the Console Window, click the right mouse button whilst the pointer is over the title bar of the Console Window and select Properties

to display the dialogue window below.

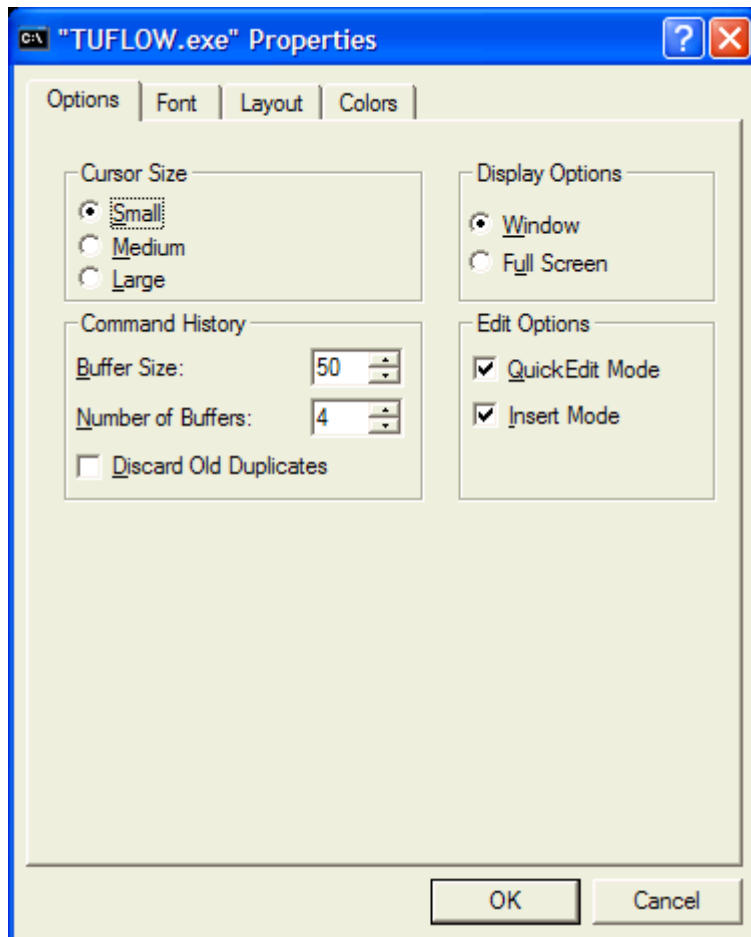


- 2 Select the Layout tab and set the Screen Buffer Size Height to as many lines as you need. Typically 1000 is sufficient. The actual window size width and height may also be changed.
- 3 Click OK to display the dialogue below and select "Save properties for future windows with same title". Click OK and future TUFLOW simulations will use a Console Window of with the buffer and window dimensions as specified.



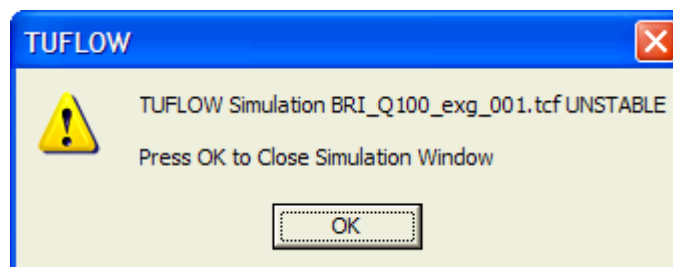
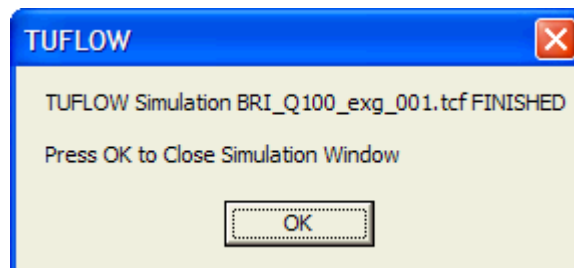
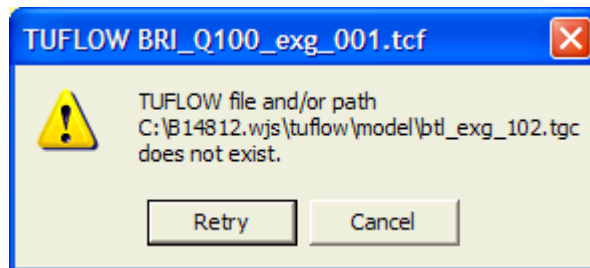
To change the font and colours of the TUFLOW window, use the Font and Color Tabs in the dialogue above.

A useful tip to cut and paste text from the Console Window, and to be able to use your scroll wheel on the mouse to move up and down within the window, first check that on the Options Tab in the dialogue (shown below) has Quick Edit Mode ticked. Text may then be selected with the left mouse button and copied to the clipboard by clicking the right mouse button. The scroll wheel should also pan the contents of the buffer up and down within the window.



7.1.2 Message Boxes

As of Build 2007-07-AA (and Build 2006-06-BA), windows message boxes are used to alert the user to an input problem and when a simulation has stopped/finished. A couple of examples of message boxes are shown below.



7.1.3 _ TUFLOW Simulations.log Files

TUFLOW activity is written to two log files:

- A local file named “_ TUFLOW Simulations.log” located in the same folder as the .tcf file.
- A global log file that can optionally be located in a fixed location on your organisation’s intranet.

7.1.3.1 Local .log File

The “_ TUFLOW Simulations.log” file is a text file containing a record of every simulation initiated from that folder, and is located in the same folder as the .tcf file(s). Information contained in the file includes the following (depending on the TUFLOW release):

- Date and time of the log entry.
- Dongle ID (if applicable).
- Type of TUFLOW licence. The notation on the type of licence varies depending on the licence origin as follows:
 - BMT WBM WIBU (Metal) Dongle:
 - LOC3/4 for a Local (Standalone) Licence (the numbers in this example indicate it was the third licence out of four available).
 - NWK03/10 for a Network Licence (the numbers in this example indicate it was the third licence out of ten available).
 - BMT WBM SoftLok (Blue) Dongle:
 - “SL” for Standalone Licence
 - “SNL03/10” for Started Network Licence (the numbers in this example indicate it was the third licence out of ten available).
 - “RNL03/10” for Restarted Network Licence (this may occur if the network was down and the simulation had to restart the licence).
 - “FNL03/10” for Finished Network Licence (the network licence was released).
 - Aquaveo SMS Licence:
 - “SMS” for all types of licences.
 - Halcrow Software Licence:
 - “HalcS” for a local Standalone licence
 - “HalcN” for a network licence
 - XP-Software Licence:
 - “XP” for all types of licences.

- A Tutorial Model (no dongle required):
 - “TUT” for all tutorial models.
- Computer Name on which the simulation is being run
- TUFLOW Build ID
- CPU hours
- Simulation status as one of the following:
 - “Started”
 - “Finished”
 - “Interrupted”
(the simulation was stopped by pressing Ctrl+C)
 - UNSTABLE
(the simulation became unstable based on the water level exceedance checks).
- Simulation name
- .tcf filename
- .tlf filename
- For “Finished” or “Interrupted”, indicators of the mass error and stability performance are also provided as follows (see Table 7.9 for a discussion on these indicators):
 - “fCME” is the Final Cumulative Mass Error at the end of the simulation.
 - “pCME” is the Peak CME throughout the whole simulation.
 - “pCME5” is the Peak CME for the period of the simulation that the flow in and out of the model exceeds 5% of the peak flow in and out (this value generally excludes any initial high ME values that may occur at the start of some simulations).
 - “pddV” is the Peak change in dV (change in volume) over one timestep divided by the peak dV value expressed as a percentage.
 - “pddV5” is the same as pddV but over the period of the simulation where the flow in and out of the model exceeds 5% of the peak flow in and out.

It is strongly recommended this file is not deleted or edited as it could provide a valuable trace back to old simulations.

This feature was released with Build 2003-03-AE and further enhanced for Builds 2007-07-AA and 2010-10-AA.

Excerpts from a local “_ TUFLOW Simulations.log” file are shown below:

```

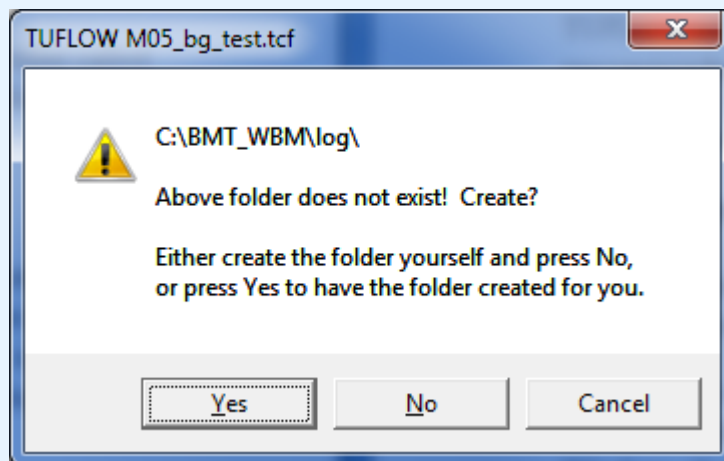
2007-Jul-25 10:55 SL      DGR7VM1S      Build: 2007-07-AA      Started: BRI_Q100_exg_001.tcf
2007-Jul-25 12:14 SL      DGR7VM1S      Build: 2007-07-AA      Finished: BRI_Q100_exg_001.tcf
2007-Jul-25 12:21 SL      DGR7VM1S      Build: 2007-07-AA      Started: BRI_Q100_exg_001.tcf
2007-Jul-25 12:36 SL      DGR7VM1S      Build: 2007-07-AA      Interrupted: BRI_Q100_exg_001.tcf
2007-Jul-25 13:31 SL      DGR7VM1S      Build: 2007-07-AA      Started: BRI_Q100_exg_001.tcf
2007-Jul-25 13:33 SL      DGR7VM1S      Build: 2007-07-AA      Started: BRI_Q100_exg_001.tcf

```


2007-Jul-25 13:33	SL	DGR7VM1S	Build: 2007-07-AA	Finished: BRI_Q100_exg_001.tcf
2007-Jul-25 13:37	SL	DGR7VM1S	Build: 2007-07-AA	Started: BRI_Q100_exg_001.tcf
2007-Jul-25 13:38	SL	DGR7VM1S	Build: 2007-07-AA	UNSTABLE: BRI_Q100_exg_001.tcf

7.1.3.2 Global .log File

For the 2010-10 release and the DB or later builds for the 2006-06, 2007-07, 2008-08 and 2009-07 releases, this file is named “_ All TUFLOW Simulations.log” and by default is located in a folder called “C:\BMT_WBM\log”. If this folder does not exist TUFLOW will prompt you with the message below to create it (ensure you have write permission to create this folder).



The path to the global .log file can be changed using the following options:

- For all types of licenses use the “Simulations Log Folder ==” command in the Dongle Control File (see Section 5.1.3).
- For the SoftLok (Blue) dongles, you can also use the -slp TUFLOW.exe option (see Table 5.2).
- Alternatively use [Simulations Log Folder](#) in the .tcf file (this option is given priority over the options above).
- For any of the options above, if the keywords “DO NOT USE” occur within the folder path or URL name, writing to the global .log file is disabled.

Prior to the 2010-10 release and the DB builds for prior releases, this file was named “_ TUFLOW Dongle <serial_no> Simulations.log”, and is created for each dongle owned by your organisation. The file is by default is written to C:\, but the dongle can be coded using the -slp TUFLOW.exe option (see Table 5.2) to define a URL to a folder on your organisation’s intranet. This is particularly useful for organisations operating network licences across their intranet.

The entries to the global .log file are as described for the local .log file above.

7.2 Check Files

TUFLOW and ESTRY produce check files for quality control of a model's input data. **It is strongly recommended that models are quality controlled through reviews of the check files.**

Effective use of the check files can save days during a model's development and application.

7.2.1 Simulation Log File (.tlf or .elf file)

TUFLOW and ESTRY produce a log file (.tlf or .elf file) containing a record of the simulation. The file is very useful for establishing data input problems and identifying instabilities.

Take time to familiarise yourself with the content of the log file. Much of it is a repeat of the information displayed to the Console Window, so if you can't access information from the Console Window, check the log file using a text editor.

At key stages during the model development and application search the file for any "WARNING", "CHECK" or "NOTE" messages. "WARNING" messages in particular should be checked regularly. An "ERROR" keyword indicates an unrecoverable error and causes the simulation to stop. As many errors as possible are trapped before stopping.

An "XY:" at the beginning of a line indicates the error, warning, check or other message has also been redirected to a .mif file (see Section 7.2.2). Opening the .mif file in the GIS often provides a far more rapid location of the message within the model domain(s) than via other ways.

As of Build 2002-10-AC, for 2D/1D models, the 1D domain log file output is now directed to the .tlf file (was previously sent to the .elf file). The .elf file is now only created for 1D only (ESTRY) models.

7.2.2 ERROR, WARNING and CHECK Messages (in .tlf and _messages.mif Files)

Error, warning, check and other useful messages that are output to the Console window and log file are also output to a .mif file provided the message can be geographically located within the model domains.

As of Build 2004-02-AA, the messages and other information are written to a file called <.tcf filename>_messages.mif located in the same folder as the .tcf file or in the [Log Folder](#).

Prior to Build 2004-02-AA, errors and warnings (denoted by "ERROR" or "WARNING") were located in _errors & warnings for <.tcf filename>.mif. Checks (denoted by "CHECK") and other messages were placed in _checks for <.tcf filename>.mif. The .mif files were located in [Output Folder](#) for 2D/1D models and [Output Folder](#) for 1D only models.

This feature allows rapid location within the GIS environment of data input and simulation errors and potential problems. Use of this feature can save days when setting up and checking new models.

As of Build 2005-05-AN, some messages are being displayed using an object rather than text, with the message provided as attribute data to the object. Examples are:

- Messages relating to HX cells that overlap are now shown as a blue hatched cell, with the message as attribute data to the cell.
- HX and SX cells automatically adjusted using the Z flag are shown as magenta and yellow cells respectively. Information on the adjustment is available in the attribute data for the cell.

Build 2007-07-AA introduces a message numbering system that is being incorporated. As of this build, the bulk of the 2D messages are prefixed by a four digit number. These numbers are being used to cross-reference to a message database that contains more detailed information on the CHECK, WARNING or ERROR to help check/resolve the issue. Once the Microsoft Access database of the messages and additional information is uploaded to www.tuflow.com a notification will be issued on the TUFLOW online forum.

7.2.3 .wor File

This file is a MapInfo workspace and is created for every simulation. It is named <tcf_filename>.wor and is written to the same folder as the .tcf file. The workspace contains all GIS layers used as input to the simulation, and is an excellent way of ascertaining which GIS layers were used to set up a model, particularly large models with many GIS inputs.

The .wor file when opened in MapInfo simply opens the .tab layers. No Map or Browser windows are automatically opened. The file may also be viewed in a text editor.

This feature was first released with Build 2004-02-AA.

7.2.4 .eof File

The .eof file contains a complete output of the final set of 1D domain input data after reading all the GIS layers and other data sources. It also contains all time-series output in text format (see Section 7.4.1 for details). Use this file to check whether the 1D input data has been correctly interpreted.

7.2.5 Using the [Write Check Files](#) Command

Table 7.1 lists the various check files available using [Write Check Files](#) (.tcf file) and [Write Check Files](#) (.ecf file). At key stages in a model's development, produce these check files, and check their contents to ensure that the input data are as expected.

Many of the GIS .mif/.mid files can also be used for creating new, pre-formatted, GIS layers.

Table 7.1 Types of Check Files

Filename or Prefix/Suffix	Description
2D Domains	
_2d_bc_tables_check.csv	Tabular data as read from the boundary condition database via any 2d_bc layers and after any adjustments (eg. time shift). Provides traceability to original data source. Note: the boundary values do not include the effects of any 2d_bc attributes such as f.
_dom_check.mif	Contains a rectangle for each 2D domain. Incorporated in Build 2005-05-AN. See also
_grd_check.mif	GIS .mif/.mid files of the final 2D grid. Represents the final grid including modifications from the .tgc file, boundary specifications and flow constrictions. Note that the Material and bed resistance (eg. Mannings_n) attributes do not include any modifications due to flow constrictions as these are applied directly to the cell mid-sides (rather than the cell centre). To view these use the _uvpt_check.mif file. Can also be written at different stages within a .tgc file (see Write MI Grid). The file contains all modifications to the 2D grid at the point in the .tgc file that it is written.
_bc_check.mif	Not available as of Build 2006-06-AA – replaced by the _bcc_check.mif layer. GIS .mif/.mid files of the final 2D boundary conditions (BC). Note, the layer does not include any 2D/1D connections (“CN” type).
_bcc_check.mif	New as of Build 2006-06-AA. GIS .mif/.mid files that has replaced the _bc_check.mif layer. The layer provides trace back information and uses cells, rather than point/line objects to show 2D BCs. The BC ID (eg. BC000001) allows easy trace back to the 2d_bc_tables_check.csv file.
_fc_check.mif	GIS .mif/.mid files of the final arrangement of flow constrictions (FC). The flow constrictions are written as individual square cells of the same shape as the grid cells, even if the FC was specified using points or lines/polylines.
_fcsh_uvpt_check.mif	New as of Build 2008-08-AA. Contains information on adjustments to the ZU/ZV cell sides as modified by Read GIS FC Shape commands.
_glo_check.mif	GIS .mif/.mid files of any gauge level output (GLO) location.

Filename or Prefix/Suffix	Description
_lfsh_uvpt_check.mif	New as of Build 2008-08-AA. Contains information on adjustments to the ZU/ZV cell sides as modified by Read GIS Layered FC Shape commands.
_lp_check.mif	GIS .mif/.mid files of any 2D longitudinal profile(s).
_po_check.mif	GIS .mif/.mid files of any 2D plot output location(s). The layer shows points and lines occurring from the cell centres, rather than their exact locations in the original file(s).
_sac_check.mif	New as of Build 2008-08-AA. Shows the lowest cell selected for Read GIS SA inputs and cells selected if using Read GIS SA PITS.
_sh_obj_check.mif	New as of Build 2008-08-AA. Contains objects such as buffer polygons used for wide lines, triangles generated for TINs within polygons, and regions and polylines for thick and thin lines to illustrate areas that have been modified by Create TIN Zpts (if the WRITE TIN option is specified), Read GIS Z Shape , Read GIS Variable Z Shape , Read GIS FC Shape and Read GIS Layered FC Shape commands.
_uvpt_check.mif	New as of Build 2006-06-AA. GIS .mif/.mid files containing the initial velocities, roughness value, FLC, W _r F, FC lid depth and FC BD factor at the U and V points. For materials that vary Manning's n with depth, the Manning_n attribute contains the Manning's n value at the higher depth (Incorporated in Build 2007-07-AE).
_vzsh_zpt_check.mif	New as of Build 2008-08-AA. Contains information on Zpts that have been modified by Read GIS Variable Z Shape commands.
_zln_zpt_check.mif	New as of Build 2006-06-AA. GIS .mif/.mid files containing Zpts that have been modified by Read GIS Z Line commands, the type of Z Line and the Z Line filename. This feature is very useful for checking which Zpts that the Z Lines have modified. Note: It does not include any GULLY lines. As of Build 2008-08-AA, the points are given different symbology according to whether they have been raised or lowered (up or down triangles). There is also a new attribute dZ which is the amount by which the elevation has changed.
_zsh_zpt_check.mif	New as of Build 2008-08-AA. Contains Zpts that have been modified by Read GIS Z Shape commands. The points are given different symbology according to whether they have been raised or lowered (up or down triangles). The layer includes an attribute dZ which is the amount by which the elevation has changed.
_zpt_check.mif	GIS .mif/.mid files of the final 2D Zpts. Represents the final Zpts including all modifications from the .tgc file, and any flow constrictions in the .tcf file. Can also be written at different stages within a .tgc file (see Write MI Zpts). The file contains all modifications to the 2D Zpts at the point in the .tgc file that it is written. This allows checking of the elevations at different stages of building the topography.

Filename or Prefix/Suffix	Description
1D Domains	
_1d_bc_tables_check.csv	Tabular data as read from the boundary condition database via any 1d_bc layers and after any adjustments (eg. time shift). Provides full traceability to original data source.
_1d_ta_tables_check.csv	Tabular data as read from tables via the 1d_tab layers for cross-section, storage and other data. Provides full traceability to original data source and additional information such as hydraulic properties determined from a cross-section profile. As of Build 2003-10-AA, ISIS XZ and processed, and MIKE 11 processed cross-section data included.
_bc_check.mif	GIS .mif/.mid files of the final 1D boundary conditions (BC). If no boundary conditions were specified, empty .mif/.mid files are written that can be used to set up a new layer.
_1d_hydrop_check.mif	Contains the hydraulic properties at the top of the hydraulic properties tables as attributes of the 1D channels. Other information such as the primary Manning's n is also provided. Very useful for carrying out quality control checks on the 1D channels. Incorporated into Build 2003-07-AE. As of Build 2005-05-AN, attributes containing the upstream and downstream node IDs have been added.
_1d_inverts_check.mif	Contains the inverts of the 1D nodes and at the ends of the 1D channels. Very useful for checking for smooth transitions from one channel to another and with the nodes. Incorporated into Build 2003-05-AG.
_iwl_check.mif	GIS .mif/.mid files of the initial water levels at the 1D model nodes.

Filename or Prefix/Suffix	Description
_nwk_C_check.mif _nwk_N_check.mif	GIS .mif/.mid files of the final 1D model network. As of Build 2008-08-AA, this layer is split into two layers. The _nwk_check.mif layer contains the channels, and the _nwk_N_check.mif layer represents the nodes. This makes it easier to query and view the 1D network. For the 2010-10 release the _nwk_check layer has been renamed _nwk_C_check and has two new attributes: • “i_1d_nwk” is an integer indicating which 1d_nwk input layer the channel came from (in order that they occur in the input). • pSlope is the percentage slope of the channel (allows easier identification of steep channels). Note: Prior to Build 2008-08-AA, the channels are not written as exactly the same polylines as this information is not retained during the data input process. Note: Prior to Build 2008-08-AA, the UCS (Use_Chan_Storage_at_Node) attribute is always shown as F (false), and information supplied in the Topo_ID, Branch, Chainage and some other fields maybe not be as per the original data as this information is not available at the time the check file is written. As of Build 2005-05-AN the following additions/changes occurred: • Node symbology is displayed as a red circle for nodes connected to two or more channels, a larger magenta circle for nodes connected to one channel and a large yellow square for nodes not connected to a channel. This is very useful for checking for channel ends or nodes that are not snapped. • Any generated pit channels are shown as a small channel flowing from north to south into the pit node. The upstream pit channel node that is generated is also shown. The length of the pit channel is controlled by Pit Channel Offset. • The top and bottom elevations of the NA table at nodes is now shown using the Upstream_Invert and Downstream_Invert attributes.
_mhc_check.mif	Manhole check layer including any automatically created manholes.
_wllo_check.mif	GIS layer of all the WLL objects read. The attributes provide information on which nodes that area associated with, etc.
_wllp_check.mif	GIS layer of where the points were generated along the WLLs. These points can then be used for Read GIS WLL Points (see Section 4.11.2.2).
2D/1D Models	

Filename or Prefix/Suffix	Description
_1d_to_2d_check.mif	Displays the 2D cells connected to 1D nodes via 2D HX and 2D SX 2d_bc objects. Cells connected to the same node are given the same colour to allow for easy visualisation of whether the right connections have been made. Additional information is supplied through the attributes. Incorporated into Build 2003-06-AB.

7.3 2D Domains

7.3.1 SMS (Map) Output (.dat or .xmdf Files)

TUFLOW SMS formatted output produces the files described in Table 7.2. The range of map output is controlled by [Map Output Data Types](#).

The 2010-10 release introduces an alternative format to the .dat files used in all prior releases. The new .xmdf format offers a range of advantages over the .dat format:

- All map output is in the one file (ie. all .dat files become one .xmdf file).
- Much faster to access and instant load up when first opened due to an indexing system located in the file header.
- Data are stored in a folder structure that software such as SMS can use to access the data. See image to right as an example.
- Data stored as either static or temporal (time) based.
- Format is an industry standard and recognised by other software. Executable code available to write applications that access the data.

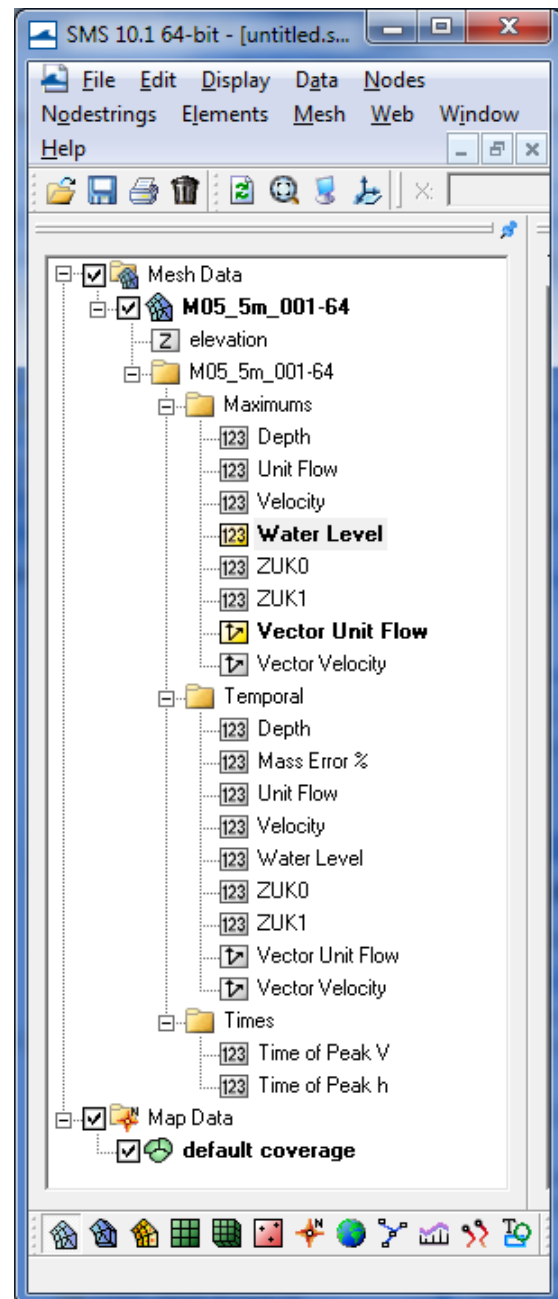
The main reason not to use .xmdf files is the utility programs such as TUFLOW_to_GIS and dat_to_dat are not yet updated to work with .xmdf files (plan is to update these programs early 2011).

To output in the .xmdf format use [Map Output Format](#) == XMDF.

The envelope of maximum and/or minimum values is available for some output types using the options in [Store Maximums and Minimums](#). Minimums are assigned a time of -99999.0 and maximums a time of 99999.0.

For the 2009-07 release or later:

- The new _Times.dat file contains the time the maximum velocity value occurred (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 and the time of minimum values under time 910,001).

- The time of peak water level is no longer written to the `_h.dat` file as time 99999.1.
- Additional time outputs are available using [Time Output Cutoff Depths](#) to output maps of the duration of inundation and the time of first inundation above specified depth(s) to the `_Times.dat` file.
- Hazard output has been extended to include up to 10 different hazard outputs per simulation, and 5 more hazard output classifications have been added (Z5 to Z9), and are available for users to customise via the `TUFLOW_USER_DEFINED.dll`. Each hazard is tracked for its maximum if maximums output is on (see [Store Maximums and Minimums](#)). The code used for calculating the hazard outputs is available from support@tuflow.com.
- For the 2010-10 release the maximum (or minimum) output for unit flow (`_q.dat`) and energy (`_E.dat`) has been disabled as the time of peak velocity is no longer necessarily at the same time of peak water level. These outputs relied on the maximum water level and maximum velocity being synchronised in time, which is not necessarily the case if using the [Maximum Velocity Cutoff Depth](#) feature, which is activated by default for the 2010-10 release.

For releases prior to 2009-07 note that:

- the time at which the maximum water level occurred is stored in the `_h.dat` file and assigned a time of 99999.1
- for some data types such as velocity (`_V.dat`), unit flow (`_q.dat`) and energy (`_E.dat`), the minimum and maximum output is when the minimum or maximum water level occurs (not when the minimum or maximum actually occurs). This is because high velocities can briefly occur during the wetting process, and are not particularly representative of the peak velocity.

Note: In SMS Version 9.2 or higher when reading `.dat` files, the maximums and minimums are automatically filtered out as separate data sets, therefore the 99999 times do not appear in the time list. They are given the extensions `_max`, `_min` and `_maxtime`.

The SMS super (`.sup`) file containing the various files and other commands that make up the output from a single simulation. Opening the `.sup` file in SMS opens the `.2dm` file containing the model mesh and the any of the `_h`, `_V`, `_q` and `_d` `.dat` files. Other `.dat` files (whether from the same simulation or another simulation) are opened in SMS using `File, Open`. If the `.sup` file is not used to open the results, the `.2dm` file must be opened before opening any `.dat` files. If opening `.dat` files from another simulation, the number and location of non-land (Code $\neq$ 0) cells must be the same in both simulations.

The SMS Data Calculator feature is useful for comparing the results of different simulations.

Table 7.2 SMS (Map) Output Files and XMDF Types

File Extension and XMDF Data Type	Description	Flag
.hV.sup .ALL.sup .xmdf.sup	SMS super file containing the various files and other commands that make up the output from a single simulation. Opening the .hV.sup file in SMS opens the .2dm file and the _h.dat and _V.dat files. The .ALL.sup file opens all the .dat files specified using Map Output Data Types. For the XMDF format there is only the one .sup file as all map output is stored within the one .xmdf file. The .2dm, .dat and .xmdf files can also be opened in SMS individually by using <u>F</u>ile, <u>O</u>pen..., or dragging and dropping from Windows Explorer onto SMS. Note: the .2dm file must be the first file opened.	n/a
.2dm	An SMS two-dimensional mesh file containing the information on elements' and nodes' location, shape and the connectivity between elements and nodes. It also contains information on the different materials and cell codes (display the SMS mesh materials). Note that the elevations (bathymetry) in the .2dm file only show the ZH values (ie. top right corner of cell). Other Zpts cannot be shown (as yet). Additional information for each element that is not used by SMS, is used by the utility program sms_to_mif.exe to convert the .2dm file to a GIS layer. In the TUFLOW .2dm's present format, nodes only occur at the corners of the cells (elements). The bed elevations at the nodes are set to the ZH values. All hydraulic parameters are interpolated to the nodes (cell corners).	n/a
_AP.dat Atmospheric Pressure	SMS scalar data file containing the atmospheric pressure in hPa. Maximum and minimum output is not available.	AP
_BSS.dat Bed Shear Stress	Bed Shear Stress as given by the equation below where ρ is density, g gravity, V velocity, n Manning's n and y depth. $\tau_{bed} = \frac{\rho g V^2 n^2}{y^{1/3}}$	BSS
_Cr.dat Courant No.	Courant number (2D domains only at present)	Cr
_CWF.dat Cell Width Factor	Cell side flow width factor. Used to monitor over time the changes in cell side widths if using Read GIS Layered FC Shape . Only available for the Map Output Format == SMS HIGH RES option.	CWF

File Extension and XMDF Data Type	Description	Flag
_d.dat Depth	SMS scalar data file containing water depths at the nodes (cell corners). The depths are calculated as the interpolated water level at the nodes (see _h.dat below) less the ZH value. The interpolated water level may occasionally lie below the ZH value, in which case a negative depth may result which is set to zero by default (see Zero Negative Depths in SMS). Both maximum and minimum output is available.	d
_E.dat Energy	SMS scalar data file containing the energy levels at the element nodes (cell corners). The energy levels are based on the interpolated water levels calculated at the cell centres plus the dynamic head ($V^2/2g$). Due to the interpolation, occasionally an “increase” in energy can occur - an alternative approach to correctly display energy without interpolation is being trialled using the HIGH RES options (see Map Output Format). 1D energy output was included in Build 2006-06-AA. This output should be treated with caution as it is derived from interpolation of water levels and approximations of the channel velocities across the WLLs, which can be problematic in 1D channels with high velocities. Maximum energy levels were incorporated in Build 2003-03-AE. The maximum output is for when the maximum water level occurs (Note: This may cause undulations in the energy due to variations in the time of the maximum water level, except for the HIGH RES options (see Map Output Format), which monitor the energy level every timestep to set the maximum.)	E
_F.dat Froude No	SMS scalar data file containing the Froude Number at the element nodes (cell corners). No maximum and minimum output is available at this stage.	F
FLC Form Loss Coeff	Form Loss coefficient. Used to monitor over time the changes in the form loss coefficient if using Read GIS Layered FC Shape . Only available for the Map Output Format == SMS HIGH RES option.	FLC
_h.dat Water Level	SMS scalar data file containing water levels at the nodes (cell corners). The water levels are interpolated from the water levels calculated at the cell centres. Both maximum and minimum output (times 99999 and – 99999) are available. Time 99999.1 is the time in hours that the peak water level occurred. If using the HIGH RES options (see Map Output Format), interpolated water levels allow for the effects of upstream controlled flow regimes (eg. supercritical flow).	h

File Extension and XMDF Data Type	Description	Flag
_MB1.dat MB1	2010-10 release or later: Measure of the convergence level of the solution. The measure is a cumulative value since the last output time, therefore is an effective way of identifying problem areas in a model that repeatedly have poor convergence and most likely mass error. Very useful for identifying problem areas within a model. Prior to the 2010-10 release: Percentage mass error at 2D cells, however, can be misleading as output is at an instant in time and can produce high values in areas of very shallow flows due to imprecision's in the calculation. Does not include 1D output from WLLs for all releases. Changed from _ME.dat to _MB1.dat for Build 2007-07-AF.	MB1
_MB2.dat MB2	Introduced for the 2010-10 release: Same as MB1 above, but is accumulated over the entire simulation. Does not include 1D output from WLLs for all releases.	MB2
_n.dat Manning n	Manning's n values. The n values only vary over time for materials using the Manning's n varying with depth feature. The output was enhanced to cater for Bed Resistance Cell Sides == INTERROGATE in Build 2007-07-AE. The n values at the cell corners in the _n.dat file are interpolated from the surrounding four cell mid-sides.	n
_q.dat Vector Unit Flow	SMS vector data file of unit flow (m²/s, ie. flow per unit width) at the nodes (cell corners). The resulting flow vector is calculated from the surrounding u and v-points and the depth determined in _d.dat above. Unit flow may also be used as a measure of flood hazard (ie. velocity by depth or VxD). Note: The maximum unit flow output (time 99999.0) is for when the maximum water level occurs, except for the HIGH RES options (see Map Output Format), which monitor the unit flow every timestep to set the maximum. The Z0 hazard value option can also be used.	q
_R.dat Flow Regime	SMS scalar data file containing a number indicating the flow regime. The value is 0 (zero) for normal (sub-critical flow with momentum); greater than 1 for upstream controlled friction flow (eg. supercritical flow); -1.5 for broad-crested weir flow; and -1 for flow through a flow constriction when the deck is submerged. No maximum and minimum output is available at this stage.	R
_SP.dat Stream Power	Stream Power as given by the equation below where τ_{bed} is bed shear stress (see BSS above) and V is velocity. $\tau_{bed} = \frac{\rho g V^2 n^2}{y^{1/3}}$	SP

File Extension and XMDF Data Type	Description	Flag
_SS.dat Source Flow	The net source/sink inflows. Note the flow rate for a cell is shown at the ZH point (top right of the cell), except for the HIGH RES options (see Map Output Format), which are spatially correct (note the HIGH RES CORNERS ONLY option will interpolate sink/source flow rates to the cell corners).	SS
_t.dat Viscosity Coeff	SMS scalar data file containing the variation in eddy viscosity coefficient. This is useful for checking the Smagorinsky coefficient values. Prior to Build 2002-10-AH was named _e.dat. No maximum and minimum output is available at this stage.	t
_V.dat Vector Velocity	SMS vector data file of flow velocities at the nodes (cell corners). The resulting velocity vector is calculated from the surrounding u and v-points. Note: The maximum and minimum velocities (Times 99999.0 and -99999.0) are when the maximum and minimum water level occurs.	V
_Z0.dat Z0	VxD product	Z0

File Extension and XMDF Data Type	Description	Flag
_Z1.dat Z1	Flood hazard category based on the Australian NSW Floodplain Management Manual. The output is a number from 1 to 3 as follows and as illustrated in the figure below. 1 Low Hazard 2 Intermediate Hazard (dependent on site conditions) 3 High Hazard Note: The maximum hazard value (Time 99999.0) is monitored throughout the simulation and is not necessarily when the maximum water level occurs as with some other output.	Z1
_Z2.dat Z2	$V > 2.0$: Category 5 (Extreme Hazard) $D * V > 1.0$: Category 4 (High Hazard Floodway) $D > 1.0$: Category 3 (High Hazard Depth) $V + 3.3 * D \leq 2.64$: Category 2 (High Hazard) Otherwise Category 1 (Low Hazard) Dry points are assigned Category 0	Z2

File Extension and XMDF Data Type	Description	Flag
_Z3.dat Z3	$V > 2.4$: Category 7 (Extreme Hazard) $D * V > 2.0$: Category 6 (Extreme Hazard Floodway) $V > 2.0$ OR $V * D > 1.0$: Category 5 (High Hazard Floodway) $D > 2.0$: Category 4 (Extreme Hazard Depth) $D > 1.0$: Category 3 (High Hazard Depth) $V + 3.3 * D \leq 2.64$: Category 2 (High Hazard) Otherwise Category 1 (Low Hazard) Dry points are assigned Category 0	Z3
_Z4.dat Z4	Flood hazard mapping approach based on the Australian Guidelines using the following logic: $D \leq 0.3$ AND $V \leq 0.38$: Category 1 (Low Hazard) $D \leq 0.6$ AND $V \leq 0.8$ AND $D + 0.64 * V \leq 0.82$: Category 2 (Intermediate Hazard) $D \leq 1.2$ AND $V \leq 1.5$ AND $D + 0.69 * V \leq 1.38$: Category 3 (High Hazard) Otherwise Category 4 (Extreme Hazard) Dry points are assigned Category 0	Z4
_ZMW1.dat ZMW1	Based on Melbourne Water FM&M Technical Specifications to quantify Property Safety Risk. The logic used is as follows: $D * V < 0.2$: Category 1 $D * V < 0.4$: Category 2 $D * V < 0.6$: Category 3 $D * V < 0.84$: Category 4 Otherwise Category 5 Dry points are assigned Category 0	ZMW1
_ZMW2.dat ZMW2	Based on Melbourne Water FM&M Technical Specifications to quantify Safety Risk in Roads. The logic used is as follows: $D * V < 0.2$ and $D < 0.2$: Category 1 $D * V < 0.4$ and $D < 0.4$: Category 2 $D * V < 0.6$ and $D < 0.6$: Category 3 $D * V < 0.84$ and $D < 0.84$: Category 4 Otherwise Category 5 Dry points are assigned Category 0	ZMW2
_ZUK0.dat ZUK0	The value of the UK Hazard formula (see UK Hazard Formula , UK Hazard Land Use and UK Hazard Debris Factor).	ZUK0

File Extension and XMDF Data Type	Description	Flag
_ZUK1.dat ZUK1	The UK Hazard category as shown below (also see UK Hazard Formula , UK Hazard Land Use and UK Hazard Debris Factor).	ZUK1
_ZH.dat Bathymetry	Elevations at the cell corners (ZH points). This information is already contained in the .2dm file, however, this option is useful if the model's bathymetry varies over time if using variable geometry (VG) boundaries or for morphological modelling. No maximum and minimum output is available at this stage.	ZH

7.3.2 Time-Series Output

Time series data output is available in the following forms:

- `_PO.csv` and `_LP< name>.csv` files (also referred to as plot output (PO) or longitudinal profile (LP) data) created using `2d_po` and `2d_lp` layers (see Section 4.8). These files are typically used in spreadsheet software for graphing and analysing time-series results.
- In the `_TS.mif` file (`2d_po` locations only). The `_TS.mif` file also contains all 1D time based output. This file is used for graphing time series output within a GIS (see Section 7.4.4 and Figure 7-1 for an example).

Use [Start Time Series Output](#) and [Time Series Output Interval](#) to control the output times.

7.3.3 Conversion to GIS Formats

The utilities `TUFLOW_to_GIS.exe` (Section 11.2), `data_to_dat.exe` (Section 11.3) and `asc_to_asc.exe` (Section 11.6) offer a range of options for post-processing and translating TUFLOW map output into GIS and 3D grid formats for high quality mapping and reporting of results.

7.4 1D Domains

7.4.1 Output File (.eof file)

The .eof file is ESTRY's original output file, although it is also used to output additional information to assist with checking of data input and warning/error trapping. If the file when opened in UltraEdit appears in hexadecimal format, press Ctrl H to view as text (this is only for earlier versions of TUFLOW).

The file is very useful for checking input data. It contains a complete output of the final input data before the simulation commences. For example, if a second table overwrites a channel cross-section properties table during the input process, the table in the .eof file is that of the second table. Therefore, the .eof file contains the data actually used for the simulation. Note that adjustments to data, for example, a datum shift in a gradient channel's cross-section based on the upstream and downstream inverts, are also incorporated.

The .eof file also contains the complete results of the simulation, including useful information such as culvert flow regimes at each output time, time of maximum water level, etc. The channel and node regime flags are located in the two spaces after the velocity, flow and head values in the time based output. The flags, described in Table 7.3, are useful for interrogating the hydraulic regime at nodes and channels.

As of Build 2007-07-AA, most of the channel regime flags were moved from Flag Space 1 to Space 2.

Table 7.3 Channel and Node Regime Flags (.eof File)

Flag (Space 1)	Flag (Space 2)	Description
*		The depth at a node fell below -0.1m. A WARNING is also output to the _messages.mif file. The occurrence of significant negative depths may cause mass conservation errors in the 1D domain.
	*	One end of a normal channel is close to being dry and a transitioning algorithm was used to dry/wet the channel.
	#	The gradient channel algorithm was applied. This occurs when one end of the channel is either dry or very shallow. The gradient channel algorithm applies a weir equation at the dry or shallow end in combination with the momentum equation by adjusting the water surface slope along the channel.
	A	Adverse flow, ie. flow gradient is against the slope of the channel.
	D	Upstream controlled friction flow occurred in a Steep (S) channel when the downstream end was dry (Build 2004-06-AA).
	S	Upstream controlled friction flow occurred in a Steep (S) channel with a Froude Number greater than one (1).

Flag (Space 1)	Flag (Space 2)	Description
	T	Upstream controlled friction flow occurred in a Steep (S) channel with a Froude Number between 0.5 and one (1). T stands for Transitioning from normal flow to upstream controlled friction flow.
	N	Upstream controlled friction flow occurred in a Steep (S) channel with a Froude Number less than 0.5. N stands for normal flow, however, in this case the upstream controlled friction flow approach was adopted. This may occur during the transitioning of flow from downstream controlled to upstream controlled. If it occurs repetitively, the configuration of the channel should be reviewed.
	Culvert Flow Regime Flag	The culvert flow regime flag as documented in Table 4.23. Culvert channels only.
	E	The node is empty or dry (ie. the head or water level is at the bottom of the node). E stands for Empty.
	E	The channel is empty or dry. E stands for Empty.
	F	The head exceeds the top of the nodes elevation versus surface area table (NA table). F stands for Full.
	F	The head at the mid-point of the channel exceeds the top of the channel's hydraulics properties table (CS table). F stands for Full.
L		The velocity rate limit was applied to the channel to try and prevent oscillations or instabilities – non-inertial channels (structures) only. See Vel Rate Limit .
	U	The uni-directional flag assigned to the channel was invoked and the velocity/flow was set to zero.

7.4.2 SMS Output

1D domain results can be output in combination with 2D domain(s) by using the 1d_wll layer (see Section 4.11 and [Read GIS WLL](#)). Note: viewing the results must be carried out in SMS Version 8.

In viewing the results in SMS, if the 1D and 2D domains overlap, the 1D results are displayed on top of the 2D results. However, when observing the scalar and vector magnitudes as the pointer is moved around, the 2D values are given precedence over the 1D where overlap occurs. This is a characteristic of SMS.

7.4.3 GIS and Text 1D Domain Check Files

GIS check files of the network, initial water levels and boundary conditions are produced using [Write Check Files](#) (also see Section 7.2). These files are based on the final 1D domain inputs after any overrides. Text .csv files containing any 1d_tab link tables and 1d_bc boundaries are also written so that the tabular input can be cross-checked.

7.4.4 Time-Series Output

Time-series data for water levels at nodes, and flows and velocities in channels are output in three separate formats:

- A .csv file that can be used in spreadsheet software or similar to produce graphs, tables, etc. (the data can be output with values for a node/channel along rows or down columns – see [CSV Format](#)).
- An _TS.mif file that can be used for graphing time series output in a GIS as illustrated in Figure 7-1.
- in the .eof file (good for viewing the time output in a text editor).

The .eof file contains a complete output of all results including flags to indicate the various flow regimes.

7.4.4.1 _TS.mif Layers

_TS.mif layers are GIS layers containing time-series output. The layers available as of Build 2007-07-AA are:

- _TS.mif that contains all 1D channel (velocities and flows), 1D nodes (water levels) and 2D PO (plot output locations from 2d_po layers).
- _TSF.mif contains the flow regime flags for culverts (see Figure 4-6 and Figure 4-7), for other types of channels (see Table 7.3).
- As of Build 2008-08-AC, _TSL.mif contains the culvert and bridge loss coefficients after any adjustments if [Structure Losses](#) == ADJUST or the “A” flag was specified, and/or if there is a manhole connected to one/both ends of a culvert, the loss coefficients will be effected by any manhole energy losses as discussed in Sections 4.5.2.4 and 4.5.2.5. For culverts the three values shown for each time are the Inlet (entry) loss coefficient; Additional loss coefficient (this value is the sum of any 1d_nwk Form_Loss (bend loss) value on the channel or an upstream pit, and any manhole energy loss contribution); and Outlet (exit) loss coefficient; For bridges, the bridge loss coefficient adopted is shown.
- _TSMB.mif that contains the mass errors at 1D nodes.
- _TSMB1d2d.mif that contains the mass errors across 1D/2D interface linkages (HX links).

Each layer contains several attributes at the start summarising the times-series data. These attribute are:

- the maximum and minimum values;
- the time in hours of the maximum and minimum values; and

- for the MB output, the average and the average of the absolute mass error values are provided.

Graphing and tabular viewing tools are supplied for easily viewing times-series data in _TS.mif files in the MapInfo GIS (see Section 11.10).

As of Build 2007-07-AA, the output frequency of the time output in _TS.mif formats is automatically adjusted so that the limit of 245 output times is not exceeded. For example, if based on the [Time Series Output Interval](#) setting there are, say 400 output times, then every second time will be written to the _TS.mif formats giving a total of 200 output times, but at least the full hydrograph can now be displayed! In previous builds all the time-series data was output, which if it exceeded 245 values, then only the first 245 output values could be viewed.

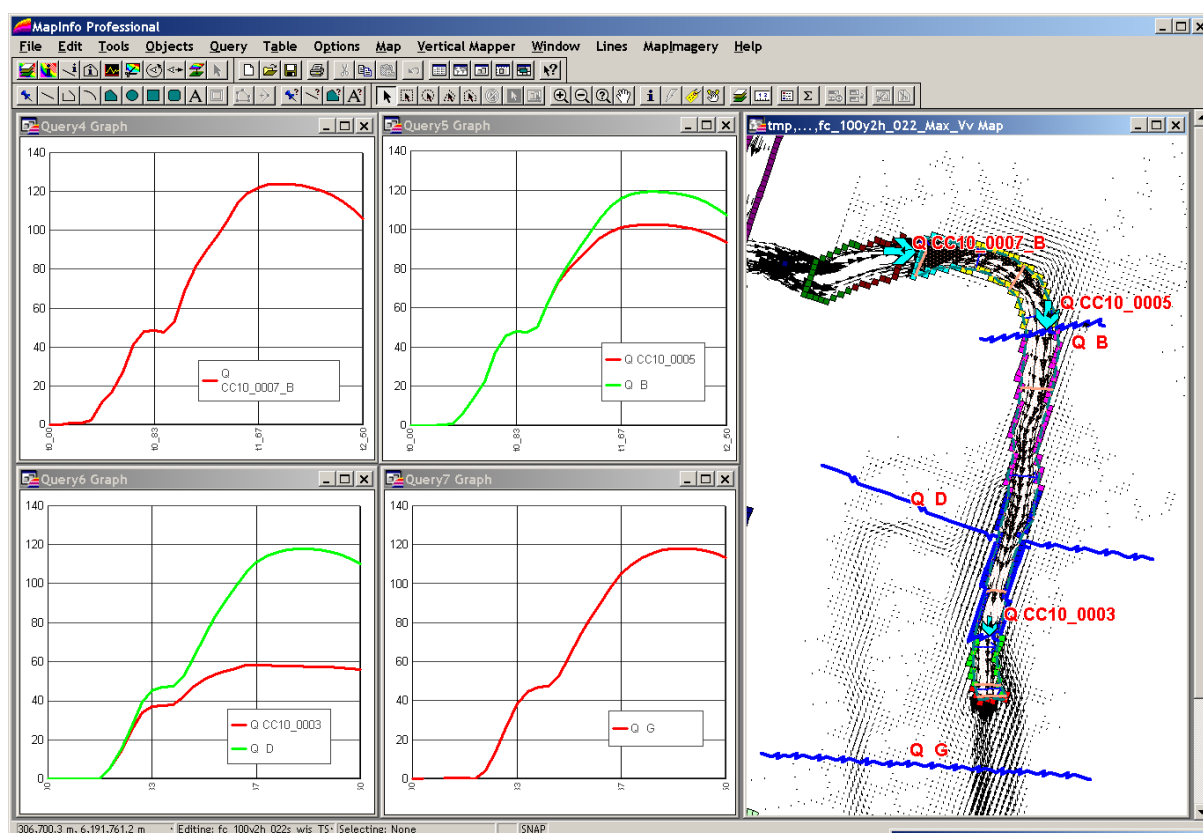


Figure 7-1 Viewing Time-Series Data in MapInfo – Checking Flow Balance in a 2D/1D Model

7.4.4.2 _ccA.mif Layer

The _ccA.mif layer contains information on culverts and bridges (closed channels) using the attributes as follows:

- pFull_Max = The percentage of the peak flow area over the culvert/bridge area. If the culvert/bridge flowed full at any point during the simulation this will be 100%.
- pFull_Time = The percentage of time the culvert/bridge ran full over the time the culvert/bridge ran at least 10% full.
- Area_Max = The peak flow area that occurred during the simulation.
- Area_Culv = The culvert/bridge flow area (when full).
- Time_pfull = The time in hours the culvert/bridge was running full.
- Time_10pFull = The time in hours the culvert/bridge ran 10% full or more.

The layer's lines are coloured and thickened according to the pFull_Max attribute. The thinner and paler the line the smaller the pFull_Max value. The pFull_Time attribute is excellent for thematically mapping in a GIS so as to show the culverts are performing well, and the ones that are not. The example shown in Figure 7-2 illustrates the _ccA.mif output. The thicker and darker the line, the better the pipe performed in terms of reaching full flow capacity.

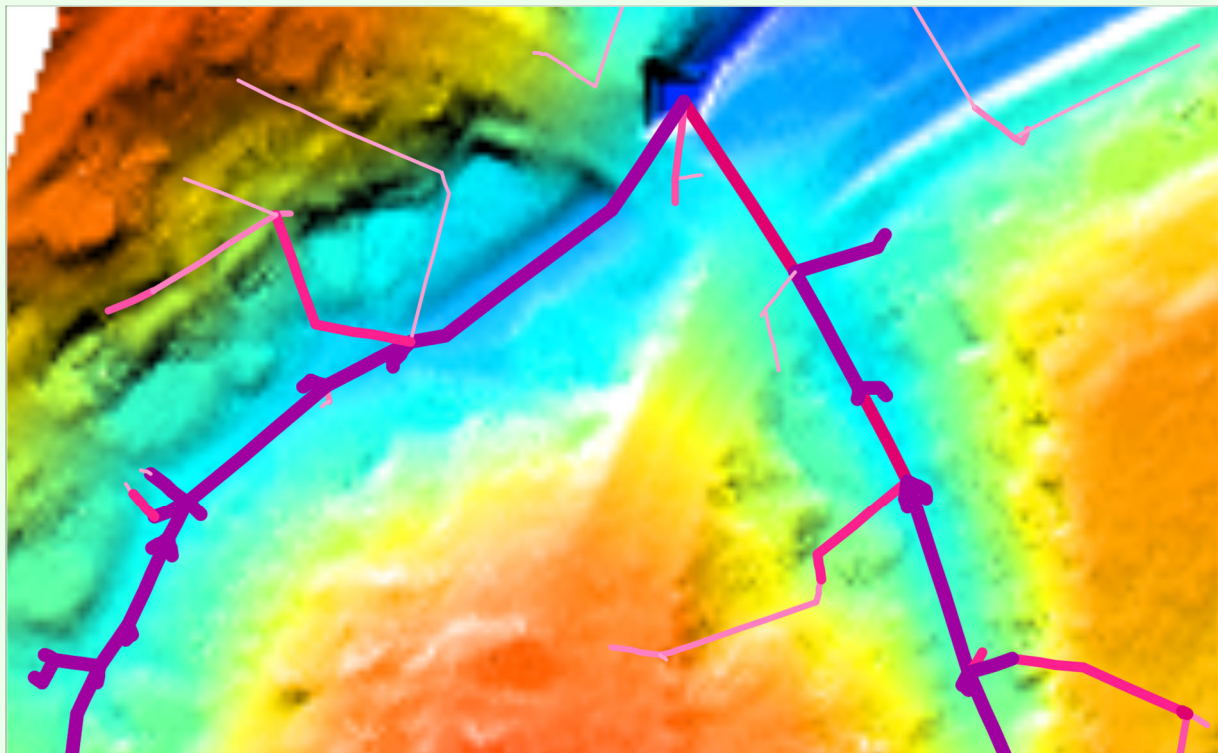


Figure 7-2 Example of the _ccA.mif Layer Showing Culvert Performance

7.4.5 Maximum/Minimum Output

Maximum/minimum values for water levels at nodes, and flows and velocities in channels are output in three separate formats:

- at the top of the .csv files containing the time-series output (see previous section);
- .mif/.mid files with the extensions 1d_mmH, 1d_mmQ and 1d_mmV; and
- at the end of the .eof file (good for viewing the time output in a text editor).

The GIS .mif/.mid files contain the maximum and minimum values, and the time of the maximum and minimum values, for water levels, flows and velocities. The files are given a “1d_mmH”, “1d_mmQ” and “1d_mmV” extension and contain the maximum, minimum, time of maximum and time of minimum values as attributes to the GIS objects at each node or channel. For the flows and velocities, an additional attribute (Qpeak and Vpeak) equal to the maximum of the maximum and minimums in terms of magnitude is provided – this is particularly useful for tidal reaches or where a channel has significant flows in both directions.

The water levels are output at the nodes and flows and velocities at the channel midpoints. The flows and velocities are shown as arrow symbols scaled according to their peak value (if they do not appear as such, then the Wingdings font set is not installed on your computer). The direction of the arrow is in the positive flow direction.

As of Build 2007-07-AA, a new and useful attribute, dH, was added to the 1d_mmH.mif output layer. dH contains the largest water level drop across the channels that end at that node owning the attribute. Only channels that are digitised so that their downstream end is at the node are used to determine dH. Provided channels are digitised from upstream to downstream this is useful for identifying any increases in water level caused by any instabilities (thematically map the dH attribute – negative values indicate the water levels are increasing downstream). Pit channels are excluded from determining dH.

7.5 Mass Balance Output

Mass balance information is generated by setting [Mass Balance Output](#) to ON (the default as of Build 2006-06-AA). Note: once the model has been established and mass balance checks have been carried out, there would be some benefit in setting [Mass Balance Output](#) to OFF to reduce run-times.

If [Mass Balance Output](#) is ON the following mass balance output is available:

- As discussed in Section 7.1.1 the cumulative mass error percentage appears as three numbers after the letters “CE” on the Console Window. The first percentage is the overall model (all 1D and 2D domains), the second is for all of the 1D domains and the third for all of the 2D domains (see the description of the “Cum ME (%)” column in
- Table 7.4, Table 7.5 and Table 7.6. Ideally all these percentages should tend to be within $\pm 1\%$. Monitoring these numbers is important so as to establish the “healthiness” of the model as discussed in Section 8.1. If at the start of a simulation, especially if there are 2D domain(s) rapidly wetting, much higher values occur, this should not be an issue provided the model quickly settles down and the CE percentages fall within acceptable amounts.
- Three **_MB.csv** files are output reporting on the various inflows and outflows, volume, predicted volume error and the mass and cumulative mass errors as a percentage as follows:
 - The **_MB.csv** file is for the overall model (all 1D and 2D domains). It is written to the .tcf [Output Folder](#).
 - Table 7.4 provides a description of the data in the file.
 - The **_MB1D.csv** file contains mass balance reporting for all the 1D domains. It is written to the .ecf [Output Folder](#). Table 7.5 provides a description of the data in the file.
 - The **_MB2D.csv** file contains mass balance reporting for all 2D domains together and for each individual 2D domain. It is written to the .tcf [Output Folder](#). Table 7.6 provides a description of the data in the file.
- In and out flows across HX and SX connections for 1D/2D links into those between TUFLOW’s 1D scheme (ESTRY) and those to an external 1D scheme. Overall mass balance reported does not include flows in any external 1D scheme, but does include flows across links to an external 1D scheme.
- Map output of the 2D mass error can be output by specifying the MB1 and/or MB2 option for [Map Output Data Types](#). The MB1 has changed for the 2010-10 release and provides a much more accurate depiction of any problem areas within the 2D domain(s).
- Time based 1D mass error is output as a GIS layer to a **_TSMB.mif** layer (see Table 7.7 for a description of the attributes). Using GIS thematic mapping of the ME_Avg_Abs attribute is a powerful way of identifying any problematic 1D nodes.
- Time based mass error reporting across 1D/2D HX links is output as a GIS layer to a **_TSMB1d2d.mif** layer. Note that when viewing this layer that each 1D node point object is connected to a collection of 2D cell objects that make one overall GIS object (called a Collection). This layer is therefore useful for identifying which 2D cells are connected to a

1D node for the 2D HX links. See Table 7.8 for a description of the attributes. Using GIS thematic mapping of the ME_Avg_Abs attribute is a powerful way of identifying any problematic 2D HX links.

For the 2010-10 release the mass balance output has been reworked so that the display on the screen and the _MB.csv output files can be at different time intervals. The new tcf command [Mass Balance Output Interval](#) is used for setting the interval in the _MB.csv files. There is also now a summary at the end of the simulation on the display console and .tlf file of key model performance indicators (see Section 7.6).

For builds prior to the 2010-10 release the mass balance reporting is at the same time interval as the display to the Console Window, ie. as set by [Screen/Log Display Interval](#) (the default being every timestep).

Healthy models fall within $\pm 1\%$ cumulative mass error (see Section 8.1 for discussion on “unhealthy” models). If a model experiences higher mass errors this may be due to the following.

- Using too large a timestep and/or areas of the model are sensitive or slightly unstable.
- Models with significant areas of complex, steep flows, that use the direct rainfall approach ([Read GIS RF](#)). These models should be run using the double precision version of Build 2008-08-AA or later (without setting any backward compatibility switches) - see Section 5.2.
- High frequency of repeated wetting and drying.
- Models located well above sea level (ie. have high elevations), especially if the inflows are relatively small or direct rainfall is applied. These models should be run using the double precision version of Build 2008-08-AA or later (without setting any backward compatibility switches) - see Section 5.2. There are no fixed guidelines for when to switch to double precision, other than to carry out sensitivity tests using single and double precision versions (as a general rule all direct rainfall models and models with elevations greater than 100 to 1,000m usually require or will benefit from using double precision).
- Direct rainfall and/or steep models need to utilise very small wet/dry depths ([Cell Wet/Dry Depth](#) and [Cell Side Wet/Dry Depth](#)) to minimise mass errors that can arise from cells frequently wetting and drying with larger wet/dry depths, so the default settings or smaller should be used.
- Prior to Build 2008-08-AA, it was possible to (accidentally) specify a 1D timestep larger than the smallest 2D timestep if using multiple 2D domains.

Mass errors occur at 1D nodes that are frequently drying (undershooting), or are being limited if [Head Rate Limit](#) is being used. For the _TSMB.mif and _TSMB1d2d.mif output the mass error values are given as a flow rate (m^3/s) so that they can be compared with the total flow through the model at that location, ie. a mass error of $1\text{m}^3/\text{s}$ at a node where $1,000\text{m}^3/\text{s}$ is passing through is not an issue, while it would be if only $2\text{m}^3/\text{s}$ was passing through the node. Percentage mass errors as provided in _MB.csv files is planned for future releases.

Note that the calculation of mass errors is in itself an estimation and has errors associated with the calculation process. It is also recommended that conventional mass balance checks be carried out as a matter of course to cross-check (see Section 8.2).

Table 7.4 _MB.csv File Columns

Column	Description
Time (h)	The simulation time in hours.
H Vol In	The volume of water in m ³ flowing into the model across water level (HQ, HS, HT) boundaries since the previous time.
H Vol Out	The volume of water in m ³ flowing out of the model across water level (HQ, HS, HT) boundaries since the previous time.
Q Vol In	The volume of water in m ³ flowing into the model from flow (QH, QS, QT, RF, SA, ST) boundaries since the previous time.
Q Vol Out	The volume of water in m ³ flowing out of the model across flow (QH, QS, QT, RF, SA, ST) boundaries since the previous time.
Tot Vol In	The total volume of water in m ³ entering the model since the previous time.
Tot Vol Out	The total volume of water in m ³ leaving the model since the previous time.
Vol I-O	“Tot Vol In” minus “Tot Vol Out”, ie. the net volume of water in m ³ entering the model since the previous time.
dVol	The change in the model’s volume in m ³ since the previous time.
Vol Err	“dVol” minus “Vol I-O”, ie. the volume error or amount of water in m ³ unaccounted for since the previous time. A positive value indicates the solution may have gained mass, while a negative value indicates a possible mass loss.
Q ME (%)	(“Vol Err”/“Vol I+O”)*100, ie. the percentage mass error based on the volume of water flowing through the model since the previous time. This figure can be large at the start of a simulation (particularly if using default settings prior to those for Build 2008-08-AA) if there are 2D domains rapidly wetting and the flow through the model (“Vol I+O”) is relatively small. This is a particular characteristic of 2D domains, particularly when using the direct rainfall approach for builds prior to Build 2008-08-AA. If “Vol I+O” is less than 1m ³ , “Q ME (%)” is set to zero to avoid divide by zero calculations.
Vol I+O	“Tot Vol In” + “Tot Vol Out”, ie. the volume of water in m ³ entering and leaving the model since the previous time.
Tot Vol	The total volume of water in m ³ in the model.
Cum Vol I+O	The cumulative volume of water in m ³ entering and leaving the model, ie. the cumulative total of “Vol I+O”.
Cum Vol Err	The cumulative volume error in m ³ , ie. the cumulative total of “Vol Err”.

Column	Description
Cum ME (%)	$(\text{"Cum Vol Err"} / \max(\text{"Tot Vol"} \text{ and } \text{"Cum Vol I+O"})) * 100$, ie. the percentage mass error based on the maximum of the volume of water that has flowed through the model and total volume of water in the model. This figure can be large at the start of a simulation if there are 2D domains rapidly wetting and the flow through the model ("Cum Vol I+O") is relatively small. This is a particular characteristic of steep models, particularly when using the direct rainfall approach with default settings prior to those of Build 2008-08-AA. This figure can also be misleadingly low if the model has a very large volume of "stagnant" water such as a lake or part of the ocean. If $\max(\text{"Tot Vol"} \text{ and } \text{"Cum Vol I+O"})$ is less than 1m^3, "Cum ME (%)" is set to zero to avoid divide by zero calculations. This figure is the first number displayed after "CE" on the Console Window.
Cum Q ME (%)	$(\text{"Cum Vol Err"} / \text{"Cum Vol I+O"}) * 100$, ie. the percentage mass error based on the cumulative volume of water that has flowed through the model. This figure can be large at the start of a simulation if there are 2D domains rapidly wetting and the flow through the model ("Cum Vol I+O") is relatively small. This is a particular characteristic of steep models, particularly when using the direct rainfall approach if using builds prior to Build 2008-08-AA. If "Cum Vol I+O" is less than 1m^3, "Cum Q ME (%)" is set to zero to avoid divide by zero calculations.

Table 7.5 _MB1D.csv File Columns

Column	Description
Time (h)	The simulation time in hours.
H V In	The volume of water in m^3 flowing into all 1D domains at 1D water level (HQ, HS, HT) boundaries since the previous time.
H V Out	The volume of water in m^3 flowing out of all 1D domains at 1D water level (HQ, HS, HT) boundaries since the previous time.
SX2D V In	The volume of water in m^3 flowing into all 1D domains from 2D SX links since the previous time.
SX2D V Out	The volume of water in m^3 flowing out of all 1D domains from 2D SX links since the previous time.
Q V In	The volume of water in m^3 flowing into all 1D domains from 1D flow (QH, QS, QT) boundaries, except for 1D QT Regions, since the previous time.
Q V Out	The volume of water in m^3 flowing out of all 1D domains from 1D flow (QH, QS, QT) boundaries, except for 1D QT Regions since the previous time.
QR V In	The volume of water in m^3 flowing into all 1D domains from 1D QT Region flow boundaries, since the previous time.
QR V Out	The volume of water in m^3 flowing out of all 1D domains from 1D QT Region flow boundaries, since the previous time.

Column	Description
Q2D V In	The volume of water in m ³ flowing into hidden 1D nodes from 2D QT flow boundaries, since the previous time.
Q2D V Out	The volume of water in m ³ flowing out of hidden 1D nodes from 2D QT flow boundaries, since the previous time.
HX2D V In	The volume of water in m ³ flowing into all 1D domains across 2D HX links since the previous time.
HX2D V Out	The volume of water in m ³ flowing out of all 1D domains across 2D HX links since the previous time.
Vol In-Out	Sum of all the volumes in less the sum of all the volumes out, ie. the net volume of water in m ³ entering all the 1D domains since the previous time.
dVol	The change in the 1D domains' volume in m ³ since the previous time.
Vol Err	"dVol" minus "Vol In-Out", ie. the volume error or amount of water in m ³ unaccounted for since the previous time. A positive value indicates the 1D domains may have gained mass, while a negative value indicates a possible mass loss.
Q ME (%)	("Vol Err"/(Σ V In + Σ V Out))*100, ie. the percentage mass error based on the volume of water flowing through the 1D domains since the previous time. If (Σ V In + Σ V Out) is less than 1m ³ , "Q ME (%)" is set to zero to avoid divide by zero calculations.
Total Vol	The total volume of water in m ³ in the 1D domains.
Cum Vol In+Out	The cumulative volume of water in m ³ entering and leaving the 1D domains, ie. the cumulative total of (Σ V In + Σ V Out).
Cum Vol Error	The cumulative volume error in m ³ , ie. the cumulative total of "Vol Err".
Cum ME (%)	("Cum Vol Error"/max("Cum Vol In+Out" and "Total Vol"))*100, ie. the percentage mass error based on the maximum of the volume of water that has flowed through the 1D domains and the total volume of water in the 1D domains. This figure can be misleadingly low if the 1D domains have a very large volume of "stagnant" water such as from lakes or part of the ocean. If max("Cum Vol In+Out" and "Total Vol") is less than 1m ³ , "Cum ME (%)" is set to zero to avoid divide by zero calculations. This figure is the second number displayed after "CE" on the Console Window.
Cum Q ME (%)	("Cum Vol Error"/"Cum Vol In+Out")*100, ie. the percentage mass error based on the volume of water that has flowed through the 1D domains. If "Cum Vol In+Out" is less than 1m ³ , "Cum Q ME (%)" is set to zero to avoid divide by zero calculations.

Table 7.6 _MB2D.csv File Columns

Column	Description
Time (h)	The simulation time in hours.
H V In	The volume of water in m ³ flowing into the 2D domain/s at 2D water level (HQ, HS, HT) boundaries since the previous time.
H V Out	The volume of water in m ³ flowing out of the 2D domain/s at 2D water level (HQ, HS, HT) boundaries since the previous time.
Es HX V In	The volume of water in m ³ flowing into the 2D domain/s across HX links to TUFLOW 1D (ESTRY) domains since the previous time. Note, this figure includes any 2D QT boundaries and 2D links as these become HX links connected to hidden 1D nodes.
Es HX V Out	The volume of water in m ³ flowing out of the 2D domain/s across HX links to TUFLOW 1D (ESTRY) domains since the previous time. Note, this figure includes any 2D QT boundaries and 2D links as these become HX links connected to hidden 1D nodes.
x1D HX V In	The volume of water in m ³ flowing into the 2D domain/s across HX links to an external 1D scheme since the previous time.
x1D HX V Out	The volume of water in m ³ flowing out of the 2D domain/s across HX links to an external 1D scheme since the previous time.
SS V In	The volume of water in m ³ flowing into the 2D domain/s from 2D flow sources (RF, SA, SH, ST) boundaries since the previous time.
SS V Out	The volume of water in m ³ flowing out of the 2D domain/s from 2D flow sources (RF, SA, SH, ST) boundaries since the previous time.
Es SX V In	The volume of water in m ³ flowing into the 2D domain/s through SX links to TUFLOW 1D (ESTRY) domains since the previous time.
Es SX V Out	The volume of water in m ³ flowing out of the 2D domain/s through SX links to TUFLOW 1D (ESTRY) domains since the previous time.
x1D SX V In	The volume of water in m ³ flowing into the 2D domain/s through SX links to an external 1D scheme since the previous time.
x1D SX V Out	The volume of water in m ³ flowing out of the 2D domain/s through SX links to an external 1D scheme since the previous time.
V In-Out	Sum of all the volumes in less the sum of all the volumes out, ie. the net volume of water in m ³ entering the 2D domain/s since the previous time.
dVol	The change in the 2D domain/s' volume in m ³ since the previous time.
V Err	“dVol” minus “V In-Out”, ie. the volume error or amount of water in m ³ unaccounted for since the previous time. A positive value indicates the 2D domain/s may have gained mass, while a negative value indicates a possible mass loss.
Q ME (%)	(“V Err”/(ΣV In + ΣV Out))*100, ie. the percentage mass error based on the volume of water flowing through the 2D domain/s since the previous time. This figure can be large at the start of a simulation if the 2D domain/s are rapidly wetting and the flow through

Column	Description
	the 2D domain/s is relatively small. This is a particular characteristic of steep 2D domains, particularly when using the direct rainfall approach for builds prior to Build 2008-08-AA. If (ΣV In + ΣV Out) is less than 1m^3 , “Q ME (%)” is set to zero to avoid divide by zero calculations.
Total V	The total volume of water in m^3 in the 2D domain/s.
Cum V In+Out	The cumulative volume of water in m^3 entering and leaving the 2D domain/s, ie. the cumulative total of (ΣV In + ΣV Out).
Cum V Error	The cumulative volume error in m^3 , ie. the cumulative total of “V Err”.
Cum ME (%)	(“Cum V Error”/max(“Cum V In+Out” and “Total V”))*100, ie. the percentage mass error based on the maximum of the volume of water that has flowed through the 2D domain/s and the total volume of water in the 2D domain/s. This figure can be large at the start of a simulation if the 2D domain/s are rapidly wetting and the flow through the 2D domain/s is relatively small. This is a particular characteristic of steep 2D domains, particularly when using the direct rainfall approach with builds prior to Build 2008-08-AA. This figure can also be misleading low if the 2D domain/s have a very large volume of “stagnant” water such as from lakes or part of the ocean. If max(“Cum V In+Out” and “Total V”) is less than 1m^3 , “Cum ME (%)” is set to zero to avoid divide by zero calculations. This figure is the third number displayed after “CE” on the Console Window.
Cum Q ME (%)	(“Cum V Error”/“Cum V In+Out”)*100, ie. the percentage mass error based on the cumulative volume of water that has flowed through the 2D domain/s. This figure can be large at the start of a simulation if the 2D domain/s are rapidly wetting and the flow through the 2D domain/s is relatively small. This is a particular characteristic of steep 2D domains, particularly when using the direct rainfall approach (for builds prior to Build 2008-08-AA). If “Cum V In+Out” is less than 1m^3 , “Cum Q ME (%)” is set to zero to avoid divide by zero calculations.

Table 7.7 _TSMB.mif Attributes

Column	Description
ID	ID of the 1D node.
ME_Avg_Abs	The average of the absolute mass errors throughout the simulation. This is an excellent attribute for identifying 1D nodes that are regularly “bouncing”. By using the average of the absolute values, rather than the ME_Avg attribute below, any nodes that are bouncing either side of zero mass error will be identified. The best approach to identify these nodes is to using GIS thematic mapping tools to show, for example, large circles around nodes with high ME_Avg_Abs values down to small or no circle around nodes with zero ME_Avg_Abs values. The units are in m ³ /s.
ME_Peak_m3s	The peak (positive or negative) mass error in m ³ /s.
ME_Avg	The average mass error in m ³ /s.
Not_used	This attribute is not yet used.
t_____	The mass error in m ³ /s at time t_____ hours.

Table 7.8 _TSMB1d2d.mif Attributes

Column	Description
ID	ID of the 1D node. The object appears as a 1D point for the node collectively combined with the 2D cells connected to that 1D node via the 2D HX link.
ME_Avg_Abs	The average of the absolute mass errors throughout the simulation. This is an excellent attribute for identifying HX links that have poor mass error or are “bouncing”. By using the average of the absolute values, rather than the ME_Avg attribute below, any HX links that are bouncing either side of zero mass error will be identified. The best approach to identify these links is to using GIS thematic mapping tools to show, for example, dark red shading for links with high ME_Avg_Abs values down to light red shading for links with zero ME_Avg_Abs values. The units are in m ³ /s.
ME_Peak_m3s	The peak (positive or negative) mass error in m ³ /s.
ME_Avg	The average mass error in m ³ /s.
Not_used	This attribute is not yet used.
t_____	The mass error across the 2D HX link in m ³ /s at time t_____ hours.

7.6 Simulation Summary – Healthy Model Indicators

The 2010-10 release or later provides a summary of healthy model indicators at the end of the simulation on the display console, and at the end of the .tlf file. A selection of these indicators are also written to the “_ TUFLOW Simulations.log” files – see Section 7.1.3.

The indicators are discussed in Table 7.9 below.

Note that these indicators are only indicators and should be used in conjunction with good model review practices, not as proof that a simulation was or was not healthy.

Table 7.9 Simulation Summary Healthy Model Indicators

Column	Description
Total Negative Depths	The occurrence of negative depths at 1D nodes or 2D cell sides is an indication that the solution has not converged or has over-stepped at that point. WARNING 1991 for 1D nodes and WARNING 2991 at 2D cell mid-sides are issued each time a negative depth greater than -0.1m occurs. The location of these warnings can be viewed using the _messages layer. From a healthy model perspective, the occasional negative depth is not necessarily a concern, but repeat occurrences at the same location are an indication of poor topography or a difficult location in the model to solve. See Sections 8.1 and 9.4 for further discussion. Negative depths are often an indicator to the start of an instability. Negative Depth Approach = METHOD B introduced in the 2009-07 release, and the default setting for the 2010-10 release offers much improved performance in areas where negative depths repeatedly occur.
WARNINGS and CHECKS prior to and during simulation	Number of CHECKs and WARNINGS issued. At key stages of the modelling, review any CHECKs and WARNINGS, and if needed, resolve any issues, particularly for any WARNINGS. If a CHECK or WARNING is not in the _messages layer, this means that it could not be located geographically and only occurs in the .tlf file (search the .tlf file to review them).
Peak Flow In and Out (m ³ /s)	Review these numbers in the sense that they are in accordance with your expectations. Usually the Peak Flow In exceeds the Peak Flow Out for flood simulations due to the flood wave being attenuated as it travels through the model. Note that at boundaries where a circulation develops (which is an indication the boundary line is not well aligned and possibly should be changed), there will be flow in and out and these amounts will contribute to the Flow In and Out of the model as reported here and in the _MB.csv files.
Volume at Start and End (m ³)	The volume of water in the model at the start and end of the simulation. Review these numbers as to whether they make sense. A very large residual volume at the end of the simulation may indicate that the simulation wasn't run for long enough, for example, the flood may not have reached its peak.

Column	Description
Volume In and Out (m ³)	The total volume of water in and out of the model during the simulation. Once again, review these numbers as to whether they make sense. Usually the volume out is less than the volume in as the model has a residual amount of water left in it at the end of the simulation.
Volume Error (m ³) Final Cumulative ME%	Volume Error is the loss or gain in water over the course of the simulation. Volume error is equal to: $(\text{Total Volume In} - \text{Total Volume Out}) - (\text{Volume at End} - \text{Volume at Start})$ The Volume Error % value is the Volume Error divided by the Volume In + Out. The Final Cumulative Mass Error % is calculated throughout the simulation using a similar formula so should be similar to the Volume Error %. Ideally these values should be less than 1%, but 2 or 3% can be acceptable depending on the objectives of the modelling. Values exceeding 3% usually indicate there are problems within the model, and these problems are usually poor input data or schematisation - see Section 8 for further discussion.
Peak +ve and -ve dV (m ³)	dV is the change in volume over the whole model in one timestep, and the values shown here are the peak positive and negative dV values. Note: these values will be different to those shown on the display console or in the _MB.csv files if the Screen/Log Display Interval or Mass Balance Output Interval are not set to the computational timestep. The time in hours that the Peak dV values occurred is also shown.
Peak ddV (m ³) (% of Peak dV)	Peak ddV is the maximum (positive or negative) change in dV over one timestep, and the % value is the % of the maximum Peak dV value. A large ddV value or % indicates the model at some point may have been unsteady and not running particularly smooth. This may not be unusual in a model with complex hydraulics, however, it is another indicator of whether there may be somewhere in the model that needs reviewing.
Peak Cumulative ME	The peak CME% value that occurred. As discussed above, ideally this value should be less than 1%, but 2 or 3% can be acceptable depending on the objectives of the modelling. Values exceeding 3% usually indicate there are problems within the model, and these problems are usually poor input data or schematisation - see Section 8 for further discussion.
Values under “Qi+Qo > 5%”	The values under “Qi+Qo > 5%” are for the period of the simulation when the flow in and out exceeded 5% of the peak flow in and out, and are representative of the bulk of the simulation once the flood wave has begun flowing. These indicators are designed to exclude an initial period of any unsteadiness at the start of the simulation that may occur in some models. However, note that initial unsteadiness is often due to poor initial or incompatible boundary conditions, and should be checked/reviewed.

8 Quality Control

Section Contents

7.6	Simulation Summary – Healthy Model Indicators	7-42
8	QUALITY CONTROL	8-1
8.1	Unhealthy Models	8-2
8.1.1	Timestep	8-3
8.1.2	Tips for an Unhealthy 2D Domain	8-4
8.1.3	Tips for an Unhealthy 1D Domain	8-6
8.1.4	Tips for Unhealthy 1D/2D Links	8-9
8.2	Check List	8-11

8.1 Unhealthy Models

Proficient and effective 1D and 2D hydrodynamic modelling is a skill that takes years to develop to the point where the modeller can produce unproblematic, healthy models that consistently simulate floods and tides without drama! During the development of these skills, most modellers produce “unhealthy” models at some point, ie. models that are problematic in that they regularly go unstable, produce strange flow patterns, and so on. While in most cases the reasons for problems are within the quality of the input data, other reasons include poor model schematisation, and, of course, human error. With mentoring from experienced modellers, and/or following an iterative testing process, unhealthy models can be turned into healthy ones, and hydrodynamic modelling skill levels greatly enhanced. This section attempts to try and convey some of the ways of identifying problematic areas within an unhealthy model, and solutions to resolving the problem. Any constructive suggestions on how to expand this section further are welcome!

Unhealthy models usually exhibit one or more of the following characteristics:

- The model only remains stable if using a smaller than recommended timestep.
- Poor mass error ($> \pm 1$ to 3%) as indicated by the “CE” percentages displayed to the Console Window (see Section 7.1.1), and output to the various mass balance files as described in Section 7.5.
- “Unnatural” fluctuations of flow in/out and change in volume values (ie. the Qi, Qo and dV values displayed to the Console Window) discussed in Section 7.1.1.
- Locations in the model that repetitively have negative depth WARNINGS. These repeatedly appear as a message such as:
“WARNING 2991 - Negative U depth at [030;088], Time = 0:01:30, Depth = -0.4...”
The occurrence of the message several times at a location is usually not an issue (this means that the model experienced a short and slight numerical disturbance), however, if it repeatedly occurs for a period of time, it is good practice to resolve the problem as this numerical disturbance is likely to be causing mass errors, possibly forcing the use of a too small a timestep, and may initiate an instability in a future simulation.

If one or more of the above apply, the model needs to be reviewed and reasons identified. This can be a daunting and difficult task for inexperienced modellers, however, the guidelines in the sections below are hopefully of some assistance!

Tip: Take an iterative approach to solving problems as one problem often causes other problems. When searching through the _messages.mif file, resolve the problems in order of occurrence (ie. in the order the messages appear in a MapInfo Browser Window).

The most common cause for an unhealthy model is poor underlying topography. In the case of 2D domains, the quality of the DTM is often the problem, therefore, investing in time to create truly representative, well constructed, DTMs is highly recommended from both the modelling perspective and the quality of the inundation mapping.

For 1D domains, topographic inaccuracies in cross-section data and at structures is often a problem, although as 1D modelling is more of an art than 2D modelling (there is much more a modeller can

intentionally, or unintentionally, “fiddle” in a 1D model), the selection of cross-section locations and schematisation of the 1D domain is often an issue.

The underlying message is invest in good quality input data and experience! As the saying goes:

“garbage in, garbage out”

8.1.1 Timestep

There is a tendency for hydraulic modellers to “solve” an instability by simply reducing the timestep. Whilst this may “work”, it is usually not solving the fundamental cause of the model’s poor hydraulic performance or instability.

For the majority of flood models, the 2D [Timestep](#) in seconds should be somewhere between $\frac{1}{4}$ to $\frac{1}{2}$ of the 2D [Cell Size](#) in metres. For example, a 10m 2D grid should use a timestep of between 2 and 5 seconds. 2D domains with predominantly sub-critical flow usually can have timesteps larger than those for steeper models with significant areas of supercritical flow.

For coastal models, models with large cell sizes ($>50\text{m}$) or models with significant areas of deep water ($>5\text{m}$), the above rule-of-thumb may not apply with the timestep often being smaller. This is due to the Courant condition (see Section 3.6).

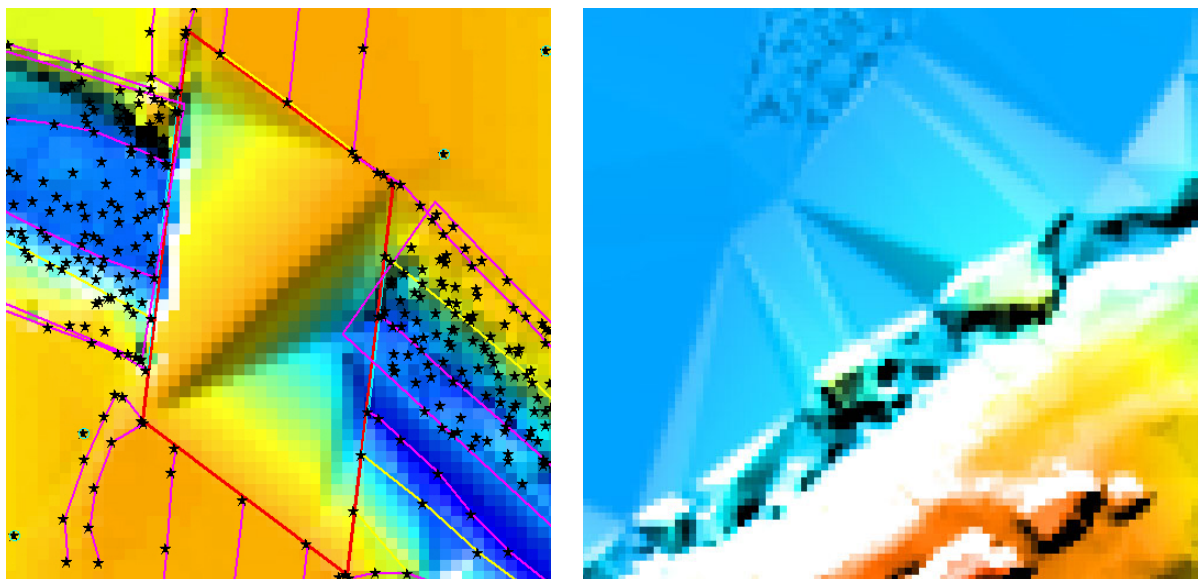
Using too small a timestep can tend to mask fundamental problems in the input data, and hide mistakes in the construction of the model.

Using a too large a timestep will cause mass errors. If the model runs stable without any negative depth warnings, yet the cumulative mass error is poor throughout the simulation, this is often an indication that the 1D and/or 2D timesteps are too large.

8.1.2 Tips for an Unhealthy 2D Domain

To identify problematic areas within a 2D domain, the most common approach is to run the model at a reasonable timestep and investigate where the WARNING 2991 messages occur. If mass error is an issue in the 2D domain, the _MB1.dat output (see [Map Output Data Types](#)) may be of use.

Of particular note is that models based on high quality DTMs are as-a-rule rarely problematic, however, if the DTM is rough or “bumpy” (as a lot of air-borne laser DTMs are), or has poor triangulation causing sharp ridges as illustrated in the images below, models based on these DTMs are much more likely to be problematic in/near these areas.



The following steps are useful to help resolve the problem.

- 1 Set the 2D [Timestep](#) to somewhere between $\frac{1}{4}$ to $\frac{1}{2}$ of the cell size (in metres) – see Section 8.1.1.
- 2 Run the model until it becomes unstable or has generated the WARNING 2991 messages.
- 3 In MapInfo, import the _messages.mif layer.
- 4 Open a MapInfo Browser Window of the _messages.mif layer and try to trace back from the “UNSTABLE 2999” messages to the initial “WARNING 2991” messages that were most likely the trigger for the instability.
- 5 In the Browser Window select a few of these WARNINGS.
- 6 Add the Selection to a Map Window, turn the display off for the _messages.mif layer if it is on, and zoom into the Selection (Ctrl+G is useful here) (note, that Selection is often renamed QueryX after adding to the Map Window).
- 7 Using the various _check.mif layers, carry out some fundamental checks:
 - (a) 2d_zpt_check: Check the Zpt values are as you would expect. If the Zpt values are particularly “bumpy”, try smoothing the ones at/near the WARNING 2991 messages. Where the Zpts change in elevation rapidly (eg. the outside bend of a river), deep ZU and ZV elevations can be problematic. If modifying the ZU and ZV values, tend to assign an elevation closer to the higher of the two ZC values either side of the ZU/ZV point. To modify Zpts don’t edit the original 2d_zpt layer, instead:

- (i) Select the Zpts you wish to modify from the 2d_zpt_check.mif layer;
 - (ii) File, Save As..., and choose Selection and provide a name such as 2d_zpt_for_stability.tab;
 - (iii) Open 2d_zpt_for_stability.tab (this will be the first file on the recent files list);
 - (iv) Make the layer editable, label the points with their elevations and using the Info tool, edit the elevation attribute (must be the fourth attribute) to smooth/change the topography.
 - (v) Save and export 2d_zpt_for_stability.tab as a .mif file.
 - (vi) Add a line such as [Read RowCol Zpts](#) == m\2d_zpt_for_stability.mid to the .tgc file after all the other Zpt commands, noticing that this is a Read RowCol, not a Read GIS command.
- (b) 2d_uvpt_check: check the Manning's n values are as expected, if not identify why.

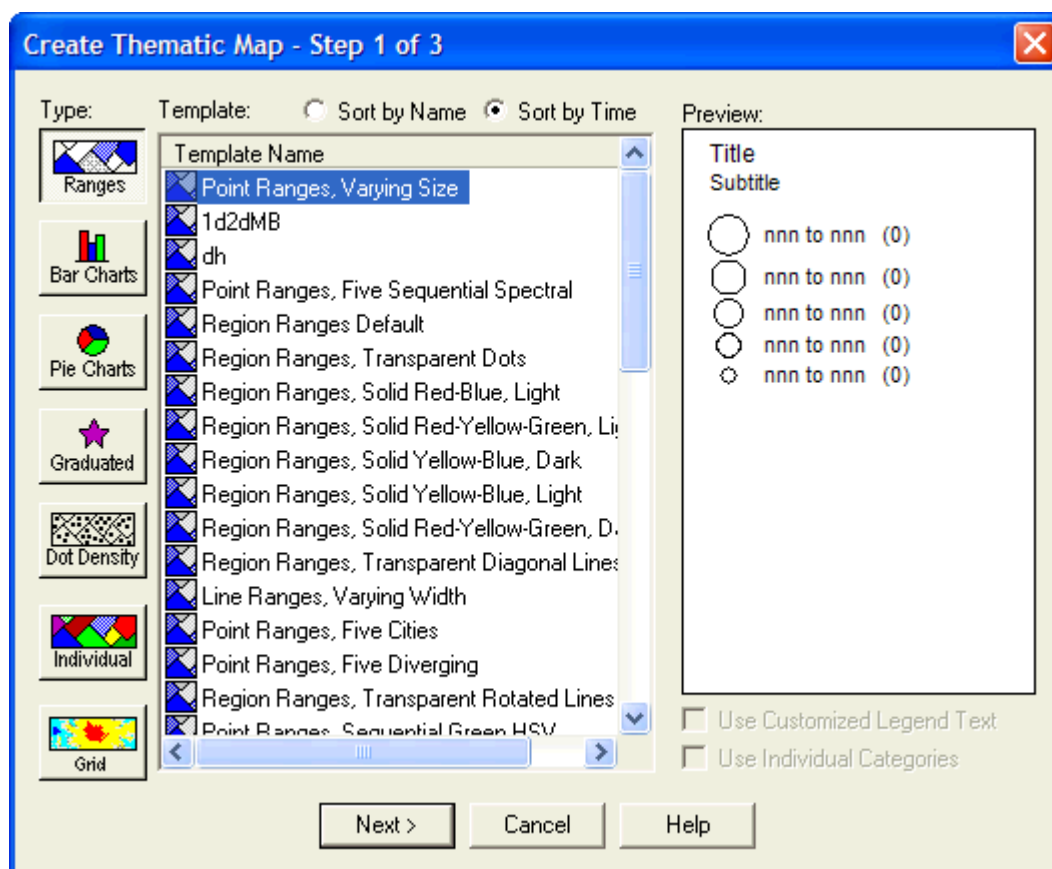
Other points to note for 2D domains are:

- 1 If the model becomes unstable quickly and the instability location is near a 2D water level boundary check that the initial water level setting is compatible with the starting water level of the boundary.
- 2 Direct rainfall modelling on high elevations may experience unacceptable mass errors (>100m) due to a floating point imprecision problem. These models need to use the double precision version of Build 2008-08-AA or later (see Section 5.2).

8.1.3 Tips for an Unhealthy 1D Domain

To identify problematic areas within a 1D domain, the most effective approach besides investigating where “WARNING - Negative depth at Node...” messages occur, is to thematically map the _TSMB.mif layer using the ME_Avg_Abs attribute. To do this, use the following steps as described for the MapInfo GIS. Note, the units of the mass error values for this layer are presently in m^3/s , not in percent.

- 1 In MapInfo, import the _TSMB.mif layer ([Mass Balance Output](#) must have been set to ON and the file is written to the .ecf [Output Folder](#)).
- 2 Go to Map, Create Thematic Map..., choose Varying Ranges and one of the point style options as in the dialogue below:



- 3 Click Next > and select the _TSMB layer and the ME_Avg_Abs attribute. The Ignore Zeros or Blanks can be ticked (zero mass error is not we are looking for!). Click Next >.
- 4 Use the various options in the last dialogue to change the styles, ranges, etc and click OK. It is worthwhile having your GIS specialist to explain all the different options for thematic mapping so as to fully utilise the power of this feature.
- 5 Figure 8-1 illustrates an example of thematically mapping problematic 1D nodes in a large 1D/2D model during the process of reviewing and fine-tuning the model during its development. The larger the circle the greater the mass error at those locations.

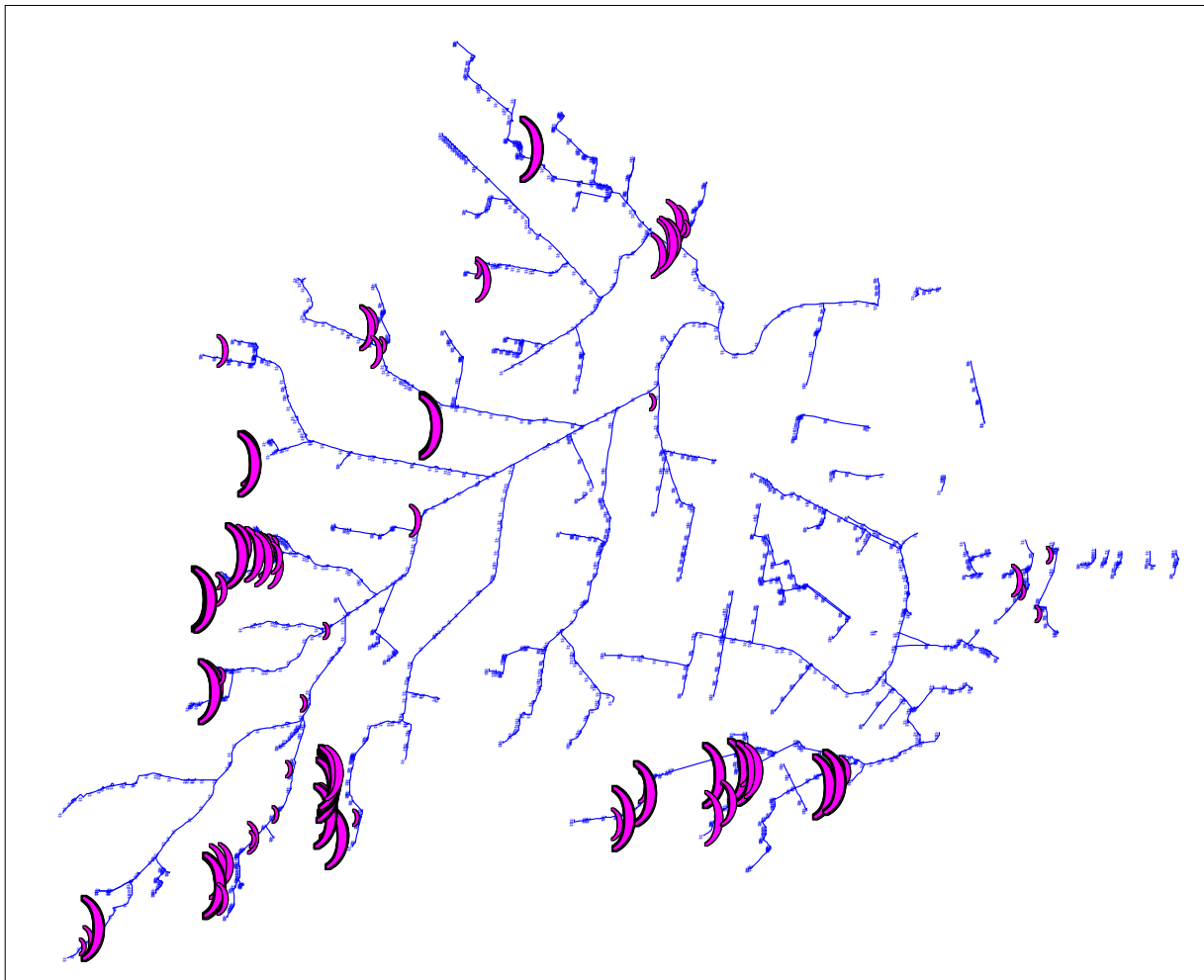


Figure 8-1 Thematic Map Example of _TSMB.mif Layer of a 1D Domain

Common options for helping resolve unstable or problematic 1D nodes and channels are:

- 1 If a 1D node or channel has become unstable, yet the water levels appear stable in the UNSTABLE messages, check the [Depth Limit Factor](#) setting. It may be that the water level is simply exceeding the maximum depth of the node/channel times the [Depth Limit Factor](#).
- 2 If a 1D node has repeated “WARNING - Negative depth at Node” messages, and/or has a mass error problem, check that the cross-section/structure dimensions of adjoining channels are correct and appropriate. Common causes are:
 - (a) A large change in cross-sectional area for successive channels. If the channel is natural, then it is likely that one or more of the cross-sections are not representative of the real situation.
 - (b) One or more incorrect upstream and downstream bed levels. Import the 1d_inverts_check.mif file, label the Invert attribute, and cross-check that the inverters are correct.

- (c) A very steep channel entering a gently sloping channel. If this is the real situation, then additional 1D channels giving a higher computational resolution maybe needed, or inserting a structure at the transition may help.
- (d) If there really is a sudden change in 1D flow area, then a more appropriate, and more stable, approach would be to insert a structure to model the sudden transition in velocities that occurs.
- (e) Trial using a smaller 1D [Timestep](#) to establish whether the problem is timestep related. If it is not timestep related, reducing the timestep should have little change in results. In some problematic models, 1D instabilities may actually magnify with a reducing timestep!
- (f) A common solution is to add additional storage to the node. This can be done by using the `ld_nwk Length_or_ANA` attribute (see Table 4.14), or using [Minimum NA](#), [Storage Above Structure Obvert](#) or [Minimum Channel Storage Length](#). Usually adding additional storage at problematic nodes does not adversely affect the results (unless there is a lot of problematic nodes!), however, this should be checked through sensitivity testing (this can be carried out by adding even more storage again and ascertaining the affect on the hydraulic results).

8.1.4 Tips for Unhealthy 1D/2D Links

In most 1D/2D models the 1D/2D links rarely have any mass error issues. However, as discovered in one large model, mixing up HX and SX connections along 1D/2D interfaces caused some serious mass error problems. In another model, specifying too small 1D NA tables for modelling 2D inflows by using a 1D node with a 1D QT boundary connected to a HX line also caused major mass errors (this approach has been replaced by the 2D QT boundary, which is not known to experience mass error issues). In these situations the 1D and 2D mass error reporting can appear satisfactory, however the overall model mass error is poor. In these cases, the _TSMB1d2d.mif layer is invaluable for identifying mass error issues across 2D HX lines. Note, the units of the mass error values for this layer are presently in m^3/s , not in percent. Figure 8-2 shows thematic mapping of 1D/2D HX links using a similar process to that described in the previous section. Those shaded red experienced a high mass error, orange and yellow less, and grey no mass error.

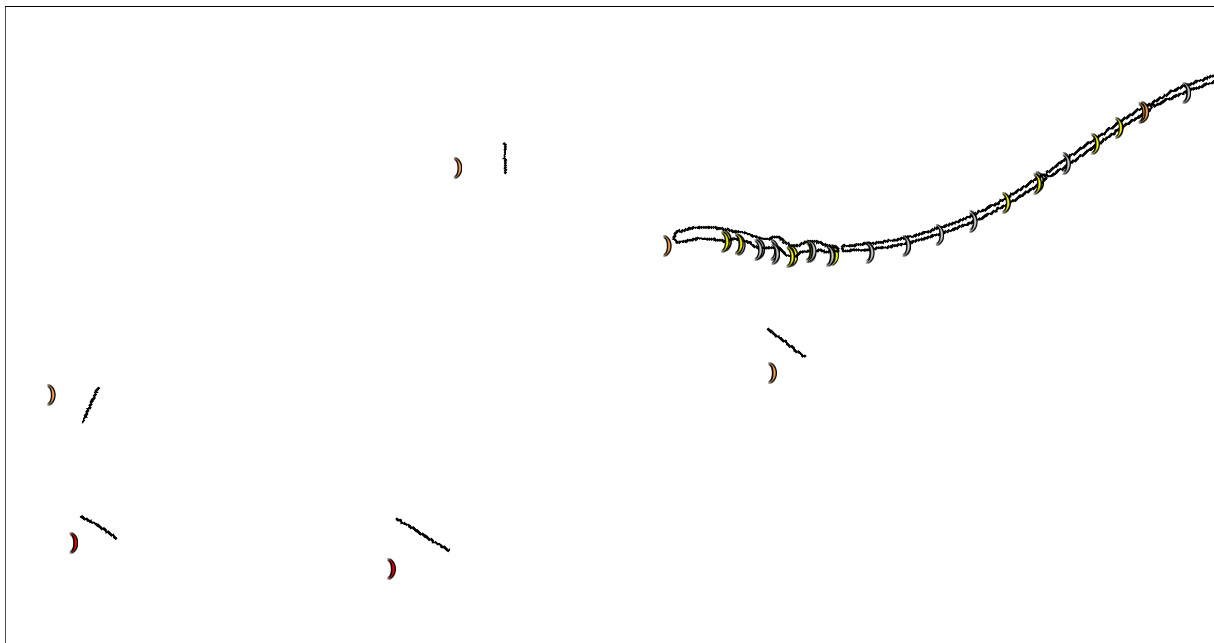


Figure 8-2 Thematic Map Example of _TSMB1d2d.mif Layer of 1D/2D HX Links

The known issues with 1D/2D links and options for helping resolve them are:

- 1 Using a mixture of connected HX and SX lines along a river bank can cause mass errors. This is not a recommended configuration, but can occur due to typing errors, etc.
- 2 If using the old approach of a 1D node with a QT boundary connected to a 2D HX line to create an inflow boundary for a 2D domain, if the NA table of the node is too small and the model remains stable, significant mass errors can occur. Either specify appropriate NA table storages or use the 2D QT boundary (the latter being preferred).

- 3 Unrealistic flow patterns occurring across the 2D HX lines, sometimes causing strange flow patterns and circulations in the 2D domain. This may be due to one of the following reasons:
- (a) Having insufficient spatial resolution in the 1D domain. If the interval between 1D nodes is too large, then the linearly interpolated 1D water level profile that is transferred from the 1D domain to along the 2D HX line does not accurately reflect the real situation. This forces water to enter or and/or leave the 2D domain in sometimes weird and wonderful ways! Check that there is a sufficiently fine resolution of 1D nodes along the river to be representative of the river's longitudinal water surface slope. For example, where the river's floodplain widens out and water spreads out over the floodplain, there are likely to be significant changes in longitudinal water surface gradient, hence requiring the 1D resolution to be finer in this section than that needed elsewhere.
 - (b) At a junction of three or more 1D channels, care should be exercised in how the levels from the side branches are transferred to the HX line(s). If the side branch has much higher water levels than the main branch, and a HX line segment is connected at one end to the side branch and the other end to the main branch a steep water level gradient may be applied along the HX line segment that is not particularly representative of the real situation.
 - (c) Similarly, at 1D structures where there is a significant drop in water level, the HX line may need to be stopped upstream and restarted downstream of the structure to prevent a steep gradient being applied across HX cells over the structure.

8.2 Check List

Table 8.1 presents a generalised list to help guide reviewers and modellers in carrying quality control checks on the modelling. This list is not exhaustive, and experienced modellers who know what to look for must at all times carry out the reviews.

Table 8.1 Quality Control Check List

Item	Description	Checked
Modelling Log	A modelling log is highly recommended and should be a requirement on all projects. The log may be in Excel, Word or other suitable software. A review of the modelling log is to be made by an experienced modeller. It should contain sufficient information to record model versions during development and calibration, along with observations from simulations. A model version naming and numbering system needs to be designed prior to the modelling. The version numbering system should be reflected in input data filenames to allow traceability and the ability to reproduce an old simulation if needed.	
File Naming, Structure and Management	A review of the data file management should check: - files are named using a logical and appropriate system that allows easy interpretation of file purpose and content; - a logical and appropriate system of folders is used that manages the files; - relative path names to be used for input files (eg. “..\model\geometry.tgc”) so that models are easily moved from one folder to another. - documentation of the above in, for example, the projects Quality Control Document and/or Modelling Log.	
2D Cell Size and 1D resolution	Check whether the 2D cell size is appropriate to reproduce the topography needed to satisfactorily meet the objectives of the study (see Section 3.1), and that the 1D spatial resolution is appropriate to reproduce the water longitudinal surface gradient.	
Topography	The topography review should focus on: - correct interrogation of DTM; - correct datum; - modifications to the base data (eg. breaklines) have been checked. Regarding the latter, this is effectively carried out by producing a _zpt GIS check file (see Table 7.1) using Write Check Files. The _zpt layer contains all modifications including any flow constriction adjustments. A DTM can be created from the Zpts using Vertical Mapper, or other 3D surface software, to aid in the review. Note: Reviewing the elevations in the .2dm file is not appropriate as only the ZH Zpt is represented in the .2dm file (the	

Item	Description	Checked
	ZH elevation is not used in the hydrodynamic calculations). 1D cross-section locations and conveyance should be reviewed. As a general rule, conveyance should steadily increase downstream. Sudden changes in conveyance need to be cross-checked (these are often easily identified by sudden changes in velocities of successive channels).	
Bed Resistance Values	Bed resistance values are to be reviewed by an experienced modeller. The review should focus on checking: - the Manning_n attribute values in the uvpt_check.mif layer created by Write Check Files; - the material values in the .2dm file; - the Material attribute values in the grd_check.mif layer created by Write Check Files; or - specifying weir output (see Map Output Data Types) if using the weir approach. The reviewer should be looking for: - relative consistency between different land-use (material) types; and - values are within accepted calibration values. GIS thematic mapping is an excellent way to visually and quickly review the variation in bed resistance and other parameter values.	
Calibration / Validation	Check that the model calibration or validation is satisfactory in regard to the study objectives. Identify any limitations or areas of potential uncertainty that should be noted when interpreting the study outcomes.	
Mass Conservation	In addition to the mass balance reporting (see Section 7.5), it is good practice to carry out independent mass checks. Standard practice is to place 2d_po flow lines (see Section 4.8.1) at a minimum of several locations through the model. They are typically aligned roughly perpendicular to the flow direction. The locations should include lines just inside each of the boundaries. Other suitable locations are upstream and downstream of key structures, through structures and areas of particular interest. The flows are graphed and conservation of mass checked (ie. the amount of water entering the model equals the amount leaving allowing for any retention of water in the model). Ensure that the flows from any 1D channels crossed by a 2d_po line are also included in the mass check, and that the 2d_po flow lines are digitised so that they cross the 1D channel where there is a change in colour of the linked 2D HX cells as shown in the 1d_to_2d_check.mif or _TSMB1d2d.mif layers. In dynamic simulations, an exact match between upstream and downstream will not occur due to retention of water, however, examination of the flow lines should reflect this phenomenon. For steady-state simulations, demonstration of reaching steady flow conditions is demonstrated when the flow entering the model equals the flow	

Item	Description	Checked
	leaving the model.	
Free-Overfall & Weir Flow	If Supercritical is set to OFF (not recommended), see this table in manuals prior to 2007 (downloadable from www.tuflow.com).	
Hydraulic Structures	Head losses through a structure need to be validated through: • Calibration to recorded information (if available). • Crosschecked using desktop calculations based on theory and/or standard publications (eg. Hydraulics of Bridge Waterways). • Cross-checking results using other hydraulic software. Simple checks can be made by calculating the number of dynamic head losses that occur and checking that this is in accordance with that expected (see Section 4.7.1). It is important to note that contraction and expansion losses associated with structures are modelled very differently in 1D and 2D schemes. 1D schemes rely on applying form loss coefficients, as they cannot simulate the horizontal or vertical changes in velocity direction and speed. 2D schemes model these horizontal changes and, therefore, do not require the introduction of form losses to the same extent as that required for 1D schemes. However, 2D schemes do not model losses in the vertical or fine-scale horizontal effects (such as around a bridge pier) and, therefore, may require the introduction of additional form losses. See Syme 2001b for further details.	
Eddy Viscosity	Check that the eddy viscosity formulation and coefficient is appropriate (see Section 3.7).	

9 Troubleshooting

Section Contents

9	TROUBLESHOOTING	9-1
9.1	General Comments	9-2
9.2	Suggestions and Recommendations	9-2
9.3	Large Models (Exceeding RAM)	9-3
9.4	Identifying the Start of an Instability	9-4
9.5	Why Do I Get Different Results?	9-5

9.1 General Comments

Problems in the input data are readily identified by using [Write Check Files](#) (.tcf file) and/or [Write Check Files](#) (.ecf file) to generate GIS check files. These files represent the final combination of the 2D and 1D data inputs and are excellent for identifying data input problems.

If the model becomes unstable, TUFLOW writes output data for the last timestep. The location of stability is easily found by viewing the results in SMS for the last timestep. Very large velocity vectors and/or excessively high or low water levels occur in the vicinity of the instability.

Always search the .elf and .tlf files for “UNSTABLE”, “WARNING”, “CHECK” and “ERROR” **and use the _messages.mif files** (see Section 7.2.1).

ERRORs stop the simulation, while WARNINGs and CHECKs do not. TUFLOW and ESTRY attempt to trap as many errors as possible before stopping to minimise the number of start-ups whilst setting up a model. It is possible that latter errors are caused by earlier errors, therefore, search through the .elf and .tlf files, or start at the beginning of the .mif attributes to find the first errors.

Also see Section 8.1 for further troubleshooting tips!

9.2 Suggestions and Recommendations

The following suggestions and recommendations are provided when troubleshooting a model. The list is not complete, but offers solutions to the more commonly found problems.

- 1 If the Console window does not appear at all, check virtual memory congestion (see Section 5.8).
- 2 If the Console window disappears for no apparent reason (this is now very rare!) first check the following:
 - (a) You have sufficient disk space on the drive you are writing your results to and where the .tcf or .ecf files are located (this is where the .tlf or .elf files are being written to).
 - (b) Your computer network is/was not down.
 - (c) Check the water level to be used for detecting instabilities. Prior to Build 2006-06-AA, if you have not allocated every Z point an elevation (the default Z elevation is 99999.) or if you have very high Z points in your geometry (relative to your water levels), this allows any instabilities to oscillate in a very large range. Consequently, the instability can become so extreme that floating point errors (ie. the computation is unresolvable) may occur before TUFLOW stops the simulation and declares it unstable. If this occurs, the Console window disappears. However, in most cases there should be some water level exceedance warnings at the end of the .tlf file and/or negative depth warnings in the _messages.mif file. To remedy the situation use [Instability Water Level](#) to set a realistic maximum water level. This same effect can occur in 1D domains if the maximum height in a node storage table or a channel cross-section is very high or the [Depth Limit Factor](#) is set unrealistically high.

If the problem persists, please contact WBM (support@tuflow.com). There is no charge for fixing bugs of this nature!

- 3 If TUFLOW or ESTRY indicate that GIS objects are not snapped, not connected, could not be found or are outside the model domain, check that the relevant GIS layers are in the correct projection and that the objects are snapped to each other. A GIS can handle layers of different projections, however, TUFLOW and ESTRY require that all layers be in the same projection. This projection must be a Cartesian projection (not lat/long) and is specified using [MI Projection](#) (.ecf file) or [MI Projection](#) (.tcf file). As of Build 2007-07-AA, the default setting is that all input layers are of the same projection otherwise an ERROR occurs (see [MI Projection Check](#)).
- 4 If you are having stability problems, check that the computational timestep is appropriate (see Section 3.6 and Section 8.1.1).
- 5 Discontinuous initial water levels, particularly at 2D/1D interfaces are a common source of instabilities. If the model is going unstable near a 2D/1D interface shortly after starting, check that the initial water levels in the 1D and 2D domains are similar.
- 6 It is not possible to specify a node as a flow boundary as well as being connected to a 2D SX boundary (which automatically applies a HX boundary to the node). This combination of flow boundary and water level (HX) boundary is incompatible. The result is that the HX boundary overrides the flow boundaries. An ERROR check for this occurrence was incorporated in Build 2003-03-AD.
- 7 Deep bends with “bumpy” topography may cause instabilities in 2D models. Smoothing the topography, rather than reducing the timestep is recommended.
- 8 Under-sized 1D node storage (NA tables) connected to 2D HX boundaries may cause instabilities near the 2D/1D interface. Over-sized storage attenuates the inflow hydrograph. As a rule-of-thumb, the node surface area should be similar to the width of the 2D/1D interface times 3 to 10 cell widths.
- 9 Irregular topography just inside a 2D boundary may cause instabilities. If problems occur, smooth the rough topography.
- 10 Use a timestep that divides neatly into 3600, ie. 1, 2, 3, 4, 5, 6, 7.5, 10, 12, 15, 20, 30, 45, 60, etc.

9.3 Large Models (Exceeding RAM)

As of Build 2007-07-AA (and 2006-06-BC), TUFLOW is compiled to handle very large models that require in excess of 1.5Gb of RAM. The computer may also need to be reconfigured to handle large demands on memory by setting the /3Gb switch in the boot.ini file. The following, taken from an entry on SearchWinComputing.com, explains the process for reconfiguring your computer to handle memory hungry processes.

“Recent versions of Windows support a switch option in BOOT.INI called the /3GB switch, which adjusts the way memory is allocated between the user and the operating system. By default, the split is fifty-fifty -- up to two gigabytes for the user and two for the operating system. The /3GB switch option sets those limits at three gigs maximum for the user and one gig for the operating system.

As the name implies, /3GB was originally meant to be used in systems that have 3 GB or more of RAM -- something that is no longer quite as rare as it used to be! However, even if you don't have 3 GB or more of memory, you can still use the /3GB switch

successfully if you are running memory-hungry desktop applications. Photoshop, for instance, is infamous for staking out as much RAM as it can. With /3GB enabled, there's that much more memory that the application can use.

I've written previously about using the /3GB switch in both Windows 2000 Server and Windows 2003 Server environments to bolster performance of memory-intensive services such as Exchange Server and SQL Server. But, in a desktop environment, you should consider any use of /3GB to be provisional until you determine that the application set you're using plays nicely with it. (My usual rule of thumb for seeing if something holds up under stress is to try it for two weeks under varied conditions; if it doesn't crash, it's probably okay.)

To add the /3GB switch as a separate boot option:

1. In My Computer | Properties | Advanced | Startup and Recovery, click Settings.
2. Click the Edit button under System Startup to edit the BOOT.INI file.
3. Find the current boot entry, which typically looks like this:

```
multi(0)disk(0)rdisk(0)partition(1)\WINDOWS="Microsoft Windows XP Professional" /fastdetect
```

4. Copy this line and paste it at the end of the BOOT.INI file, and change it along these lines:

```
multi(0)disk(0)rdisk(0)partition(1)\WINDOWS="Microsoft Windows XP Professional 3GB Switch" /fastdetect /3GB
```

This lets you choose between a conventional boot entry and the /3GB boot entry. If something goes wrong when you use /3GB, you can always boot back into the original configuration.

5. Save and close BOOT.INI.
6. Click OK to close the Startup and Recovery panel.
7. Click Settings again, and in the "Default operating systems:" dropdown, choose the newly created boot entry with the /3GB switch.
8. Click OK to close everything. Reboot into the new profile."

9.4 Identifying the Start of an Instability

Instabilities usually start with a one or a few computational points "bouncing" as a result of poor convergence of the mathematical equations being solved. To help identify the start of an instability, negative depth warnings are issued if the depth in a 2D cell or a 1D node becomes falls below -0.1m. Negative depth warnings are usually a pre-cursor to an instability. It is not uncommon, particularly in areas of rapid wetting and drying for negative depths to occur before the computational point is made dry (inactive). Hence a buffer of -0.1m is used before reporting a WARNING.

The WARNINGS are sent to the _messages.mif file. Import these into the GIS and they point directly at the location of the negative depth. If the number of these warnings are substantial (eg. if a model remains stable but with minor instabilities), select some of the first negative depth warnings in the attribute data (Browser Window in MapInfo) and display just those. The warnings are in order of occurrence. By tracing through the negative depth warnings in the vicinity of the instability, the trigger point of the instability can often be located.

Also see the discussions on rectifying problematic models in Section 8.1, [Depth Limit Factor](#) (1D domains) and [Instability Water Level](#) (2D domains).

9.5 Why Do I Get Different Results?

Hopefully, this never occurs! However, like most (all?) computational software, providing backward compatibility (ie. getting the same results with different versions of the software) is at times very difficult. This is mainly associated with complications in the code when making improvements and, of course, the occasional bug. They can even be due to vagaries of the source code compiler (as has been found with immense frustration on several occasions!).

During the life of TUFLOW since 1989, every effort has been made to provide full backward compatibility. Models developed back then can still be simulated to achieve the same results. Old file formats are still recognised or can be translated into more recent formats. There have, however, been numerous enhancements, improvements and the inevitable bug fixes. Models using some of these new features during the development stages may produce slightly different results using latter versions of the software.

One known reason for very slight differences between identical models is if one model has 2D FCs that are all inactive (not wet) and the other model has no 2D FCs anywhere, very slight differences (< 1mm) may occur as the computational engine uses slightly different lines of computational code if there are 2D FCs anywhere in the model, than if there isn't.

Since March 2001, a unique software build identifier has been used to track and manage new versions of the software. The build number is in the format of year-month-xx where xx is two letters starting at AA then AB, AC, etc for each new build for that month. The build number is written to the first line in the .elf and .tlf log files so that it is clear what version of the software was used to simulate the model.

Table 9.1 presents known possible changes between different TUFLOW builds that may change model results. In nearly all cases a backward compatibility switch is provided.

Table 9.1 Possible Reasons for Different Results between TUFLOW Builds

Build	Description	Consequences/Workaround
2010-10-XX Builds		
2010-10-AA	New Intel Fortran Compiler version produces slightly different results (usually fractions of a mm).	No workaround.
2010-10-AA	w32 and w64 versions will give slightly different results for the same simulation.	No workaround. Use the same platform (w32 or w64) for all simulations. Use Model Platform to force which platform should be used.
2010-10-AA	New default settings – see Defaults == PRE 2010-10 for a list of the commands that have changed in their default setting.	Set Defaults == PRE 2010-10 if similar results are required to the 2008-08 or 2009-07 releases.
2010-10-AA	Generation of TINs for polygons in Read GIS Shape layers is more robust and uses an improved approach. In rare cases, the TIN would fail and TUFLOW would abort the startup.	No workaround.
2008-08-XX and 2009-07-XX Builds		
2008-08-AC	The default setting for Shallow Depth Stability Factor has changed.	Set Shallow Depth Stability Factor == 3 for models without direct rainfall to achieve the same results as Builds 2008-08-AA and 2008-08-AB. See Shallow Depth Stability Factor for more information.
2008-08-AA	Uses a new set of defaults for a number of commands (see Section 10 and Defaults).	The new defaults produce slightly different results, and very slight differences also occur between the three versions offered (see Section 5.2 and Table 5.1). For established models run using the 2007-07-XX builds, use Defaults == PRE 2008-08 to use the default settings used by the 2007-07-XX builds. Testing of a range of models has shown zero change in results if Defaults == PRE 2008-08 switch is set, and the Compaq Fortran compiled version (cSP) is used. Each of the new default settings and their effects are discussed in the rows below.

Build	Description	Consequences/Workaround
2008-08-AA	The method for interpolating n values where the 2D Manning's n varies with depth has been enhanced from a linear interpolation of the M (1/n) value to a spline interpolation of the n value. See Bed Resistance Depth Interpolation .	Generally has little effect other than when the flow is predominantly in the depth range that the n value is varying. The new approach offers a smoother transition in n values from one depth to the other.
2008-08-AA	The default viscosity coefficient is now a combination of a 0.2 Smagorinsky and 0.1 constant coefficient, and there are some enhancements to the application of the viscosity term. See Section 3.7 and Viscosity Coefficient .	For the majority of models slight effect. For fine grid models (<2m cell size) with low bed resistance and significant variations in velocity vectors the effect is more pronounced but is still slight.
2008-08-AA	Inertia and viscosity terms are now not transferred across dry cell sides when constructing the coefficients for the solution arrays. This was having the effect of generating a circulation on the other side of the wall (albeit a very weak one), which of course shouldn't happen!	Generally little effect, but can have some minor influence for urban models where buildings and fences are modelled as solid thin Z lines.
2008-08-AA	1D weir flow has been improved as the water level difference across the weir approaches zero. The new method is more stable. See Weir Flow .	Very little difference other than improved stability.
2008-08-AA	Incorporates minor improvements for transitioning between Regimes A and B, and between inlet and outlet controlled regimes, for circular culverts.	Very little difference other than improved stability.
2008-08-AA	The new automatic selection of cells for 2D SX connections using the 1d_nwk Conn_2D attribute may choose more than one 2D cell. See Section 4.5.1.3.	Very little difference other than improved stability at the pit 2D connections.

Build	Description	Consequences/Workaround
2007-07-XX Builds		
2007-07-AA	Uses a new set of defaults for a number of commands (see 13Appendix F and Defaults).	The new defaults may produce slightly different results. For established models run using the 2006-06-XX builds, use Defaults == PRE 2007-07-AA to use the default settings used by the 2006-06-XX builds. Testing of a range of models has shown zero change in results if Defaults == PRE 2007-07-AA switch is set. Each of the new default settings and their affects are discussed in the rows below.
2007-07-AA	Change Zero Material Values to One == OFF (previously ON)	Will not cause different results if a Set Mat == 1 is specified before other material settings in the .tgc file, or if every cell has been assigned a material value.
2007-07-AA	Inside Region == Method B (previously Method A)	Testing thus far has not shown any difference between the two methods (other than the substantial gains in processing time of polygons).
2007-07-AA	Line Cell Selection == Method D (previously Method C)	May change results slightly, but improved stability and a smoother water levels along HX lines result.
2007-07-AA	VG Z Adjustment == MAX ZC (previously ZC)	May change results slightly, but stability should be significantly enhanced in some situations.
2007-07-AA	Bed Resistance Cell Sides == INTERROGATE (previously AVERAGE M)	Will influence results, usually slightly, but more pronounced where there are sudden changes in Manning's n values such as in the urban environment.
2007-07-AA	Culvert Flow == Method D Culvert Critical H/D == OFF (previously Method C and Culvert Critical H/D == 1.5)	The most significant influences are the selection of upstream or downstream controlled regimes depending on the H/D ratio, and the bug fix relating to Regime E if Structure Losses == ADJUST (see Section 4.7.4.3). Offers improved stability, better convergence for Regime C and smoother transitioning between some regimes.

Build	Description	Consequences/Workaround
2007-07-AA	Changed the setting of the default width (if $eN1 < 0.001$) of automatic weirs over R and C channels (ie. RW and CW) to be the diameter/width multiplied by the number of culverts (previously, the width was not multiplied by the number of culverts).	For backward compatibility, original weir width can be set by manually setting the eN1 attribute to the Diameter_or_Width attribute value of the culvert.
2007-07-AA	Bug fix that when using a restart file TUFLOW occasionally set the 2D FC bridge deck additional loss value incorrectly.	No backward compatible workaround provided.
2007-07-AA	Bug fix that incorrectly set the water levels on dried VG cells (only applies to simulations with source inflows, eg. SA or RF, somewhere within in the model).	May cause slight changes in results. Backward compatibility provided if Defaults == PRE 2007-07-AA is set (noting that setting this command reinstates the bug). This bug also causes the mass error calculations to falsely give a mass error that is not occurring.
2007-07-AA	Fixed bug that did not correctly apply the reduction in conveyance for a FC BD (bridge deck) of FD (floating deck) cell using the 2d_fc Mannings_n attribute.	Backward compatibility applied if Defaults == PRE 2007-07-AA is set, however, note that this reinstates the bug and the resistance to flow at FC BD and FD cells may need to be reviewed. Indications are that only minor changes in results occur. The flow area under 2D FC BD and FD cells is correctly calculated.
2006-06-XX Builds		
2006-06-AA	Uses a new set of defaults for a number of commands.	The new defaults will produce different results. For established models run using the 2005-05-XX builds, use Defaults == PRE 2006-06-AA to use the previous default settings. Each of the new default settings and their affects are discussed in the rows below.
2006-06-AA	Cell Wet/Dry Depth == 0.002 (previously 0.05) Cell Side Wet/Dry Depth == 0.001 (previously 0.03)	The most pronounced effect of the shallower wet/dry depths is likely to occur in areas that are still filling at the flood peak, such as behind a levee that is only just overtopped. The shallower wet/dry depths provides a greater flow depth for a longer period over the levee.

Build	Description	Consequences/Workaround
2006-06-AA	Adjust Head at Estry Interface == OFF (previously ON)	Usually does not have a major influence on results except where very high velocities occur.
2006-06-AA	Boundary Cell Selection == Method C (previously Method A) Line Cell Selection == Method C (previously Method A)	May select slightly different cells along boundary/link lines. This may cause a difference where the line is along the top of levee, possibly creating a “hole” in embankment.
2006-06-AA	Viscosity Formulation == Smagorinsky (previously Constant) Viscosity Coefficient == 0.2 (previously 1.0)	Can have a significant effect where the viscosity term is influential. This occurs where the friction term is less dominant (ie. low Manning’s n and/or deeper water such as the lower, tidal, reaches of rivers).
2006-06-AA	Structure Losses == ADJUST (previously FIX)	Can have a significant affect in the vicinity of structures within a 1D network and for culvert networks. Does not affect 1D structures linked to a 2D domain or at the structure ends not connected to another 1D channel.
2006-06-AA	Storage Above Structure Obvert (%) == 5 (previously Storage Above Structure Obvert == CHANNEL WIDTH)	Usually negligible effect unless the model storage is predominantly within 1D closed sections (ie. B, C and R channels). The 1D domain is likely to be more sensitive to instabilities due to the much smaller storage above the top of the closed sections, therefore, a smaller 1D timestep may be required and/or the Storage Above Structure Obvert (%) increased.
2006-06-AA	Depth Limit Factor == 10 (previously 1)	No effect as previously the model would have become “unstable” as the trigger for an instability was the top of the channel/node.
2006-06-AA	Culvert Flow == Method C (previously Method B)	Usually only minor effects plus improved stability.
2006-06-AA	Culvert Add Dynamic Head == ON (previously OFF)	Minor influence.
2006-06-AA	Bridge Flow == Method B (previously Method A)	Negligible influence plus improved stability. However, note the different treatment of energy losses once the bridge deck obvert/soffit is submerged if a BG or LC table is specified.

Build	Description	Consequences/Workaround
2006-06-AA	WLL Approach == Method B (previously Method A)	Only affects the presentation of results. Note, that Method A is no longer recommended or supported.
2006-06-AA	Apply All Inverts == ON (previously OFF)	Does not affect hydraulic calculations, however, if a Blank, B or W channel is now lowered/raised because the inverts are now used, this will affect results/stability - see note at end of Apply All Inverts).
2006-06-AA	Conveyance Calculation == ALL PARALLEL (previously CHANGE IN RESISTANCE)	Will affect results as ALL PARALLEL can be around 10% more “slippery” than CHANGE IN RESISTANCE. For calibrated or established models developed using build prior to Build 2006-06-AA , recommend setting to CHANGE IN RESISTANCE
2006-06-AA	Flow Calculation == Method B (previously Method A)	Negligible effect.
2005-05-XX Builds		
2005-05-AN	Code restructuring, implementation of different timesteps for different domains and the separation within the engine of 2D SA inflows and 1D flows via 2D SX connections.	These changes have appeared to cause very slight (~1mm) changes to models that have been tested, with the occasional larger change around the wet/dry perimeter.
2005-05-AN	A number of attributes in the 1d_nwk layer for point (node/pit) objects are now used, where as previously they were not.	Provided the values in these attributes were empty or zero, backward compatibility should be preserved.
Builds prior to 2005-05-XX Builds		
2004-06-AC	New ecf command “ S Channel Approach == [PRE 2004-06-AA]” to use the S channel approach prior to 2004-06-AA for backward compatibility.	This only affects S channels when the downstream end is dry and also applies the Froude Check value more correctly as being squared (ie. a number of 0.64 entered for Froude Check prior to 2004-06-AA would now be entered as 0.8).
2004-05-AF	First Sweep Direction default set to POSITIVE. For backward compatibility, set to AUTOMATIC.	Testing as shown that this change should have virtually no effect, with changes in the order of less than one mm.

Build	Description	Consequences/Workaround
2004-02-AA	Set the alignment of oblique boundaries as nearest to line as the default. Need to specify “Oblique Boundary Alignment == CENTRE TO CENTRE” for backward compatibility.	May change which cells become boundary cells along a 2d_bc line.
2004-01-AE	An ERROR is now given if the ZC on a 2D HX cell lies below the bed of the 1D domain. HX ZC Check == OFF can be used to suppress this check.	This check prevents the possibility of a surge of water entering the 2D domain once the 1D domain becomes wet. It is recommended that any 2D HX cells needing adjustment are raised.
2003-07-AC	Fixed bug that assigned an incorrect value to top of section flow width for channel sections read from fixed field flow width (CS) tables.	Only effects fixed field CS table input where the water level exceeds the top of the channel. Only applies from Build 2002-07-AB.
2003-06-AD	Now use the Divergence attribute in 1d_nwk layer to represent %blockage on culverts. For R culverts, the culvert width is reduced by the %blockage, while for circular culverts the pipe diameter is reduced by the square root of the %blockage. Divergence field is now not used.	Any channel that had a non-zero divergence attribute will now have slightly different results. Divergence is not known to be used. However, a backward compatible switch will be provided upon request.
2003-06-AB	Fixed bug that failed to allocate 2D SX cells from 2d_bc SX polylines. Only effects one cell along each line segment after the first line segment. Does not affect single segment polylines or points.	Slight change in results may occur at relevant 2d_bc SX polylines.
2003-06-AA 2003-06-AD	Fixed bug that incorrectly calculated the flow interchange into a 1D node across a 2D HX line where a flow constriction (FC) cell shared a common boundary with a 2D HX cell. Not fully corrected until 2003-06-AD.	Flow balance between 1D and 2D domains significantly affected, particularly if the FC cells are submerged.
2003-06-AA	Fixed long-standing bug (at least going back to code from pre-1990) for 1D VG channels.	Possibly incorrect variable geometry changes to the 1D channel.
2003-06-AA	2d_po time-series output by default synchronised with 1D domain time-series output. No effect on results.	May change when 1D domain time-series output occurs. See Output Times Same as 2D for backward compatibility.

Build	Description	Consequences/Workaround
2003-05-AF	Fixed rare bug that incorrectly selected 2D cells or Zpts within a region (polygon) object when the first vertex in the region is repeated elsewhere in the region (except the last vertex). This may occur as a result of “sloppy” digitising, or, for example, a polygon that has a “figure of eight” shape.	Different 2D cells or Zpts adjusted by the Read GIS command.
2003-05-AD	Parallel channel calculations for a composite cross-section from a 1d_tab XZ table link now split up based on changes in material values when using materials for a 1d_tab XZ cross-section. Previously, the split was based on a change in Manning’s n value.	Only effects the calculation of a channel’s hydraulic properties if two adjacent materials have the same Manning’s n value.
2003-04-AA	Now sets initial water levels at 1D HX boundaries (which are normally automatically generated from 2D SX boundaries) to initial water levels at the 2D SX cells.	If initial water levels between 1D and 2D domains are different, slightly different results at model startup may occur.
2003-01-AE/AF	Incorporated another upstream friction controlled flow check for 1D “S” channels and 2D domains that reduces the Froude No check by the ratio of the upstream depth to the downstream depth when the downstream depth is greater than the upstream depth. This prevents overestimation of flows occurring when a steeply sloping flowpath is shallow at its u/s end and very deep at the d/s end. Use Froude Depth Adjustment (.ecf file) and/or Froude Depth Adjustment (.tcf file) in 2003-01-AF for backward compatibility.	May cause slightly different results. Froude Depth Adjustment (.ecf file) and/or Froude Depth Adjustment (.tcf file) provide backward compatibility.
2002-12-AA	Improvement to culverts Method B to trap vibrating or unstable culverts.	May cause different results at culverts previously unstable.
2002-11-AD	Improvements to automatic switching to 2D upstream controlled flow.	Will cause different results in models previously experiencing upstream controlled friction flow. See Supercritical .
2002-10-AL 2002-10-AM	Improvements to culverts Method B for regimes C and D for steep culverts with high entrance velocities.	May cause slightly different results, although previously model was likely to be unstable.

Build	Description	Consequences/Workaround
2002-10-AJ	Improved reading of .csv files. Fixes a bug where if text starting with a “F”, “f”, “T” or a “t” was found when searching for the first boundary time or value, it was interpreted as a 0 or 1, rather than ignoring it and proceeding to next line. Now requires that column names specified in 1d_bc, 2d_bc and 1d_tab layers are fully specified – previously would allow a substring. Now correctly allows spaces in a column heading.	May read boundary time-series data differently.
2002-10-AH	Adjusts velocity vector output in SMS where upstream controlled weir flow occurs to that corresponding to critical depth.	May cause slightly higher velocities to be observed at 2D weirs. Does not affect any other results.
2002-10-AE	Fixed several bugs relating to relative resistance for XZ data via a 1d_tab layer.	Unlikely to be a problem as the relative resistance did not work correctly, producing significantly incorrect hydraulic properties for cross-sections.
2002-08-AG	Included check that compares the ZC elevation with the ESTRY node bed elevation at SX 2d_bc objects. The ZC elevation should be below the lowest channel connected to the node.	This check may stop old models from running. The ZC point(s) should be lowered or the “Z” flag used to automatically lower the ZC point (see SX objects in Table 4.33). Alternatively, SX ZC Check in the .tcf file can be used to provide backward compatibility.
2002-08-AD	Changed default Culvert Flow method from Method A to Method B.	For backward compatibility, set the Culvert Flow method to Method A.
2002-08-AB	Fixed rare bug that effected the determination of 2D cells falling inside a polygon (eg. a material polygon).	May affect Read GIS commands using polygons.
2002-06-AE	Corrected an adjustment to ESTRY water levels at nodes to the highest elevation in the NA table if the water level exceeds this elevation. In unusual cases, if it is not detected as an instability, it causes mass balance problems	Not believed to be an issue as normally the model is detected as being unstable. This problem arose when water levels were allowed to rise above NA and CS tables using Depth Limit Factor .
2002-06-AB	Changed how 1D storage is allocated to nodes using the channel widths. Storage above bridge and culvert obverts is now NOT included.	May affect automatic node storage tables.

Build	Description	Consequences/Workaround
2002-05-AA	Improved upstream controlled weir flow along a HX line when flow is from the 1D domain to the 2D domain. Water level was previously being converted to an upstream energy level (as done in the 2D domain) using an extrapolated velocity based on the downstream velocity (this caused instabilities and is not really valid). Water level at HX cells now taken as a energy level.	Significantly improved stability and performance at HX boundaries if upstream controlled weir flow is occurring into the 2D domain. May very slightly change results in previous models around HX boundaries only if upstream controlled weir flow occurs into the 2D domain. Unlikely to be the case in many models as it is only with the advent of a 1D domain being carved through a 2D domain that this feature has been needed (eg. flow across a levee from 1D creek to 2D floodplain), and only a few models of this type had been developed at the time of this build.
2002-03-AC	Fixed bug that adjusted unspecified Zpts when using the ADD or MAX option when reading Zpts.	May cause changes to bed levels in areas where no Zpts specified. Unlikely to be a problem as unrealistic elevations would have occurred prior to this build.
2001-09-AM 2001-09-AN	Improvements related to suppressing inertia term at free-overfalling points when water level on downstream side falls below level of cell side.	Testing shows little effect on results, however, new feature offers marginally better stability. Also more technically correct.
2001-09-AK	Improved performance of Read GIS Z Line Thick in picking up ZC and ZH points.	May slightly change the number of ZC values picked up. Unlikely to change results to any significance.
2001-08-AF	Fixed bug where a gully line (see Read GIS Z Line Gully) segment is exactly vertical. The Zpts were not adjusted.	Prior to this build, Zpts were not adjusted along <u>exactly</u> vertical gully line segments.
2001-08-AD	Changed default Global FC Ch Factor to 0.8 (previously 0.6).	Effects results very slightly if upstream controlled pressure flow occurs at a flow constriction. Set Global FC Ch Factor to 0.6 for backward compatibility.
2001-08-AC	Number of minor improvements to upstream controlled weir flow across cell sides. Also, general weir factor changed from 1.2 to 1.0.	Test models give same results under steady-state conditions. Better transitioning between upstream and downstream regimes. May cause very slight changes to results.

Build	Description	Consequences/Workaround
2001-05-AA	Fixed bug that incorrectly applied the Manning's n value of a FC. The correct Manning's n value was applied, but in some cases, offset spatially.	Has a very slight localised influence at any FCs that specified a Manning's n adjustment. Checks on models showed slight localised changes in levels of up to a few cm.
2001-05-AA	Finalised flow calculations when a FC is submerged on the upstream side and unsubmerged on the downstream side.	May very slightly effect results local to a FC during submergence.
2001-04-AF	Fixed bug that accidentally zeroed initial velocities from a restart file.	May effect models that used restart files prior to this build. Any effects would only be discernable during the start of the simulation.
2001-04-AC 2001-04-AF 2001-12-AA	Bugs and improvements to calculation of flow across a "Q" PO line.	No effect on computational results. May only effect accuracy of post-processed flow calculations.
2001-04-AA	Bug found in relation to the Smagorinsky Coefficient. The Smagorinsky formula was not applied at all steps in the solution, with the constant viscosity being applied instead.	Smagorinsky coefficient rarely used in models up until this point. No known significant effects on models, although would explain the slightly unusual result during TUFLOW testing in Syme 1998.

10 New Features and Changes in 2010-10 and 2009-07

New software features and changes incorporated for the 2010-10 and 2009-07 releases are presented in Table 10.1, and a summary of other changes and new features are discussed below. For new features/changes of previous builds and releases, see Appendix F.

NOTE: All of the Read MI and Read MID commands in the manual have been replaced by Read GIS and Read RowCol. Read MI and Read MID remain valid commands and will do so indefinitely, so they can continue to be used. You can even use Read MI to read a .shp file if you wish! Read GIS is identical to Read MI, and Read RowCol is identical to Read MID. The Read GIS and Read RowCol syntax was adopted with the recognition of both .mif and .shp formats, and to make the intent of these commands clearer.

When using new features, thorough checks of the model (using the _check.mif layers) and of the results should be made. Whilst the features have been tested, there is no guarantee that there are no software bugs! Should a problem occur, please do not hesitate to contact support@tufLOW.com.

Of particular note for users is that documentation shaded with a pale blue represent the new or modified text and sections of the manual (the pale green shading indicates the changes from 2007-07 to 2008-08). We can't guarantee that all of the key changes and additions are shaded, however, hopefully we managed to identify most of them!

Other points to note are:

- 1 For those that are not aware, an internet based TUFLOW Forum has been set up at www.tufLOW.com/forum. The forum contains a range of topics and postings sourced from support emails over recent years, and offers the opportunity for modellers to learn from others' experiences and to post their own questions and replies. The other main benefit is that announcements of new software and other uploads to www.tufLOW.com are emailed to members. If you haven't already done so, please register for the forum and utilise the knowledge base that it offers.
- 2 A documented tutorial model is available for download from [www.tufLOW.com/Downloads Tutorial Model.htm](http://www.tufLOW.com/Downloads/Tutorial%20Model.htm). Of note is that this model can be modified and simulated using TUFLOW without the need of a TUFLOW licence. It is of particular use to those wishing to learn how to use TUFLOW, or to carry out training. The tutorial model is presently designed for the MapInfo/Vertical Mapper TUFLOW model development environment, but extensions to the SMS TUFLOW and XP-SWMM2D Graphical User Interfaces, and a similar version for ArcGIS are planned. A new version of the Tutorial Model has commenced and will be available in 2011. This version will incorporate the more popular features since 2007 (when the current Tutorial Model was developed), and five or six new modules for advanced users.
- 3 The very useful MapInfo based miTools tools is now available for purchase. This is a must for anyone using MapInfo to develop TUFLOW models as it significantly improves your efficiency of operation. Visit www.tufLOW.com for more information or contact sales@tufLOW.com.

Table 10.1 New Features and Changes for the 2010-10 and 2009-07 Releases

Links	Description
New Features, Bug Fixes and Enhancements in Build 2010-10-AB	
Section 4.5.1	Any number of pits can now be snapped or connected to a digitised node or manhole. Prior to this build, if a digitised pit, manhole or node was snapped to another digitised pit, manhole or node (ie. two or more digitised point objects at the same location), TUFLOW would not treat these as one node.
Pit Search Distance Section 4.5.1	New .ecf Pit Search Distance command that sets the search distance for automatically connecting pits (eg. gully traps) to the nearest 1D node. This means that pits do not need to be snapped to a channel end, which can be very useful where the location of the pits is known, but the connecting pipe to the main trunk pipes is not known.
Pit Default 2D Connection Section 4.5.1.3	New .ecf Pit Default 2D Connection command sets the default 1d_nwk Conn_2D value. For example, if set to SX, then if the 1d_nwk Conn_2D attribute for a pit is empty, SX will be used, saving the user to specify SX at pits. To disconnect a pit NO can be used for the Conn_2D attribute. Also, it is now possible to have the L and Z options at the one pit (eg. SXLZ) - previously only a maximum of three characters was allowed.
Section 4.5.2.2	If a manually digitised manhole occurs more than once within all of the 1d_mh layer(s), the last occurrence prevails and overrides the prior ones
Section 4.13.2	XF files can be read directly by specifying the XF filename (with a .xf4 or .xf8 extension).
	The bug relating to problems with using -e1 or -s1 have been fixed, and additional checks have been added.
	"ERROR 0252 - ... is not a supported file format for the last command." may occur in 2010-10-AA if the .mif or .mid extension was omitted. This is now fixed.
	If the .mid file is incomplete (due to manual editing, not if exporting from a GIS), Build 2010-10-AA would crash. Build 2010-10-AB has fixed this and issues ERROR 0254 if the .mid file is incomplete.
	GIS layers referred to via the Calibration Points MI File command are now opened and closed during the startup phase so the existence of the layer is checked (rather than at the end of the simulation as this causes issues if the layer does not exist or has a save date later than the .tab file), and also so that the layers are copied if using the -c or -ca run-time option.
	WARNING 2460 is issued if the 2d_rf f1 or f2 attributes are zero.

Links	Description
	For SMS TUFLOW licencees, this build supports the new SMS Version 11 licensing.
New Features, Bug Fixes and Enhancements in the 2010-10 Release	
Section 5.2 Section 5.1.1	64-bit (w64) single and double precision versions of TUFLOW are provided with the 2010-10 release. The 64-bit build has been benchmarked on a range of models and has been found to reduce run times compared with the 2009-07 32-bit build by 25% to 50%. The 64-bit builds also have no RAM limitations (other than the physical limitation of the amount of RAM on the computer). A 64-bit compatible dongle is required to run the 64-bit builds of TUFLOW – see Section 5.1.1.
Model TUFLOW Build Model TUFLOW Release Model Platform Section 5.2	New commands Model TUFLOW Build, Model TUFLOW Release, Model Platform to force a simulation to use a specific version/build/release of TUFLOW. Also incorporated into DB builds for the prior releases back to 2006-06.
Section 4.3	Finally ArcGIS friendly! The TUFLOW code has been overhauled to use a set of generic GIS routines. The new routines can read both .mif and .shp file formats. It will also be much easier now to incorporate other GIS/CAD formats! NOTE: All of the Read MI and Read MID commands in the manual have been replaced by Read GIS and Read RowCol. Read MI and Read MID remain valid commands and will do so indefinitely, so they can continue to be used. You can even use Read MI to read a .shp file if you wish! Read GIS is identical to Read MI, and Read RowCol is identical to Read MID. The Read GIS and Read RowCol syntax was adopted with the recognition of .mif, .shp and other future formats, and to make the intent of these commands clearer.
Section 4.12.1 Event File Define Event End Define BC Event Source 1D Table 5.2 Model Events	Powerful new features that facilitate easy simulation of multiple events (see Section 4.12.1). • New .tcf command Event File that defines the name of the active event (.tcf) file. • New .tcf control file and commands Define Event and End Define that define the commands that are specific to an event. • New .tcf command BC Event Source that is an alternative to using BC Event Name and BC Event Text. BC Event Source maybe repeated to define up to 100 different combinations of BC source texts and names. • A simulation's event name(s) can be set using Model Events, or by using the new run time option <code>-e {i} <event_name></code> where {i} is optional and can be a number from 1 to 9 allowing up to 9 different defined events to be applied in one simulation. See Table 5.2.

Links	Description
Section 4.12.2 If Scenario Table 5.2 Model Scenarios	Powerful new If Scenario command that allows different commands in .tcf, .tgc, .tbc and .ecf files to be applied depending on the scenarios being simulated (see Section 4.12.2). A simulation's scenario name(s) can be set using Model Scenarios, or by using the new run time option <code>-s{i} <scenario_name></code> where {i} is optional and can be a number from 1 to 9 allowing up to 9 different defined scenarios to be applied in one simulation. See Table 5.2.
XF Files Section 4.13.2	New .tcf command “XF Files == [{ON} OFF]” to globally activate/deactivate whether to use XF files. TUFLOW .xf files are binary dumps of selected input files and markedly reduce startup times for large models. The 2010-10 release has incorporated XF files for Read RowCol Zpts and TINs generated using Create TIN Zpts, Read GIS Z Shape and Read GIS Variable Z Shape. XF files are planned to be added to other commands that read and/or generate large datasets in future builds. Commands that use XF files have a new option “XF ON” or “XF OFF” so that the user can switch XF files on or off for specific data reads. For example “Read RowCol Zpts XF OFF ==” would not generate or use XF files for that layer.
Map Output Data Types Table 7.2	New approach for identifying areas contributing to mass errors. MB1 map output now tracks the convergence at 2D cells since the last map output time, rather than at an instant in time. MB2 is the cumulative of MB1 for the entire simulation. The new approach is much more effective in identifying problem areas within your model than the old MB1 option which was the mass error as a percentage at an instant in time that would tend to highlight high (but not problematic) errors in cells just becoming wet or with very shallow flow, and would give a false lead on identifying problem cells. MB1 and MB2 are only for 2D cells (not 1D WLL triangles, which will appear as zero).
Mass Balance Output Interval Section 7.5	Mass balance output has been reworked so that the display on the screen and the _MB.csv output files can be at different time intervals. New tcf command Mass Balance Output Interval for setting the interval in the _MB.csv files. There is also now a summary at the end of the simulation on the display console and .tlf file of key model performance indicators.
Section 7.3.1	The XMDF format is now available as an alternative to the .dat files. The format offers a range of advantages over the .dat format: • All map output in one file (ie. all .dat files become one .xmdf file). • Much faster to access and instant load up when opened. • Data are stored in a folder structure. • Format is an industry standard and recognised by other software, with executable code available to write applications.

Links	Description
Section 4.4.11	Up to nine (9) separate GIS layers can now be specified for Read GIS Z Line , Read GIS Z HX Line , Create TIN Zpts , Read GIS Z Shape , Read GIS Variable Z Shape , Read GIS FC Shape and Read GIS Layered FC Shape . Prior to the 2010-10 release a only second points layer could be specified. The increase offers greater flexibility in managing the range of objects that can be required for these commands, and makes it easier to use .shp layers as individual .shp layers cannot contain objects of different types (eg. points and lines cannot occur in the same .shp layer).
ESTRY Control File Start 1D Domain End 1D Domain 1D Section 4.2.1	1D (.ecf) commands can now be included in the .tcf file between Start 1D Domain and End 1D Domain block(s), and the need for a .ecf file for a 1D/2D model is now optional. For the new .tcf file 1D commands must be prefixed by 1D . Also note that ESTRY Control File is now recognised every time if specified more than once (in previous releases only the last occurrence would apply).
Section 4.5.2 Manhole Default Loss Approach Manhole Default C Exit Coefficient Manhole Default R Exit Coefficient Manhole Default Side Clearance Manhole Default Type Manhole K Maximum Bend/Drop Manhole Minimum Dimension Manholes at All Culvert Junctions	A major new feature of the 2010-10 release are manholes. By default, manholes are automatically created and energy loss coefficients re-calculated every timestep based on the flow distribution of inlet/outlet culverts, chamber dimensions, culvert dimensions, relative angles of culverts to each other and change in invert levels. An enhanced version of the Engelhund method is used by default for calculating the energy losses. Manholes can also be manually specified using 1d_mh layer(s). Two energy loss methods are available, Fixed and Engelhund. For details refer to the Manhole commands and Section 4.5.2. To not automatically assign manholes use Manholes at All Culvert Junctions == OFF.
S Channel Approach	Changed default for S Channel Approach to METHOD B to give improved performance for modelling open 1D (S) channels where the flow is steep and fast.
Section 7.6 Table 7.9	Added in Simulation Summary info at end of simulation display console and .tlf file to provide a guide as to the healthiness of the simulation. Some of these indicators are also shown in the “_ TUFLOW Simulations.log” files.
Table 5.2	External 1D schemes such as ISIS and XP-SWMM now have access to all of the run time options in Table 5.2 including the new –e and –s event and scenario management options, and –c and –ca for making a copy of a model.

Links	Description
Free Overfall Factor	Free Overfall Factor default value changed from 0.8 from 0.6. Provides some minor enhancement in performance for models with rapidly varying flow.
Section 4.7.4.1 Structure Losses	New “EXCEPT BG TABLE” option for Structure Losses (and default for the 2010-10 release) that only adjusts entrance and exit losses to culverts and automatically generated BG tables. For backward compatibility, use Structure Losses == ADJUST. Bridges with manually specified BG (or LC) tables are by default now not adjusted because if using references such as Hydraulics of Bridge Waterways, the loss (K) coefficients derived already have an allowance for the approach and departure velocities.
Section 7.2.5	1d_nwk_check layer is now called 1d_nwk_C_check and has two extra attributes at end: • “i 1d_nwk” is an integer indicating which 1d_nwk input layer the channel came from (in order that they occur in the input). • pSlope is the percentage slope of the channel (allows easier identification of steep channels).
Table 5.2	New –ca command line option to copy ALL files associated with a layer or .xf file. This includes copying .tab, .map, etc files. If delivering a model to another user, suggest using this option. If wanting to copy the bare minimum use –c.
Section 5.3	When specifying run options you can now use long dash “_” and “/” as alternatives to a short dash “-”. Also incorporated into DB builds for prior releases back to 2006-06.
Section 4.8.1	2d_po output .csv files increased to four (4) decimal places.
	Improved CPU limiting management if using the SoftLok (blue) dongles.
Table 5.2	Added –wibu run time option that forces TUFLOW to only search for a WIBU dongle.
	Fixed bug that didn’t correctly report the connected ISIS nodes in the 1d_to_2d_check.mif layer.
Section 4.10.3	Can now have Event Text more than once in BC Database line (previously only the first occurrence was changed).
Rainfall Boundaries	Rainfall Boundaries == STEPPED is the new default for the 2010-10 release (previously SMOOTHED TIME CENTRED was the default).
HQ Boundary Approach	The new improved approach (Method B) is set as the default for the 2010-10 release.
Link 2D2D Approach	The new improved approach (Method B) is set as the default for the 2010-10 release.

Links	Description
	Fixed bug that caused TUFLOW to stop if two QT boundaries selected the same cell. ERROR 2032 is now reported.
	Fixed bug that caused problems when several simulations were trying to write the "Projection.mif" layer to the results folder simultaneously.
TIN Angles	New .tgc command TIN Angles to set the control angles in degrees for triangulating regions in Read GIS Shape commands.
Input Drive Output Drive	Fixed up bug with Input Drive and Output Drive that didn't change the drive letters of some files such as the .tlf and .wor files.
	Fixed bug that caused the last level of an automatic HQ tables to not be ascending if the difference in elevation from the lowest to highest 2D cell along the HQ boundary is less than 1m.
	Fixed bug that did not correctly write to the 2d_sh_obj_check layer if a THICK line selected only one cell and both cross-hairs of that cell were crossed by the Z line.
	The maximums and minimums map output for q and E have been disabled because of the change in tracking maximum velocities (see Maximum Velocity Cutoff Depth). As the time of peak velocity that is output is no longer that necessarily at same the time of peak water level, incorrect values can occur for the maximum q and E which were based on the q and E at the time of peak water level.
Minimum Channel Conveyance Length	New .ecf command Minimum Channel Conveyance Length that automatically increases the conveyance (and storage) length if the channel's length is shorter. The default setting is zero (ie. no change to any channel's length). Does not affect pit channels.
	Fixed bug that caused a crash if a Channel ID was an integer number exceeding 2147483647. Workaround for previous releases is to change the Channel ID!
	Improved symbology in 1d_nwk check layers.
	Finally, fixed field inputs 1D and 2D are no longer supported!
New Features, Bug Fixes and Enhancements in 2009-07 Release (as per the 2009-07 Release Notes available from www.tuflow.com)	
	Memory allocation error fixed that sometimes caused a crash if using Read GIS FC Shape .
Simulations Log Folder	New command Simulations Log Folder that allows the path to the global log folder to be set within the .tcf file.

Links	Description
	Fixes a bug that occurred if either of the f1 and f2 rainfall factor attributes in a 2d_rf layer were not set to one (1). The bug caused the rainfall to be incorrectly applied, and would have been evident by a substantially incorrect rainfall inflow onto the model, with a non-uniform application over each rainfall polygon(s). Note that this bug was corrected in the 2008-08-AG build, but remains in previous releases to 2008-08 (eg. 2006-06 and 2007-07). Due to the deficiencies of releases prior to 2008-08 to represent direct rainfall modelling it is not planned to incorporate this bug fix into releases prior to 2008-08. A workaround is to use the Col 2 multiplication factor option in the BC Database .csv file (the factor should be set to f1*f2).
	Bug fix for when a GIS layer only contains MIF objects not used by TUFLOW (eg. a rectangle) that caused TUFLOW to stop with a message stating that it could not allocate sufficient RAM. MIF objects used by TUFLOW are points, lines, polylines and regions. If other objects such as rectangles, ellipses, text, etc are encountered the object is ignored. In some instances a WARNING 2073 is issued. This fix was also incorporated into Build 2008-08-AG.
	Improved approach for removing coincident vertices around a TIN region – no backward compatibility provided.
HQ Boundary Approach	Improved approach (Method B) for automatically generated 2D HQ boundaries.
Link 2D2D Approach	Improved approach (Method B) for linking 2D domains .
Rainfall Boundaries	New .tcf command that offers alternatives to how direct rainfall boundaries are applied in either a modified hydrograph curve or in a stepped approach as a rainfall hyetograph histogram. This command was introduced for the 2009-07 release. The Rainfall Boundaries == STEPPED approach is the default for the 2010-10 release.
Read GIS Z Shape	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.
Maximum Velocity Cutoff Depth Section 7.3.1	New .tcf command Maximum Velocity Cutoff Depth for setting the depth above which the maximum velocity is tracked as the maximum velocity, rather than the velocity at the peak water level.
Section 7.3.1	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 and the time of minimum values under time 910,001). The time of peak water level is no longer written to the _h.dat file as time 99999.1.

Links	Description
Time Output Cutoff Depths Section 7.3.1	New .tcf command Time Output Cutoff Depths to output the duration of inundation and the time of first inundation above specified depths to the _Times.dat file.
Section 7.3.1 Table 7.2	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 .
Rainfall Gauges	New .tcf command, Rainfall Gauges that is useful quality checking rainfall inputs where only one rainfall time-series per 2D cell is intended.
Map Output Data Types Table 7.2	Two new options for Map Output Data Types , namely BSS (Bed Shear Stress) and SP (Stream Power) are available for both 2D and 1D WLL map output.
	Commas are now allowed within character strings when reading .csv formats. The character string must be enclosed with quotes.
	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 and crash). 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?!!!).
Table 4.32 Table 4.33	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.
	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 was not zero.).
Section 4.10.7.1 Table 4.34	A new option has been added to the .tbc Read GIS SA 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 reaches the reservoir.

Links	Description
	Due to change mentioned above to use 8-byte integer pointers, and ISIS support for iDP versions, ISIS v3.3 must be used with Build 2009-07-AB or later.
	Map output .dat files are now closed prior to writing to “_ TUFLOW Simulations.log” files to ensure that they are closed off in case there are problems writing to the .log files across the network.
Table 4.33	The 2d_bc layer’s “a” attribute for HX lines can now be used for assigning additional energy losses (FLC) along HX lines. This can be useful for 1D/2D models where additional energy losses are needed to model the flow from a river (1D) onto the floodplain (2D). Alternatively the .tgc Read GIS FLC can be used. Note, that to ensure backward compatibility the “a” attribute must be set to zero (0.0) for HX lines. Incorporated into Build 2009-07-AE.
	The maximum number of polygons within one region object has been increased from 1,000 to 10,000 as of Build 2009-07-AE.
	Read GIS Shape commands bug fixes: • Fixed bug that didn’t apply the ADD option correctly for the various Read GIS Shape commands. • Fixed bug that didn’t apply dZ attribute if NO MERGE specified and no points snapped to region perimeter. • ERROR/WARNING 2049 (zero Z value) is now not issued if using the ADD option for a Read GIS Shape command.
Section 4.8.2 Read GIS Z Shape Route Set Route Cut Off Values Set Route Cut Off Type	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.
Section 4.4.12.3	Manning’s n versus depth curves of up to 100 points can be specified for different materials. There is also a new .csv format for managing materials as an alternative to the .tmf file.
Section 4.7.4.7	New 1d_nwk M channels that allows the modeller to define the flow through a channel based on a flow matrix.
Section 4.7.4.8	New V option for 1d_nwk W (weir) channels that allows the modeller to vary the cross-section geometry of a weir over time using a trapezoidal shape.

11 Utilities

Section Contents

11	UTILITIES	11-1
11.1	Running DOS .exe Utilities	11-2
11.1.1	Using the Right Mouse Button	11-2
11.1.2	Using Batch (.bat) Files	11-4
11.2	TUFLOW_to_GIS.exe	11-5
11.3	dat_to_dat.exe	11-9
11.4	tin_to_tin.exe	11-12
11.5	12da_to_from_mif.exe	11-14
11.6	asc_to_asc.exe	11-16
11.7	convert_to_ts1.exe	11-17
11.8	xsGenerator.exe	11-19
11.9	TUFLOW_Tools.xls	11-21
11.10	MapInfo Tools	11-21

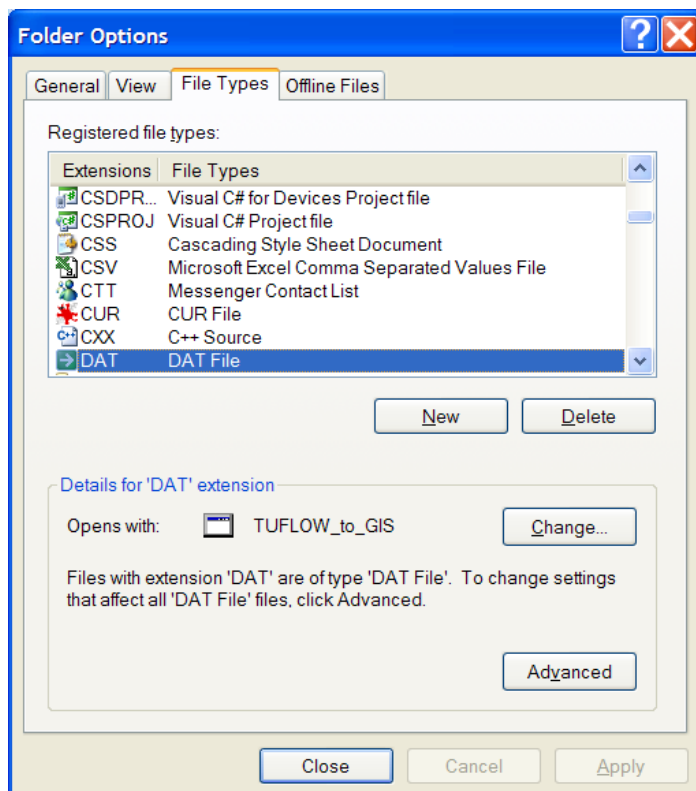
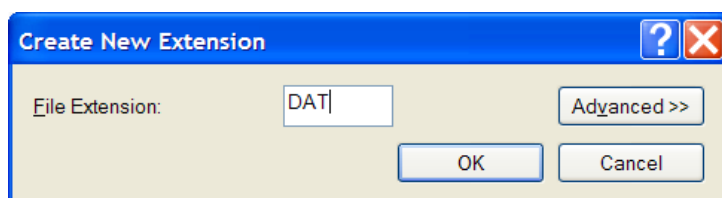
11.1 Running DOS .exe Utilities

Many of the utilities are, like TUFLOW, DOS .exe engines with no user interface. The easiest way to run them is by either customising the right mouse button or using .bat files. The latter provides the opportunity to carry out bulk processing of data.

11.1.1 Using the Right Mouse Button

The right mouse button option is suited to operations that only require one file as input. For example, to set up TUFLOW_to_GIS.exe to create a 3D surface for import into Vertical Mapper, Spatial Analyst or other grid package, based on the maximum values in a .dat file, follow the following steps. This process can be utilised for other utility functions provided there is only one input file.

In Windows Explorer, go to Tools, Options, File Types dialogue. Check to see whether the DAT file extension has been registered by scrolling down the list of file types (alternatively enter in quick succession the letters DA). If there is no DAT file type registered, create one manually by clicking New and entering DAT into the dialogue below, then press OK.

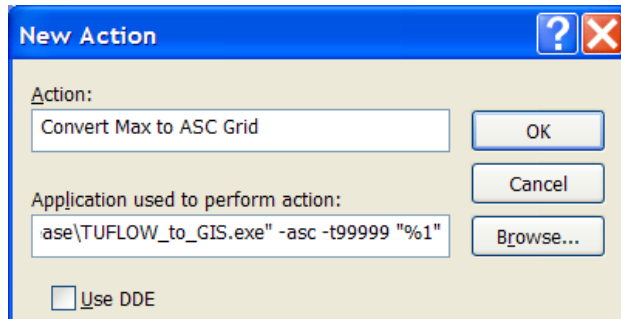


Ensure DAT is selected as the file type, and click Advanced. If you wish, change the file description and icon. Click New... and enter the text “Convert Max to ASC Grid” in the Action field. In

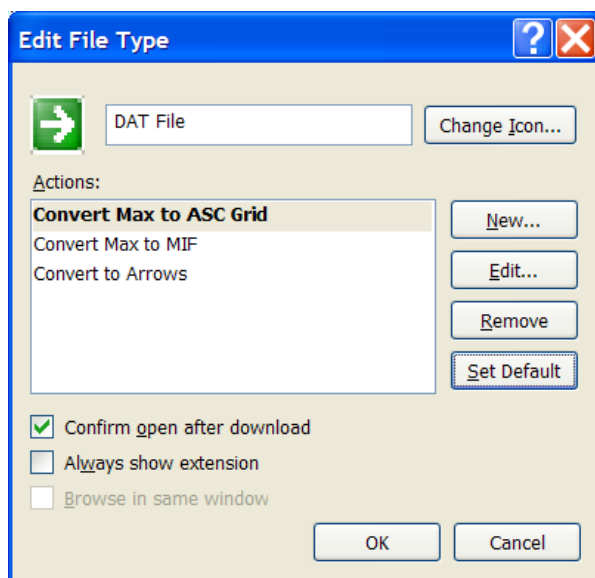
Application used to perform action:, click Browse... and select TUFLOW_to_GIS.exe on your system, then add the text:

-asc -t99999 "%1"

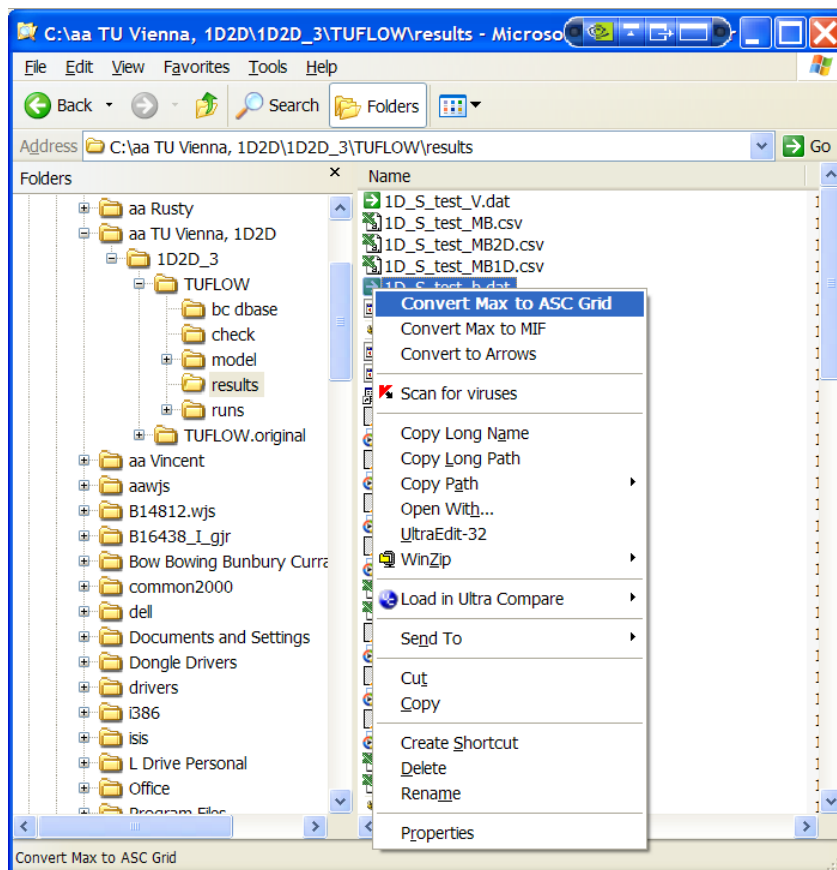
at the end of the field ensuring that there are spaces between each argument as shown in the dialogue below. Click OK.



The Edit File Type dialogue should now appear similar to that below. Other actions can be added, edited and removed as desired. One of the actions can also be set as the default (ie. that which occurs when you double click on a DAT file in Windows Explorer). When finished, click OK.



In Windows Explorer, right click on any .dat file and the actions now appear in the context list to be used as needed.



11.1.2 Using Batch (.bat) Files

Where there is more than one input file then a batch (.bat) file needs to be created. These are simply a text file containing one or more lines that execute a utility as per the examples given in the following sections. To execute the .bat file, double click on the file in Windows Explorer. To force the .bat file to pause at the end (and not simply disappear off the screen), enter the text “pause” on the last line of the file.

The example of a .bat file shown below will generate six ASCII grids for import into 3D grid software for the results from 100, 50, 20, 10, 5 and 2 year flood event simulations. Note the use of the -b switch that most utilities have to suppress the request of a press Enter key prompt, and also the “pause” on the last line.

```
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q100_h.dat
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q050_h.dat
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q020_h.dat
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q010_h.dat
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q005_h.dat
TUFLOW_to_GIS.exe -b -asc -t99999 -grid5 nile_Q002_h.dat
pause
```

11.2 TUFLOW_to_GIS.exe

TUFLOW_to_GIS.exe provides a variety of functions that converts TUFLOW output into GIS and other formats. The primary ones are:

- Convert TUFLOW .dat files to:
 - 3D surface files in the ESRI ASCII Grid format (.asc and .vmi extensions). These files can be used to create 3D grids of, for example, the TUFLOW water level surface, by importing into Vertical Mapper, Spatial Analyst and other similar software.
 - .mif files of the .dat output points (ie. at the 2D cell corners and 1D WLL points). These files can be used to interrogate in GIS TUFLOW output, create 3D surfaces generated from the points, etc.
 - Arrows of vector .dat files (eg. velocity and unit flow) in .mif format.
- Interrogate the TUFLOW results to add modelled data to a calibration point GIS data layer in .mif format. For example, the peak water levels from a TUFLOW _h.dat file can be added as an extra attribute to a layer containing recorded flood levels.
- Extract from .dat files and generate data sets for plotting and mapping longitudinal profiles. Calibration points within a specified distance from the longitudinal profile line can optionally be extracted and added to the data.
- Convert TUFLOW .2dm files to .mif or .asc (ASCII Grid format). This is useful for producing a 3D grid of the TUFLOW topography based on the ZH values.

TUFLOW_to_GIS.exe has a range of options to determine the output as discussed in Table 11.1, and requires a .dat file or a .2dm file to be specified. As a minimum either the -asc or -mif option must be specified. The remaining switches are all optional.

The program will prompt the user for the grid spacing and one or more output times in the .dat file.

Examples of using TUFLOW_to_GIS are:

```
TUFLOW_to_GIS.exe -asc -t99999 -grid5 my_model_h.dat
```

creates a 3D grid file on a 5m raster resolution of the maximum water levels in my_model_h.dat.

```
TUFLOW_to_GIS.exe -asc -grid2 my_model.2dm
```

creates a 3D grid file on a 2m raster resolution of the TUFLOW model topography in the file my_model.dat. Note that only the ZH values are available for the 2D domains. Any 1D WLL topography is included.

```
TUFLOW_to_GIS.exe -arrow -sf0.5 my_model_v.dat
```

creates a .mif file containing scaled arrows of the velocity vectors in my_model_v.dat. As -t is not specified, the user will be prompted to enter times for writing the output. Note the arrows will be scaled so that 1m/s is equivalent in length to the cell size of the first 2D domain.

```
TUFLOW_to_GIS.exe -arrow -grid5 -t99999 my_model_q.dat
```

creates a .mif file containing scaled arrows of the unit flow vectors that occurred at the peak water level in my_model_q.dat. The -grid5 option will scale the arrows so that $1\text{m}^2/\text{s}$ is equivalent to a 5m long arrow.

```
TUFLOW_to_GIS.exe -cp "calib.mif" -t99999 my_model_h.dat
```

appends an attribute to the GIS layer calib.mif and fills it with the peak water level from my_model_h.dat for each point object.

```
TUFLOW_to_GIS.exe -lp "profiles.mif" -lpmd50 -t99999 my_model_h.dat
```

produces .csv files for each profile line in profiles.mif. The .csv files contain several columns that allow graphing of the bed and water level profiles, structure inverts and obverts (soffits) and other information along each line. The -lpmd50 option will produce an additional GIS layer output containing tick marks at 50m intervals along each profile line.

```
TUFLOW_to_GIS.exe -lp "profiles.mif" -cpsd80 -cp "calib.mif" -t99999 my_model_h.dat
```

appends the peak water level to calib.mif and produces .csv files for each profile line in profiles.mif as per the examples further above. Additional information is contained in the profile .csv files to also allow plotting of calibration points within 80m of the profile lines as set by the -cpsd80 option.

Table 11.1 TUFLOW_to_GIS.exe Options (Switches)

Switch	Description
-asc	The output file will be in ESRI ASCII Grid format, a text format recognised by most 3D grid modelling software.
-mif	The output fill will be in the .mif/.mid MapInfo Interchange format recognised by most GIS systems. Note: to ensure the .mif file contains the correct GIS projection, make a copy any .mif file containing the GIS projection used for the TUFLOW model (ie. whatever was used for MI Projection), and rename the file "Header.mif". Make sure the "Header.mif" file is in the same folder as the .dat file TUFLOW_to_GIS.exe is being applied to.
-vmi	Not supported anymore. Use -asc instead.
-arrow -vector	The output will contain arrows as polygons to display vector data (eg. velocities and unit flows) in GIS. Must use with the -mif option.
-grid<dist>	The output will be on a regular grid of spacing <dist> metres. If no <dist> is specified the user is prompted to enter a distance. For -asc output it sets the raster cell size of the 3D ESRI ASCII grid. For -mif output sets the interval in metres for the output on a north-south alignment.
-cp "<cp.mif>"	Generates .dat file values at calibration or observation points. Will append values to <cp.mif> as an additional attribute. The values are interrogated from the .dat file based on the location of each point in <cp.mif>. Must be a space between -cp and "<cp.mif>".

Switch	Description
-cpsd<sd>	The search distance <sd> in metres for finding calibration points in <cp.mif> near longitudinal profile lines in <lp.mif>. To be used with -cp "<cp.mif>" and -lp "<lp.mif>" options. The default search distance is 100m.
-lpmd<md>	The distance <md> in metres between markers that are output along longitudinal profiles lines in <lp.mif> for display in GIS. The output GIS layer is named with a _lp<md>.mif extension. The default distance is 100m. A _lp.mif file is also created containing a GIS layer of the vertices of each profile line with the distance along the line as an attribute.
-lp "<lp.mif>"	Generates .dat file values along longitudinal profile lines. The first attribute of the layer is a unique label (character string) identifying the profile line. A .csv file is created for each profile line containing information to allow graphing of the .dat file longitudinal profile, vs distance along the line. Other information is produced such as the elevations of any structure inverts and obverts, etc. To use the feature the .2dm file must have been created by TUFLOW Build 2007-07-AA or later. If the -cp "<cp.mif>" option is specified, any calibration points within the -cpsd<sd> search distance from the lines are also output to the .csv file to allow plotting of calibration points along the profile. The values are interrogated from the .dat file at each vertex along the profile line. Must be a space between -lp and "<lp.mif>".
-fso	Interrogated water level values are only extracted for free-surface or open channel flow. For closed conduits that are under pressure flow, the level output is the obvert or soffit level of the conduit. Useful when producing flood maps involving models with pipe networks that are under pressure flow, which may give a false impression of the flood extent or flood height along the pipe if the pressure head is used.
-2dm "<2dm_file>"	To explicitly specify a .2dm file. By default TUFLOW_to_GIS assumes the .2dm file has the same name as the .dat file. Must be a space between -2dm and "<2dm_file>".
-b	Batch mode. Suppresses prompt to press Enter at end of processing. Used in .bat files where two or more files are to be processed.
-t<time>	Explicitly specify the time in the .dat file that the output is to be based on. Use -t99999 to specify the maximum values. If this option is omitted, the times in the .dat are displayed and the user is prompted to enter a time.
-rmp "<mesh_part>"	Remove <mesh_part> from the output. To create a .2dm file with the meshparts included set Meshparts == ON in the .tcf file. Useful if you want to remove a 2D domain or a 1d_nwk layer (eg. a pipes layer) from the output.
-sf<sf>	Scale factor to scale the size of the arrows. The default is <sf> = 1, which means either: if -grid is not specified the cell size of the first 2D domain in the .tcf file is equivalent in length to an arrow of 1m/s for V.dat files or 1m²/s for q.dat files.

Switch	Description
	(Note in previous builds of TUFLOW_to_GIS it was equivalent to 0.5m/s or 0.5m²/s. The Build 2007-07-AA therefore produces arrows half the size of previous builds of TUFLOW_to_GIS if using the same setting for -sf.) • if -grid is specified, the grid <dist> value is used instead of the cell size of the first 2D domain to scale the arrows. For example, if the first 2D domain has a cell size of 10m, by default, a 1m/s velocity arrow will be 10m long. If -sf0.5 or -grid5, is specified the 1m/s velocity arrow will be 5m long.

11.3 dat_to_dat.exe

dat_to_dat.exe has a range of options to post-process TUFLOW output into a new .dat file as discussed in Table 11.2.

One or more .dat files must be specified depending on the option specified. For the -con and -max options, any number of .dat files can be specified. For the -dif and -ddur options two input .dat files must be specified with an optional third .dat file to explicitly name the output file. All other options require just one .dat file to be specified.

Examples of using dat_to_dat are:

```
dat_to_dat.exe -max -t99999 Q100_1h_h.dat Q100_2h_h.dat Q100_3h_h.dat
```

outputs a .dat file called Q100_1h_h(maxmax).dat that contains the maximum of the maximums from the three simulations Q100_1h.tcf, Q100_2h.tcf and Q100_3h.tcf. The values will be for time 111111 in the output .dat file, and the event that caused the highest flood level will be indicated by a number 1, 2 or 3 in the 222222 output time, where 1 is the first .dat file, Q100_1h_h.dat, etc.

```
dat_to_dat.exe -k99999 Q100_1h_h.dat
```

outputs a .dat file Q100_1h_h(k99999).dat that contains the only the maximum values time 99999. Data for all other times is not included in the output file.

Table 11.2 dat_to_dat.exe Options (Switches)

Switch	Description
-b	Batch mode. Suppresses prompt to press Enter at end of processing. Used in .bat files where two or more files are to be processed in succession.
-con	Concatenates (joins) two or more .dat files into one. Each .dat file must be from the same 1D/2D mesh. The output.dat filename is the same as the first .dat file with a "(con).dat" extension.
-dif	Take the difference from the first two .dat files. If an optional third .dat file is specified, this is used as the output file, otherwise, dat_to_dat creates its own output filename using the names of the two input .dat files. The output is the first .dat file minus the second .dat file. A special wet/dry algorithm is used. If _h.dat files are not specified, the algorithm opens these files, as well as opens the .2dm file associated with the .dat files, and uses the water levels and ZH Zpt values at the 2D cells and elevations at the WLL triangle corners to determine whether a node (corner of an SMS element) is wet or dry. This allows two special values to be output in the event that a node is dry in one .dat file and wet in the other or vica versa. A value of -99 is used to indicate that a node is dry in the first .dat file, but is flooded in the second.dat file, while +99 indicates that the node was wet in the first.dat file but is dry in the second.dat file. The -t option can be used to carry out the difference at a particular time (rather than all times). The most common time is to take the difference at the flood peak, ie. specify -t99999.
-nowetdry (use with -dif only)	Suppresses the use of the special wet/dry algorithm described for the -dif option. The output .dat file simply contains difference between the values at each SMS element node.
-ddur<cov>	Determines the difference in inundation duration in hours up until the time specified by the -t option. The <cov> cut-off value is used to identify locations where the water levels have yet to start falling, and to handle locations where the water levels have become stagnant making calculation of the change in inundation duration difficult (usually a small number is specified here such as 0.1m, eg. specify -ddur0.1). The water level must fall by <cov> metres after its peak. If an optional third .dat file is specified, this is used as the output file, otherwise, dat_to_dat creates its own output filename. The output is the first .dat file minus the second .dat file.
-dur<cov>	Determines the duration in hours that <cov> in metres elevation is exceeded.
-k<t>	Keep data for time <t>. Outputs a new .dat file that contains only the data for time <t>. Useful if you want to email just the maximum output for a large model! Specify -k99999 to just keep the maximum data. The output.dat filename is the same as the .dat file with a "(k<t>).dat" extension.
-max	Maximum of all .dat files. Popular option to take the maximum of one or more .dat files and output to a new .dat file. Very useful for determining the peak value of

Switch	Description
	several flood events of differing duration. Two output data sets are written to the output .dat file: - Time 111111 is used for the values (ie. the maximum or peak value that occurred at each node of all the .dat files specified). - Time 222222 is used to indicate which .dat file the peak occurred. An integer value is output, 1 being the first file, 2 the 2nd, 3 for the 3rd, and so on. Mapping of this data set displays which duration caused the highest flood level (ie. the map shows the spatial variation in the critical durations). Can also be used to extract the maximum values from a single .dat file – useful if Store Maximums and Minimums was not set to store the maximums. The output.dat filename is the same as the first .dat file with a “(maxmax).dat” extension.
-r<t>	Remove data for time <t>. Outputs a new .dat file that is the same as the specified .dat file but with the data for time <t> removed. The output.dat filename is the same as the .dat file with a “(r<t>).dat” extension.
-t<t>	Specify a time <t> hours in the .dat file. Use -t99999 to access the peak or maximum output if requested using Store Maximums and Minimums. Used for the -dif, -max and -ddur options.
-times	Displays and exports to a text file “times.txt” the times in a .dat file.
-trim<cov>	Trims any values that exceed a cut-off value <cov> and sets the value to <cov> in the output .dat file.
-va	Outputs the flow direction in degrees relative to north (North = 0°, East = 90°, South = 180°/-180°, West = -90°).

11.4 tin_to_tin.exe

tin_to_tin.exe converts SMS and 12D triangulations to SMS, 12D and Vertical Mapper formats. It is very useful for transferring TINs from one package to another so as to utilise the various features offered by these different software, and takes a fraction of the time to convert 12D TINs as compared to the original tin_to_tri.exe program.

First save the TIN you wish to convert in the software's native text format as follows:

- For SMS, ensure you are in either the Mesh or Scatter module (depending on which one you wish to save), go to File, Save As... and select under Save as Type: the format "TIN Files (*.tin)". Provide a filename and export the file in the SMS .tin file format. If you have both Scatter and Mesh module data, you'll be prompted for which one you wish to export.
- For 12D, export the TIN in the .12da format as follows... (to be documented)

Next run tin_to_tin.exe from a batch (.bat file) or set up for your right mouse button. The type of TIN you wish to convert to must be specified using one of the following options as the first argument:

- -sms (or -tin) to convert to a SMS .tin file
- -12d (or -12da) to convert to a 12D .12da TIN file
- -vm (or -tri) to convert to a Vertical Mapper .tri file

The filename of the TIN being converted is the next argument. If the filename/path has any spaces, it must be enclosed in quotes.

For example:

```
tin_to_tin.exe -12d "My DTM.tin"
```

will convert the SMS TIN "My DTM.tin" to a 12D .12da TIN. The created .12da file will be named "My DTM.tin.12da".

To import the created TIN:

- For SMS, simply drag and drop the created .tin file from Windows Explorer onto SMS. Alternatively, use File Open... checking first that either the Scatter or Mesh Module is activated.
- For 12D, please see your 12D operator!
- For Vertical Mapper, go to Vertical Mapper, Create Grid, Interpolation..., Triangulation with smoothing, Next >>, Open .tri file..., and select the .tri file created by tin_to_tin.exe. Select Linear solution, choose an appropriate cell size and filename for the Vertical Mapper grid and click Finish.

Additional options are:

- -b: Specify a -b as one of the arguments to suppress the "Press Enter" request when the program finishes. This is useful if batching a series of conversions.

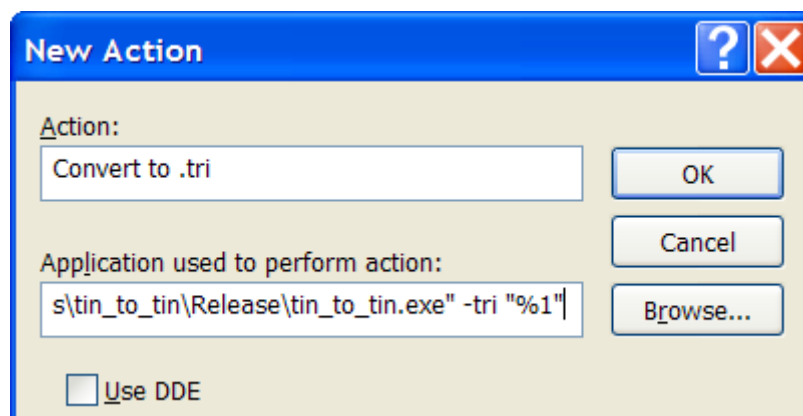
- -mif: In addition to the -sms, -12d or -tri flag, specify a -mif to also create .mif/.mid layers of the triangulation and points. Two layers are created, one for the triangles (_T.mif) and one the points (_P.mif). These could be useful for cross-checking the data and for report figures.

Notes:

- The extension of the input TIN file determines the format, so do not change the file extension.

Tips:

- In SMS, the Mesh module offers some useful options for editing TINs that the Scatter module doesn't. These include moving points (check that Nodes, Locked is not ticked), more options in terms of entering elevations, and inserting breaklines (using Nodestrings). Both modules offer the useful option of swapping triangle edges to improve the triangulation.
- To setup your right mouse button, in Windows Explorer go to Tools, Folder Options..., File Types, and, for example, for the .tin file extension, set up new action(s). For example, the New Action for converting to a .tri file would have entries similar to that below.



11.5 12da_to_from_mif.exe

12da_to_from_mif.exe converts .12da files from the 12D software (www.12d.com) to and from .mif files. If simple importing/exporting to/from 12D and a GIS simply specify the file that you wish to have translated. If the file has a .12da (or a .4da) extension, the program converts it to a .mif/.mid file format. If the file has a .mif extension, the program converts it to a .12da file.

Note, when converting from a .12da file to a .mif file without any options, 12da_to_from_mif automatically creates a .mif/.mid file suitable for use by [Read GIS Z Line](#). This is useful for importing 3D breaklines (eg. of a road design) directly into TUFLOW.

There are some very useful options as discussed in Table 11.3.

Of particular note is the -xs option to generate a TUFLOW 1D cross-section database from a 12D DTM. This approach is far more preferable to extracting cross-sections manually and is much better than extracting cross-sections from a grid based DTM (eg. Vertical Mapper or Spatial Analyst) as it only extracts points where the DTM triangle sides intersect the cross-section line, thereby keeping the number of points in the cross-section profile to a minimum, and also improving the accuracy of the profile.

The -zln option can also be very useful when the DTM river bathymetry is poor because the aerial survey is inaccurate where there is water or dense vegetation, and you need to carve a section through the DTM along the river based on a cross-section survey.

12da_to_from_mif.exe road_breaklines.12da
creates .mif/.mid files of the 2D and 3D breaklines in the file road_breaklines.12da. The .mif/.mid files can be directly used by [Read GIS Z Line](#).

12da_to_from_mif.exe 2d_hx_lines.mif
creates a file 2d_hx_lines.mif.12da. Import this file into 12D then drape these lines over the DTM and export the file, say as 2d_hx_lines_draped.12da, then execute the following

```
12da_to_from_mif.exe 2d_hx_lines_draped.12da
```

This creates 3D breakline .mif/.mid files of the TUFLOW HX lines that can be used to ensure the 2D HX cells are set to the exact elevations along the HX lines by using

[Read GIS Z Line](#) THICK == m\2d_hx_lines_draped.12da.mif
in the .tgc file.

For other examples, see Table 11.3.

Table 11.3 12da_to_from_mif.exe Options (Switches)

Switch	Description
When converting from a .mif file to a .12da file	
-zln	When converting a .mif file to .12da file, if -zln is specified polylines are interpreted in the same manner as a TUFLOW 3D breakline (see Read GIS Z Line) such that any points snapped to the polyline are used to set the elevations of any vertices along the polyline that do not have points snapped to them. The first attribute in the .mif file must be the Elevation. (Note, when converting in reverse from a .12da file to a .mif file without any options, 12da_to_from_mif automatically creates a .mif/.mid file suitable for use by Read GIS Z Line.) This is useful for creating 3D polylines for 12D where an elevation does not exist at a vertice. It is particularly useful where a river's bathymetry in a DTM is being created from cross-section surveys, and the DTM operator wishes to use the elevations at the cross-section survey points, but needs to put more shape into the breaklines being digitised between the cross-sections so as to carve out the river's bathymetry into the DTM. Using 12da_to_from_mid with the -zln option will interpolate elevations at every string vertice, something that 12D does not offer (as far as we know!).
When converting from a .12da file to a .mif file	
-hip	When converting a .12da file to a .mif file will include any 12D hipdata polylines.
-xs	When converting a .12da file to a .mif file creates a TUFLOW cross-section database (ie. one .csv file per 3D polyline) and a 1d_xs layer (1d_tab or 1d_xs format – see Read GIS Table Links and Section 4.6.3). This is used to extract 1D cross-sections from a DTM for use in TUFLOW. Cross-sections generated this way can also be viewed and edited in the SMS TUFLOW Interface. The process to create the cross-sections from a 12D DTM is as follows: - Digitise the location of cross-section lines either in a GIS or in 12D. If in a GIS, export the layer out as a .mif file and then run 12da_to_from_mif to convert the .mif file into a .12da file. For example use: <pre>12da_to_from_mif.exe xs.mif</pre> to generate a file called xs.12da. Import the .12da file into 12D. - In 12D drape the cross-section polylines over the DTM (your 12D operator should know how to do this!) to create 3D polylines with vertices where the cross-section line intersects a DTM triangle edge. - Export the 3D draped polylines from 12D as a .12da file. - Run 12da_to_from_mif using the -xs option. For example: <pre>12da_to_from_mif.exe -xs xs_draped.12da</pre> This will produce a 1d_xs layer ready for use by TUFLOW and a .csv file for each cross-section line. Keep the 1d_xs layer and the .csv files in the same folder in the event that you move them elsewhere. Using the xsGenerator.exe utility, you can also assign material values to each of the cross-section points (see Section 11.8).

11.6 asc_to_asc.exe

asc_to_asc.exe is a new utility used for generating the difference between two .asc (ESRI ASCII grid) files. It is planned to add additional functionality as required/needed.

This utility is useful for comparing two TUFLOW outputs from different .2dm meshes and for mapping previously flooded or previously flood-free areas due to proposed floodplain works.

Examples of using asc_to_asc are:

```
asc_to_asc.exe -dif Q100_dev_h.asc Q100_exg_h.asc Q100_dev_impact_h.asc
```

outputs a .asc file called Q100_dev_impact_h.asc that contains the difference of Q100_dev_h minus Q100_exg_h.

Table 11.4 asc_to_asc.exe Options (Switches)

Switch	Description
-b	Batch mode. Suppresses prompt to press Enter at end of processing. Used in .bat files where two or more files are to be processed in succession.
-dif	Take the difference from the first two .asc files (that have usually been created by TUFLOW_to_GIS – see Section 11.2). If an optional third .asc file is specified, this is used as the output file, otherwise, asc_to_asc creates its own output filename using the names of the two input .dat files. The output is the first .asc file minus the second .asc file. Both .asc files must be of the same row/column dimensions. Two .asc grids are output: - The first is the difference values between the two .asc files. A difference value only occurs at grid cells that have a value in both grids. If the cell has a null value (ie. the TUFLOW output was dry at that location) in either or both .asc grids, a null value is output. - A second grid with a “_wd” suffix is output to indicate which .asc grid cells were either previously wet and are now dry or vica versa. Importing this grid into Vertical Mapper creates a .grc (grid classification) grid that has two categories: “Was Wet Now Dry” and “Was Dry Now Wet”. This grid is particularly useful for displaying areas that were previously inundated or previously flood-free.

11.7 convert_to_ts1.exe

convert_to_ts1.exe converts output from hydrologic models to the .ts1 format recognised by TUFLOW. The .ts1 format is a .csv format, but it contains indexing and header information that significantly reduces the time to read the inflow hydrographs. If there are numerous inflow hydrographs, it is strongly recommended to use this format.

Any number of input files (of the same format) can be specified and wildcards (eg. “*.out”) can be used to specify a group of files. The options available are described in Table 11.5. One input format and one output format switch should be specified, although the default output format is -ts1, so can be optionally omitted.

For most options, an additional file “_peak_Q.csv” is output providing a summary of the peak flows for each hydrograph. If a group of files is specified, the _peak_Q.csv file is a summary of all files within the group and a second file “_peak_F.csv” contains which file caused the peak flow of all the files. This is useful for determining which storm duration produced the peak flow or is the critical duration event.

At present the program supports the hydrology models most commonly used within Australia. Other formats can be built in through supplying example files/formats and any other useful information to support@tuflow.com.

Examples of using convert_to_ts1 are:

```
convert_to_ts1.exe -wbnm -ts1 Q100_Meta.out
```

outputs two .ts1 files, one the local hydrographs and the other for the total hydrographs in Q100_Meta.out.

```
convert_to_ts1.exe -rorb -ts1 -dt5 *.out
```

outputs .ts1 files for every .out file in the folder. A summary of the peak flows can be found in the _peak_Q.csv and _peak_F.csv files.

```
convert_to_ts1.exe -rafts -ts1 Q100*.loc Q100*.tot
```

outputs .ts1 files for every Q100 .loc and .tot file in the folder. A summary of the peak flows can be found in the _peak_Q.csv and _peak_F.csv files.

```
convert_to_ts1.exe -rafts -csv Q100.tot Q050.tot Q100.loc Q050.loc
```

outputs .csv files for the four files specified.

Table 11.5 convert_to_ts1.exe Options (Switches)

Switch	Description
Input File Format Switches	
-rafts	Input files are XP-RAFTS .tot and/or .loc files.
-rorb	Input files are RORB .out files. The -dt option must also be specified.
-rows	The inflow hydrographs are in space or comma delimited files with the data in blocks of one or more rows for each hydrograph. No time data exists in the input files.
-urbs	Input files are URBS .q files.
-wbnm	Input files are WBNM _Meta.out files. Only outputs in the .ts1 format. Two files are created: • _loc_.ts1 containing the local hydrographs. • _tot_.ts1 containing the total hydrographs.
-xp	Input files are XP-SWMM .int or .ext files.
Output File Format Switches	
-csv	Output file(s) in .csv format.
-ts1	Output file(s) in .ts1 format (the default).
Miscellaneous Switches	
-b	Batch mode. Suppresses prompt to press Enter at end of processing. Used in .bat files where two or more files are to be processed in succession.
-dt<dt>	Time increment of RORB hydrographs in minutes. For example, 5 minutes would be specified as -dt5.
-s0	Insert a zero flow before the start of the hydrograph.
-e0	Insert a zero flow after the end of the hydrograph.

11.8 xsGenerator.exe

xsGenerator.exe creates TUFLOW 1D cross-section databases (ie. a 1d_xs layer and a .csv file for each cross-section) using .mif layers of survey (elevation) points and, optionally, lines.

Any number of .mif layers can be specified. The elevation points can be in the same layer, several layers or a different layer to the cross-section lines. The cross-section lines are optional (as discussed below). Each .mif layer must have four attributes as described in Table 11.6, and in the same order as in the table. For line/polyline objects of the cross-section locations, only the XS_ID attribute is used. Table 11.7 describes the options (switches) available.

The logic for interpreting the .mif layer is:

- 1 The association between an elevation point and a cross-section line is based on having the same XS_ID. It is not based on whether the point is snapped to the line or not. This means the cross-section line is purely schematic and does not have to be located at exactly the same point as the first elevation point.
- 2 The first point in a cross-section line is treated as the start of the cross-section on the left bank.
- 3 Elevation points having the same XS_ID are sorted based on the shortest distance between them. This means the points do not have to be in order across the cross-section (as often seems to be the case!). The cross-section line is not used for setting the distance (X) values for the profile.
- 4 If there is no cross-section line for a set of elevation points, this is accepted provided that at least one point (the start on the left bank) as a LCR values of “L” to denote the start of the cross-section. There must also be at least two “C” points in succession. Except for the first “L” point, subsequent isolated “L” and “R” points are ignored. This means it is possible to generate a cross-section database from a layer of surveyed points only, provided each point is assigned their respective XS_ID.

If a TUFLOW cross-section database has been created using 12da_to_from_mif.exe (see -xs option in Table 11.3 in Section 11.5), it will have created a .mif layer ready to be used for xsGenerator.exe. This layer can be used to assign material values in a GIS to the MNR attribute from a material polygon layer (eg. your 2d_mat layer), then processed by xsGenerator to recreate the TUFLOW cross-section database with material values allocated across the cross-section profiles in the .csv files.

xsGenerator can also be used to generate a cross-section database using survey data in the ISIS format (see the -isis option). The first file specified must a .mif file in the correct projection, with subsequent files being the survey files. Wildcards can be used to input numerous files (eg. *.xyz).

Examples of using xsGenerator are:

```
xsGenerator.exe -M XS_survey.mif
creates a 1d_xs layer and .csv files for each cross-section.
```

```
xsGenerator.exe -M -isis Projection.mif *.xyz
creates a 1d_xs layer and .csv files based on a group of survey files in ISIS format.
```

Table 11.6 xsGenerator .mif Layer Attributes

Attribute	Type	Description
Z	Float	The ground/bed survey elevation at that point on the cross-section. Must be in metres. Mandatory for point objects. Not used for lines.
MNR	Float	The Material, Manning's n or Relative resistance value at that point along the cross-section. Use the -M, -N or -R switch in Table 11.7 to indicate which type of value it is. Mandatory for point objects. Not used for lines.
LCR	Character(1)	One of "L", "C" or "R" to indicate whether the point is on the left, centre or right bank. If this is unknown, specify "C" for all points along the cross-section. If left blank, "C" is assumed. Optional for point objects, unless there is no associated line, in which case the start of the cross-section must be denoted by a "L" and the remaining points must be assigned a value (there must be at least two "C" points in succession, and subsequent isolated "L" and "R" points are ignored). Not used for lines.
XS_ID	Character(12)	The ID or label to be used for the cross-section. This same label is used for naming the .csv file that will contain the XZ profile of the cross-section. Mandatory for point and line objects.

Table 11.7 xsGenerator.exe Options (Switches)

Switch	Description
Input Files are .mif Layers	
-M	The MNR attribute in the .mif file is a material number.
-N	The MNR attribute in the .mif file is a Manning's n value.
-R	The MNR attribute in the .mif file is a relative resistance number.
Input Files are in the ISIS Cross-Section Format	
-isis	Indicates the input files are in the ISIS cross-section format.
-M	Use materials where the material number is equal to 1000 times the Manning's n value.
-N	Use the Manning's n values (rather than materials).

11.9 TUFLOW_Tools.xls

TUFLOW_Tools.xls contains a simple macro “Save csv”. To activate the macro, download TUFLOW_Tools.xls from www.tuflow.com/Downloads/Utilities.htm and save on your local or network drive. Open the file using Microsoft Excel and, if needed, enable macros. provided TUFLOW_Tools.xls has not been moved or deleted, each time Excel is started thereafter, the Save csv button and TUFLOW Tools menu item should appear.

If a Excel spreadsheet is open (for example, a spreadsheet containing the [BC Database](#) and all of the boundary data), ensure the sheet that is to be exported as a .csv file is active and press the Save csv button. This will:

- Save the .xls spreadsheet file.
- Export the active sheet as a .csv file of the same name as the Sheet. For example, if the sheet is named “bc dbase”, it will export the sheet as “bc dbase.csv”. It is fine to have charts, colouring and other formatting on the sheet, only the tabular data is saved to the .csv file.
- Close the exported .csv file.
- Reopen the original .xls file.

Using the above approach can save a lot of frustration when managing and exporting .csv files for boundaries and other tabular data.

If a .csv file is open the Save csv button saves and exports the .csv file without displaying all of the prompts that Excel displays if you do this manually using the File, Save As... approach.

11.10 MapInfo Tools

The MapInfo Tools are now available for purchase. These tools a must for anyone using MapInfo as their TUFLOW model development environment. Enquiries can be sent sales@tuflow.com.

12 Tips and Tricks

Section Contents

12	TIPS AND TRICKS	12-1
	12.1 MapInfo Tips	12-2
	12.1.1 Move Snapped Nodes Switch in MapInfo	12-2
	12.1.2 Setting Default Style from an Existing Layer	12-3
	12.1.3 Digitising Common Polygon Boundaries	12-3
	12.1.4 Setting Number of Undo Objects	12-3
	12.1.5 Display Nodes (Vertices) on Polygons and Lines	12-4
	12.1.6 Labelling Numbers using Fixed Number Decimal Places	12-5
	12.1.7 Selecting Subsets and Concatenating Labels	12-7
	12.1.8 Using MapInfo String Functions	12-9
	12.1.8.1 Example of Using String Functions	12-10
	12.2 UltraEdit Tips	12-16
	12.3 Vertical Mapper Tips	12-17
	12.3.1 Creating High Quality Flood Extent Maps	12-17
	12.4 Microsoft Excel Tips	12-20
	12.4.1 Save csv and Using .xls File to Manage Databases	12-20
	12.5 DOS Window Tips	12-22
	12.5.1 Copying and Pasting in a Console (DOS) Window	12-22

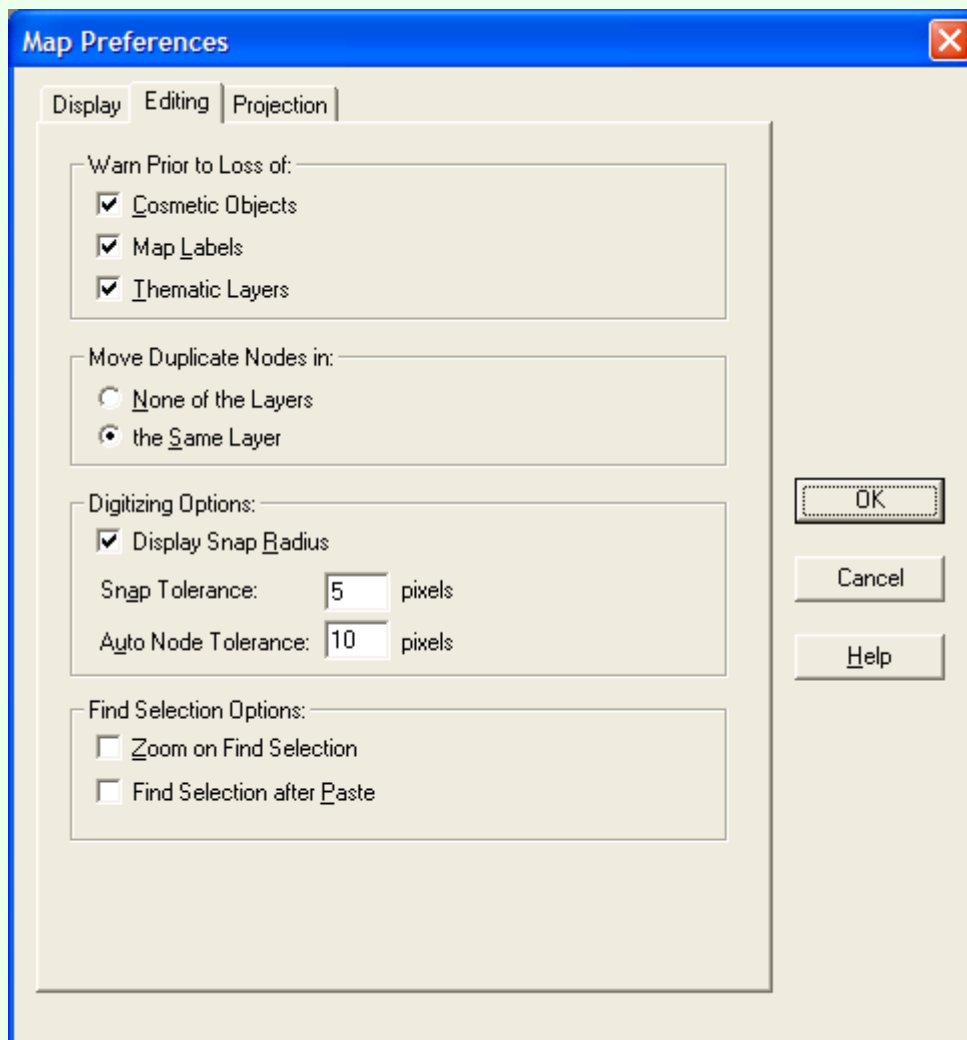
This Chapter is planned to be expanded upon in future editions to include useful tips and tricks from the variety of third party software used. Any suggestions for this section are welcome and can be emailed to support@tufLOW.com.

12.1 MapInfo Tips

12.1.1 Move Snapped Nodes Switch in MapInfo

MapInfo has an option to move or reshape other objects snapped to the object being edited. This is very useful, particularly when reshaping material polygons (so as to ensure that any other polygons that share the same node are reshaped as well), or when working with HX lines, CN lines and CD (Code) polygons in a 2d_bc layer (for example, if the HX line is reshaped, then the CN line and CD polygon are reshaped at the same time).

To use this feature, go to Options, Preferences, Map Window... to show the dialog below. Under “Move Duplicate Nodes in:” select “the Same Layer”. When this option is set, when editing an object any other objects (in that layer) that are snapped to the object being edited are also reshaped. To turn this feature off, select “None of the Layers” in the dialog below.



12.1.2 Setting Default Style from an Existing Layer

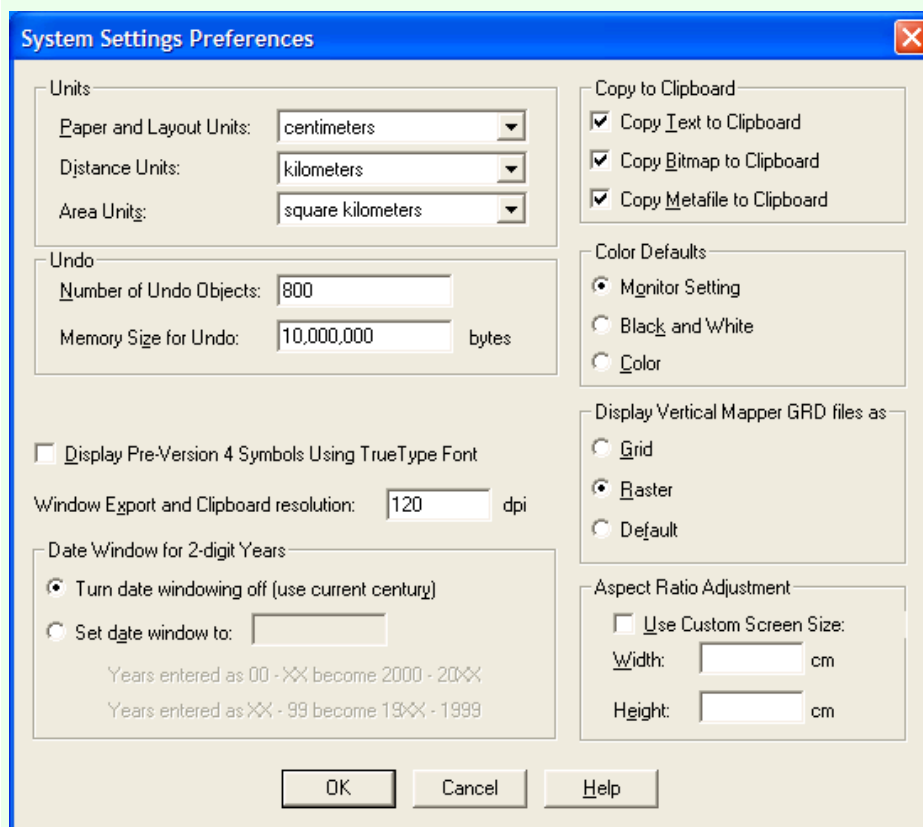
To set a point, line or region style as the default style, select an object with the style desired, click on the relevant style icon (ie. the point, line or region style button). The style of the object selected should appear in the dialog. Click OK and the default style used for any new objects should be that of the object selected.

12.1.3 Digitising Common Polygon Boundaries

When digitising polygons (regions) with common boundaries, to easily pick up the boundary of an existing polygon, use the Ctrl and Shift keys whilst Snap is on (toggle the S key to turn snapping on and off). When digitising, after snapping to the existing polygon, move the cursor further along the existing polygon's perimeter and hold the cursor over a vertice (node) so the Snap cross-hairs appear. Hold the Shift key down to pick up all of the vertices between the last digitised point and the cursor. Hold the Ctrl key down to pick up all of the vertices in the other direction (you can see which sides of the existing polygon are picked up on the screen). Decide whether the Shift or Ctrl direction is the one required, and click the left mouse button to digitise all the intermediate vertices in one click. Effective use of this feature can save a lot of time!

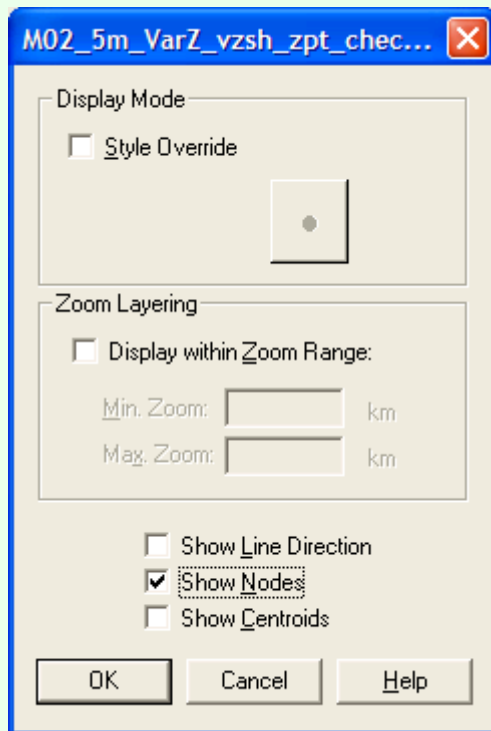
12.1.4 Setting Number of Undo Objects

The default setting for the number of Undo objects is relatively small in MapInfo. To change this, go to Options, Preferences, System Settings... to bring up the dialog below. For modern day computers there should be no problems with setting the "Number of Undo Objects:" to the maximum of 800 and "Memory Size for Undo:" to a larger number such as 10,000,000 (10Mb).



12.1.5 Display Nodes (Vertices) on Polygons and Lines

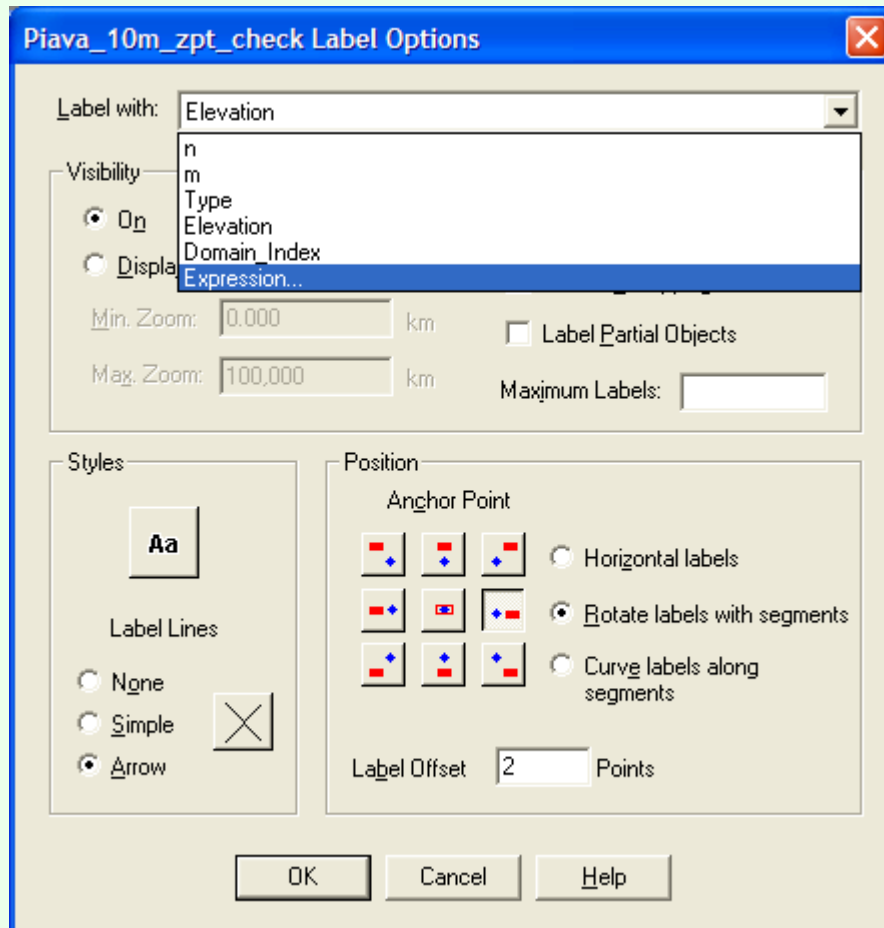
At times it is useful to display the nodes (vertices) of polygons and lines. To set this up, bring up the Layer Control, select the layer and press Display... to display the dialog below. Tick “Show Nodes” to display all nodes as small blue squares. The line direction option can also be useful.



12.1.6 Labelling Numbers using Fixed Number Decimal Places

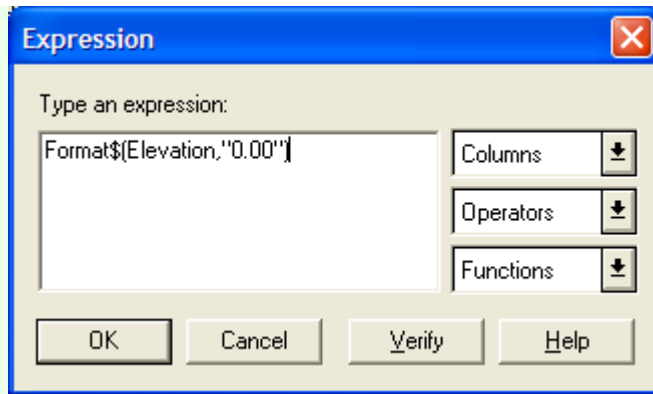
To control the number of decimal places when creating labels for a number (float) field (eg. a Z or Elevation attribute), follow these steps.

- 1 Go to Layer Control, tick the label box for the layer you wish to label and press Label.... to display the dialog below. In the “Label with:” field, select Expression....



- 2 The dialog below should now be displayed. For the expression, use the Format\$() string function as shown below (this can be entered by using the Functions drop down list on the right of the dialog). The first argument is the attribute (Elevation in the example below), and the second is a format string. The attribute can be entered by using the Columns drop down list. For the format string, use something like “0.00” making sure that you provide the quotes. MapInfo uses the same (or very similar) formatting strings as Microsoft Excel does when formatting a Number cell. Examples are “0.00” to display to two decimal places, “0.0” to display to one decimal place. Examples from the MapInfo help are shown further below. The line entered in the dialog below is:

Format\$(Elevation,"0.00")



Format\$(12345, ",#") ' returns "12,345"

Format\$(-12345, "\$#") ' returns "\$-12345"

`Format$( 12345.678, "$,##.##")` ' returns "\$12,345.68"

`Format$( 12345.678, "$,###;($,###)")` 'returns "\$12,345.68"

`Format$(-12345.678, "$,###;($,###)")` 'returns' `"($12,345.68)"`

`Format$(12345.6789, ",#.####")` ' returns "12,345.679"

`Format$(12345.6789, ",#.##")` ' returns "12,345.7"

```
Format$(-12345.6789, "#.###E+00") ' returns "-1.235e+04"
```

`Format$(-12345.6789, "#.###E-00")` ' returns "-1.235e04"

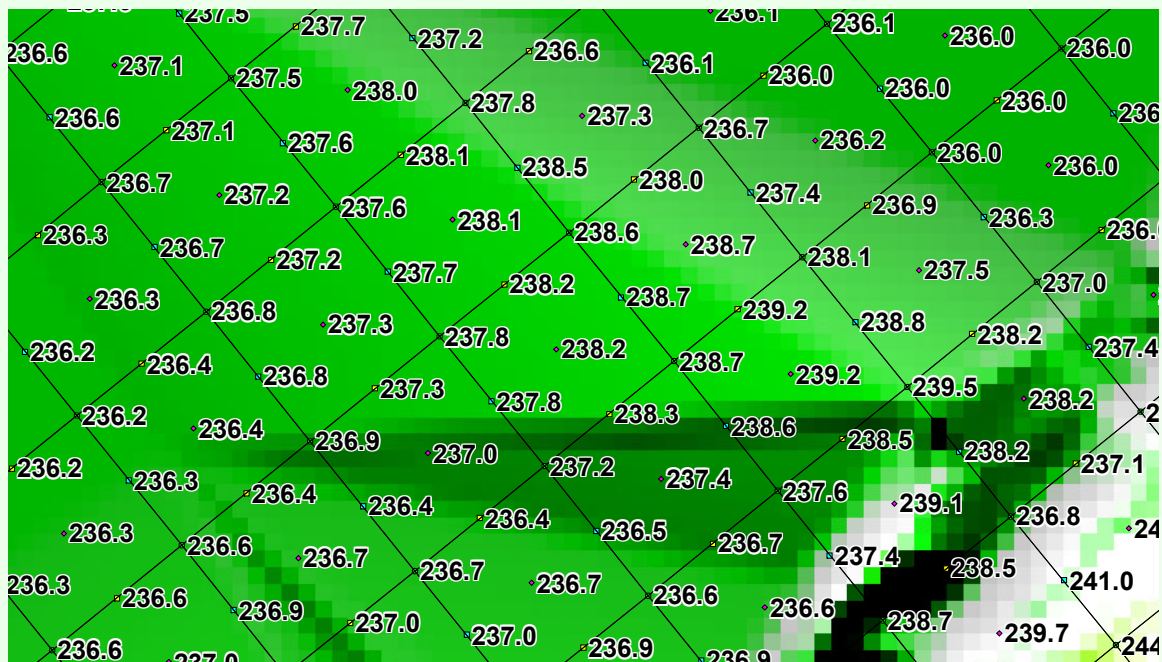
`Format$( 0.054321, "#.###E-00")` ' returns "5.432e-02"

`Format$(0.054321, "#.##%")` 'returns "5.43%"

Format\$(0.054321, "0.##\%") ' returns "0.05%"

Verify can be used to check that a valid expression has been entered. When finished press OK.

- 3 Set up the styles and other labelling options for the labels, and the labels should appear to the number of decimal places as specified, such as one decimal place in the example below.



12.1.7 Selecting Subsets and Concatenating Labels

This example illustrates how the pipes (Type = "C") in a 1d_nwk_check.mif layer are selected and their ID, number of barrels and diameter are labelled.

- 1 Import the 1d_nwk_check.mif layer into MapInfo.
- 2 To select all channels of Type "C" (ie. pipes or circular culverts) go to Query, SQL Select... to display the dialog below. Click in the "from Tables:" field and use the "Tables" drop down list on the top right to select the 1d_nwk_check layer. Click in the "where Condition:" field and using the Columns drop down list select Type (or Channel_Type for older 1d_nwk_check layers) and complete the expression as per below:

Type = "C"

ensuring that quotes are placed around the C. Finally, for the "into Table Named:" field, instead of "Selection" enter "Pipes". Use Verify to check your entries and press OK to select all circular culverts and place them in a virtual table (ie. a subset table of the 1d_nwk_check layer) called Pipes.

SQL Select

Select Columns: *

from Tables: pb_Q100_exg_001_nwk_check

where Condition: Type="C"

Group by Columns:

Order by Columns:

into Table Named: Pipes

☐ Browse Results ☐ Find Results In Current Map Window

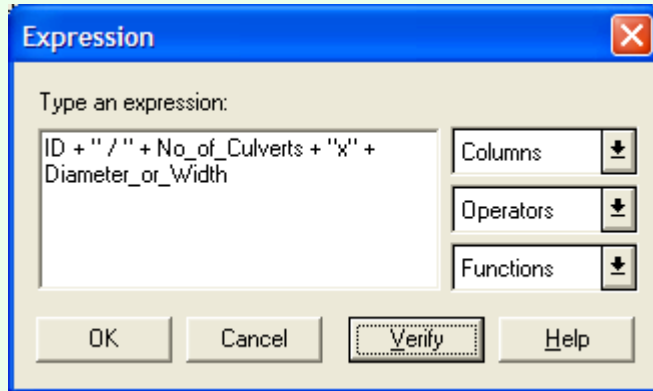
OK Cancel Clear Verify Help

Tables Columns Operators Aggregates Functions

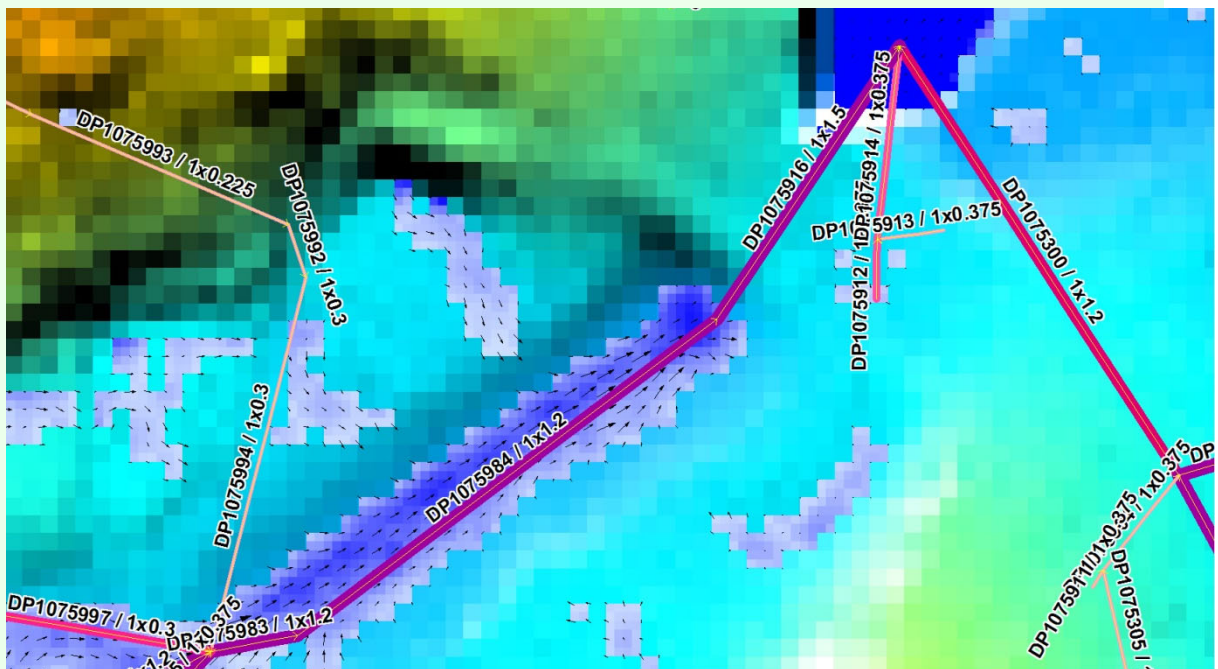
Save Template Load Template

- Go to Layer Control and add the virtual Pipes layer using Add.... Tick the label box for Pipes and press Label... to set up the labelling. For "Label with:", select Expression... at the bottom of the list, and create something like the line below using the "Columns" drop down list to select attributes to be labelled. To concatenate attributes or text use the plus symbol (+). Note that text must be enclosed in quotes otherwise MapInfo tries to interpret the text as an attribute name.

ID + " / " + No_of_Culverts + "x" + Diameter_or_Width



- Press OK on each dialog to display the labels as illustrated below.



12.1.8 Using MapInfo String Functions

MapInfo has a range of string (text) functions that offer the user excellent query and text manipulation opportunities. The string functions from the MapInfo help are documented below, and an example of using a few of these string functions for selecting and manipulating a GIS dataset is presented.

Function	Description
Chr\$(num)	Returns a character that corresponds to a character code (e.g. Chr\$(65) returns the string "A").
DeformatNumber\$(str)	Reverses the effect of the FormatNumber\$ function, returning a string that does not include thousands separators.
Format\$(num , str)	Returns a string representing a formatted number. Example: Format\$(12345.678, "\$,#.##") returns "\$12,345.68".
FormatNumber\$(num)	Returns a string representing a formatted number. This function is simpler to use than Format\$, but it gives you less control over formatting (e.g. you always get thousands separators).
InStr(num , str1 , str2)	Searches the string <i>str1</i> starting at character position <i>num</i> , and looks for an occurrence of the string <i>str2</i> . Returns the position where <i>str2</i> was found, or zero if not found. To start search at beginning, use a <i>num</i> value of one (1).
LCase\$(str)	Returns a lowercase version of the string <i>str</i> .
Left\$(str , num)	Returns the first <i>num</i> characters of the string <i>str</i> .
Len(str)	Returns the number of characters in a string.
LTrim\$(str)	Trims any spaces from the start of <i>str</i> and returns result.
Mid\$(str , num1 , num2)	Returns a portion of the string <i>str</i> starting at character position <i>num1</i> and extending for <i>num2</i> characters.
Proper\$(str)	Returns a string with proper capitalization (first letter of each word capitalized).
Right\$(str , num)	Returns the last <i>num</i> characters of the string <i>str</i> .
RTrim\$(str)	Trims any spaces from the end of <i>str</i> and returns result.
Str\$(expr)	Returns a string approximation of an expression.
UCase\$(str)	Returns an uppercase (all capitalized) version of <i>str</i> .
Val(str)	Returns the numeric value of the string; for example, Val("18") returns the number 18.

12.1.8.1 Example of Using String Functions

The points and images below illustrate an example of using some of the string functions above.

- 1 The asset database for a pipe network system was provided as a GIS layer. The layer included all pipes, manholes, kerb inlets and other data. To more effectively manage the data for the TUFLOW model, the data was split in to several 1d_nwk layers to separate the kerb inlets (pits to be connected to the 2D domain), manholes (nodes not connected to the 2D domain), culvert intakes and outlets (nodes connected to the 2D domain), and pipes.
- 2 All point objects (kerb inlets, manholes and culvert outlets) in the asset database were labelled using the key text such as “Kerb Inlets”, “Manholes”, “Culvert Intake”, and “Culvert Outfall”. To select those points to be used for the culvert intakes and outlets layer, the SQL Select dialog below was used. The LCase\$() function changes the text to lower case (to ensure there is no discrepancies due to case), and the InStr() function searches for any occurrence of the text “culvert”. The selected points are placed in a virtual layer called Culvert_Nodes.

SQL Select

Select Columns: *

from Tables: node_types_GDA94

where Condition: InStr(1, LCase\$(ASSETNAME), 'culvert') > 0

Group by Columns:

Order by Columns:

into Table Named: Culvert_Nodes

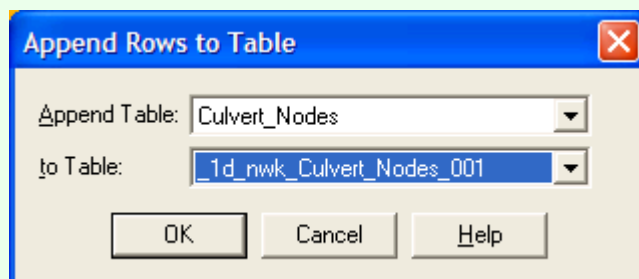
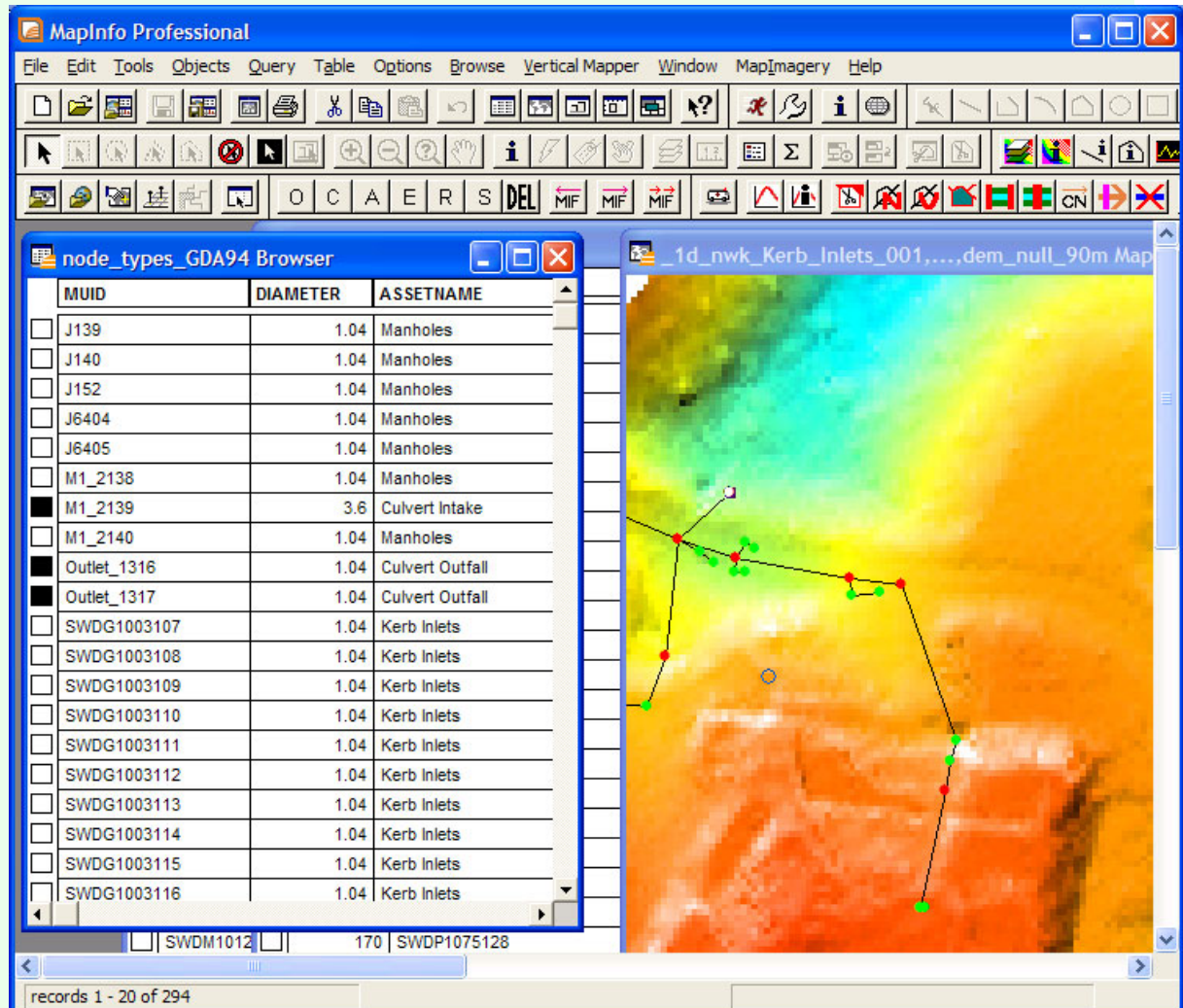
☐ Browse Results ☐ Find Results In Current Map Window

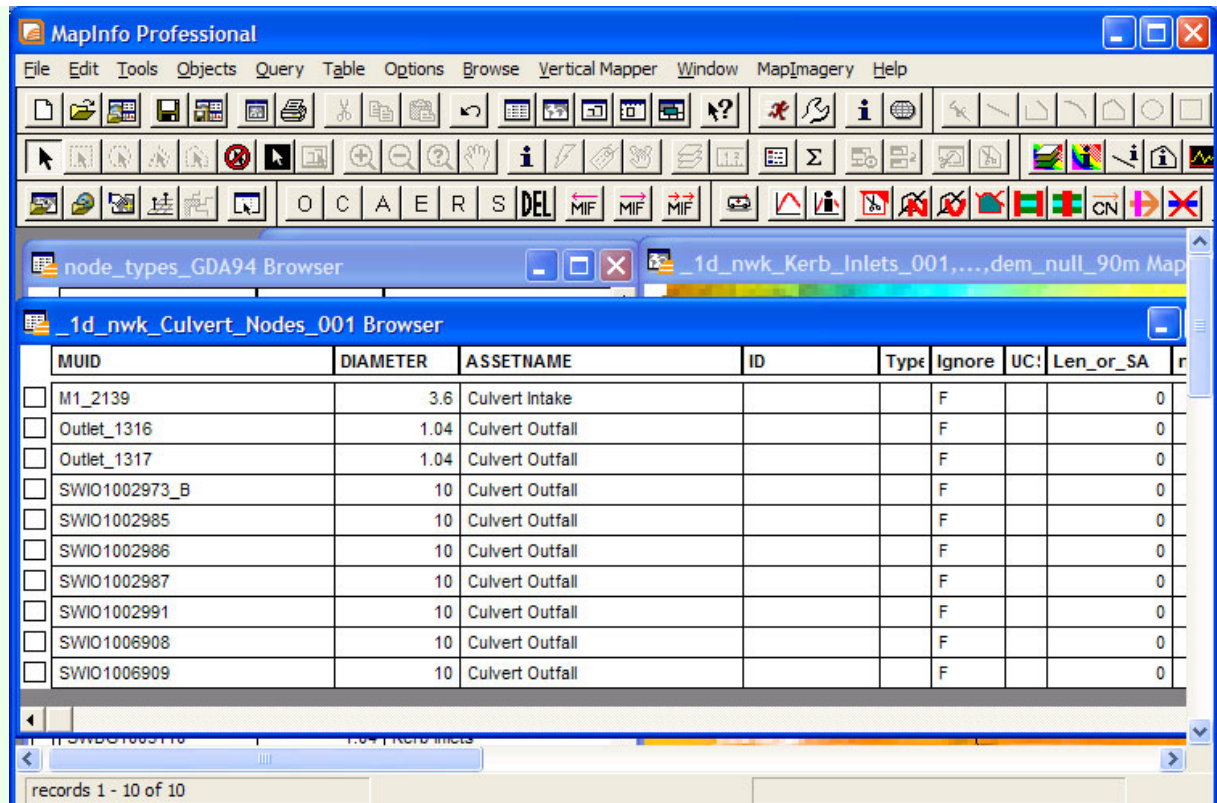
OK Cancel Clear Verify Help

Tables Columns Operators Aggregates Functions

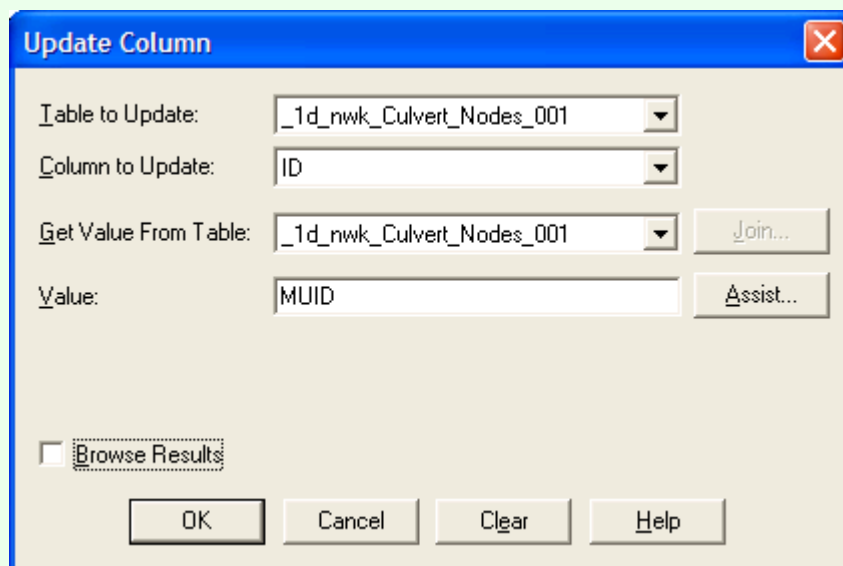
Save Template Load Template

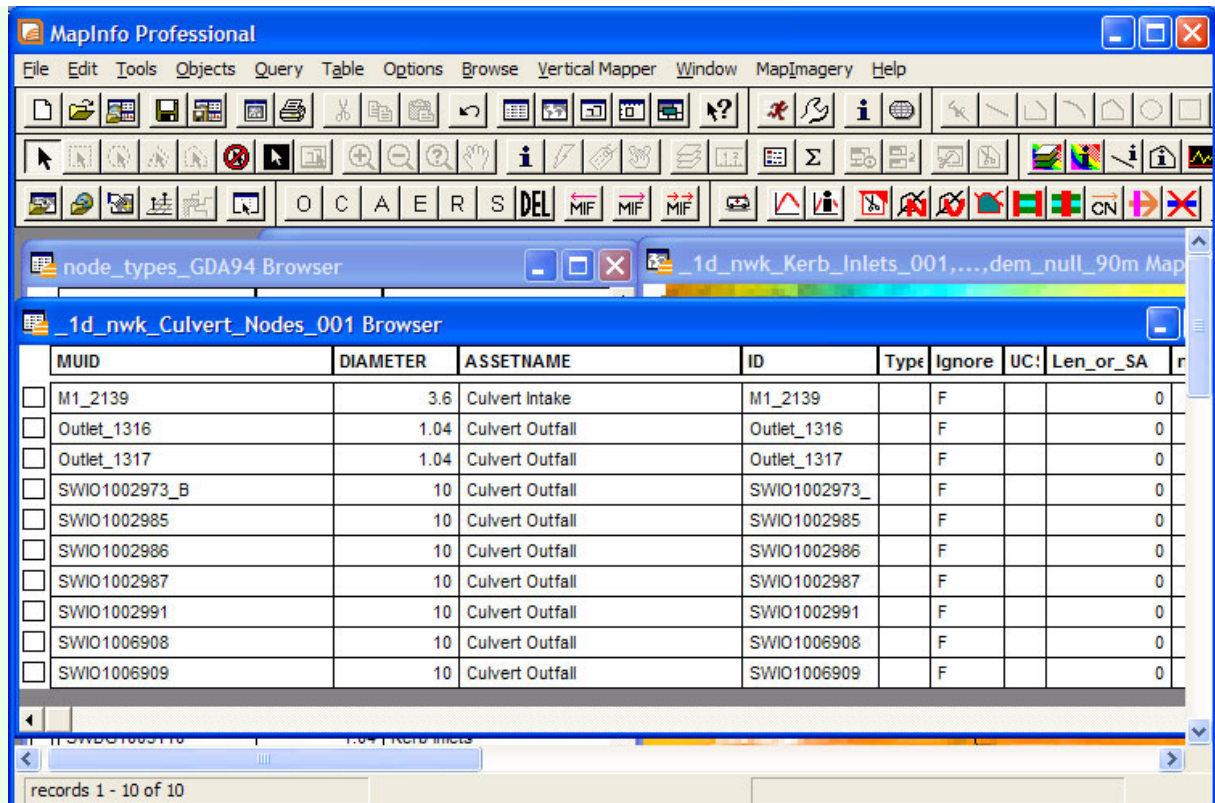
- 3 The image below shows the result of the SQL Select. As can be seen, any object with “culvert” in the attribute ASSETNAME is selected. The virtual Culvert_Nodes layer was appended to an empty 1d_nwk layer previously created. This adds the selected objects and retains the 1d_nwk attributes.





- 4 The 1d_nwk ID was set to the asset database ID attribute (MUID) using the Update Column dialog below.





MUID	DIAMETER	ASSETNAME	ID	Type	Ignore	UC!	Len_or_SA	r
☐ M1_2139	3.6	Culvert Intake	M1_2139		F		0	
☐ Outlet_1316	1.04	Culvert Outfall	Outlet_1316		F		0	
☐ Outlet_1317	1.04	Culvert Outfall	Outlet_1317		F		0	
☐ SWIO1002973_B	10	Culvert Outfall	SWIO1002973_		F		0	
☐ SWIO1002985	10	Culvert Outfall	SWIO1002985		F		0	
☐ SWIO1002986	10	Culvert Outfall	SWIO1002986		F		0	
☐ SWIO1002987	10	Culvert Outfall	SWIO1002987		F		0	
☐ SWIO1002991	10	Culvert Outfall	SWIO1002991		F		0	
☐ SWIO1006908	10	Culvert Outfall	SWIO1006908		F		0	
☐ SWIO1006909	10	Culvert Outfall	SWIO1006909		F		0	

- Other 1d_nwk attributes were updated as needed and the three asset database attributes at the beginning of the 1d_nwk layer list of attributes were moved to the end of the list using the Table, Maintenance, Table Structure... dialog. TUFLOW only reads the first 20 attributes of a 1d_nwk layer, so the three attributes can be kept within the 1d_nwk layer by placing them at the end of the twenty TUFLOW attributes (Note that if there are more than nine extra attributes, TUFLOW interprets this as a 1d_nwke layer, so the 1d_nwk needs to be converted to a 1d_nwke and the extra attributes attached to this layer - see Table 4.15).

The polylines and attribute data for the pipes were copied into a 1d_nwk layer using a similar approach as described above. A problem arose where the length of the ID text exceeded 10 characters. Whilst the ID attribute can be up to 12 characters, when TUFLOW automatically creates nodes at any channel end that has no point object snapped to it, TUFLOW appends the characters “.1” or “.2” at the end of the channel ID so as to provide the created node with an ID. If the channel ID is 11 characters or longer, TUFLOW truncates the channel ID to ten characters and adds the “.1” or “.2”. When the model was first run, the truncation to 10 characters caused a number of created nodes to have identical IDs, and an ERROR occurred. To solve the problem the “SW” was removed from the beginning of the pipe IDs so as to create unique IDs of 10 characters or less. The following steps were carried out to achieve this.

- Using the SQL Select dialog shown below with the LCase\$() and Left\$() string functions, all pipes with an ID starting with “SW” were selected.

SQL Select

Select Columns: *

from Tables: _1d_nwk_pipes_001

where Condition: Left\$(LCASE\$(ID), 2) = "sw"

Group by Columns:

Order by Columns:

into Table Named: Selection

☐ Browse Results ☐ Find Results In Current Map Window

OK Cancel Clear Verify Help

Tables Columns Operators Aggregates Functions

Save Template Load Template

- The "SW" was removed from the ID using the Update Column dialog with the Mid\$() as shown below, and illustrated in the screen shots further below, which show before and after the dialog below was applied.

Update Column

Table to Update: Selection

Column to Update: ID

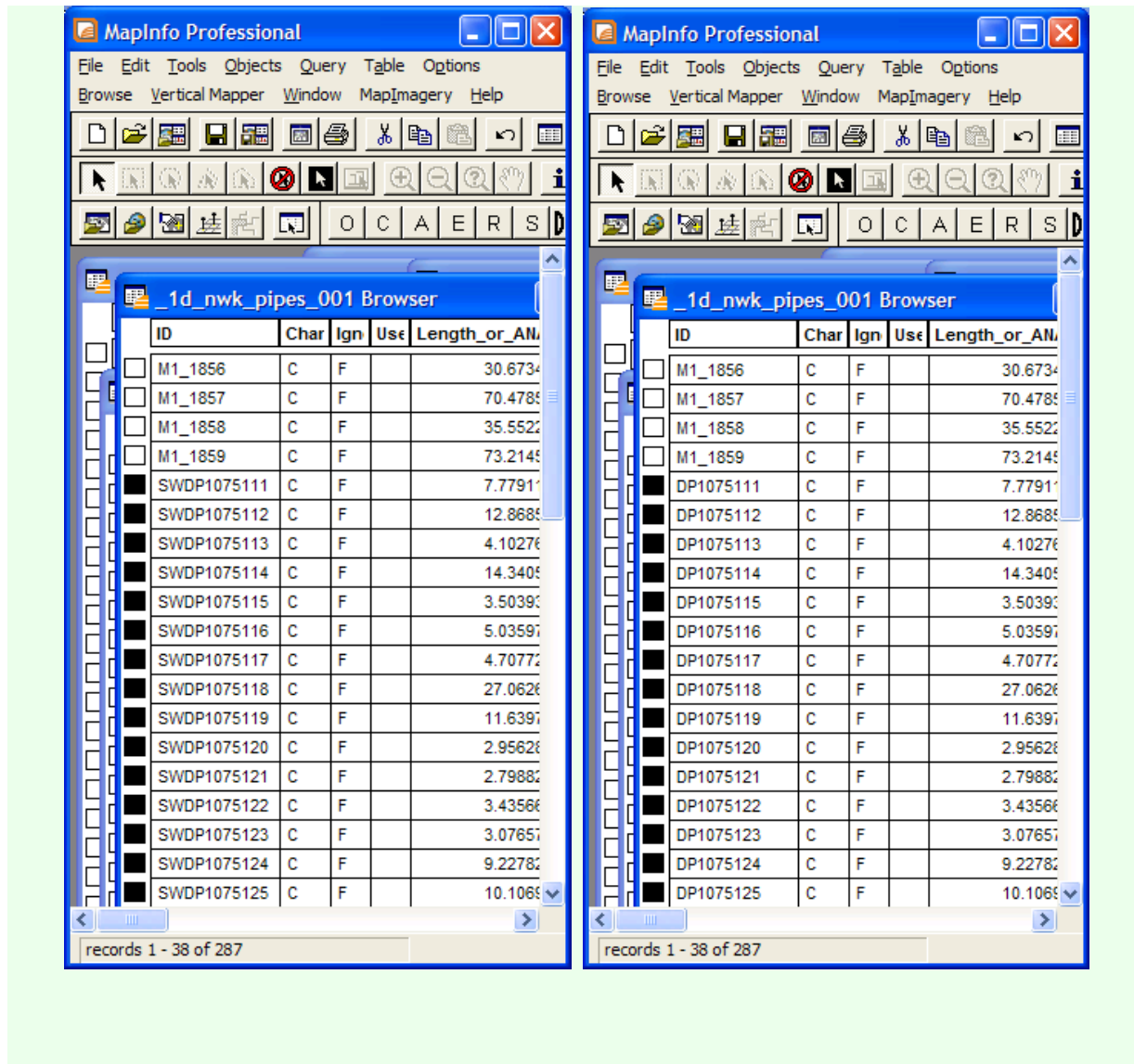
Get Value From Table: Selection

Value: mid\$(ID,3,12)

☐ Browse Results

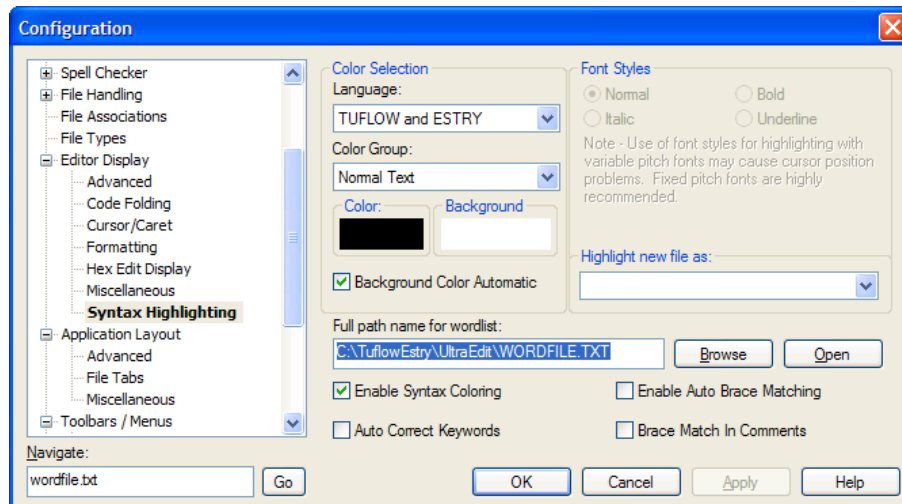
OK Cancel Clear Help

Join... Assist...



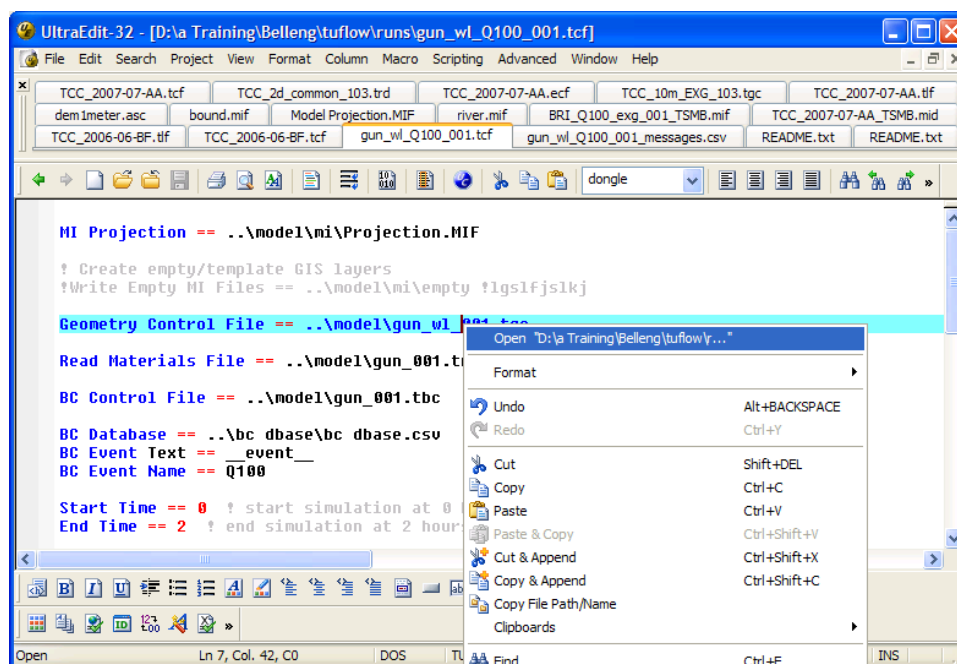
12.2 UltraEdit Tips

- 1 Use the automatic colour coding of the control files by downloading the “WORDFILE.TXT” file available at www.tuflow.com/Downloads_Misc.htm and replacing the file with the same name in the C:\Program Files\UltraEdit folder. Alternatively in later versions of UltraEdit, a path can be specified to the WORDFILE.TXT file as shown in the UltraEdit Configuration dialogue below:



- 2 Easily navigate between control and other text files by right clicking on the file in the active text file and selecting the top option which should be to open the file you have right clicked on (see image below). If this does not work, usually because there are spaces in the filename, select the file (including any “..\.” characters) and try again (this should always work).

A good tip is always start with the .tcf file and open other files by right clicking on them as this ensures that the correct file has been opened.



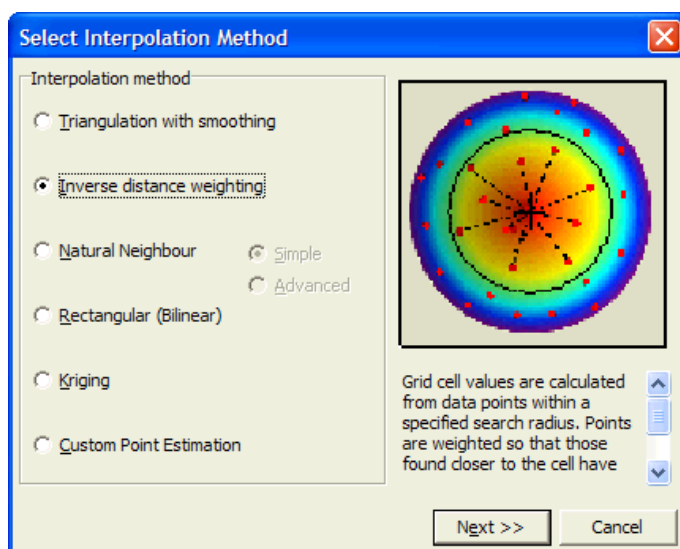
12.3 Vertical Mapper Tips

12.3.1 Creating High Quality Flood Extent Maps

Whilst 1D/2D model output may appear impressive, jagged and blocky edges due to the 2D cell resolutions and the spacing of 1D WLLs and WLL Points may occur when zooming in to the edge of the flood extent. Also, the elevations in the SMS .dat files are based on the elevations of the ZH Zpts and WLL Points, so are not on as fine a resolution as that of the DTM.

The procedure below is one way to produce high quality flood extent maps at a similar resolution to that of the underlying DTM. The process below is based on that for using the Vertical Mapper software, but can be applied in similar manner using other software.

- 1 Using TUFLOW_to_GIS create a grid of points of the peak water level. Choose a -grid spacing of around half the smallest 2D domain cell size. For example:
`TUFLOW_to_GIS.exe -mif -t999999 -grid2 my_model_h.dat`
- 2 Import the _h_Max.mif created by the previous step into MapInfo or other GIS software. The above would have created a file my_model_h_g002_Max.mif.
- 3 Using Vertical Mapper or other 3D grid software create a IDW surface. The instructions for using Vertical Mapper are:
 - (a) Go to the dialogue below using Vertical Mapper, Create grid..., Interpolation and choose the Inverse distance weighting option. Click Next>>.



- (b) Select the _h_Max table, the Value attribute and Unit Type Meters as per the below and click Next >>:

- (c) Choose a cell size (typically the same as the -grid distance). Set the Display radius to the distance which the 3D flood surface will be extended beyond the edge of the flood extent. The objective here is to extend the flood extent into dry ground. Usually a distance of 2 to 5 times the 2D cell size suffices unless the terrain is very flat in which case a greater distance maybe required. Set the Search radius to the same as the Display radius.

Importantly set Maximum # of points to a small number (say 4 – this will only use the four closest points to determine the water surface elevation at a grid cell). The larger the number, the greater the number of points that are used to determine the water surface elevation at a grid cell, and therefore, the more smoothing of the surface. Excessive smoothing of the surface can be problematic where there is a sudden drop in flood levels such as across a road embankment or levee. Edit the File name as desired and click Finish.

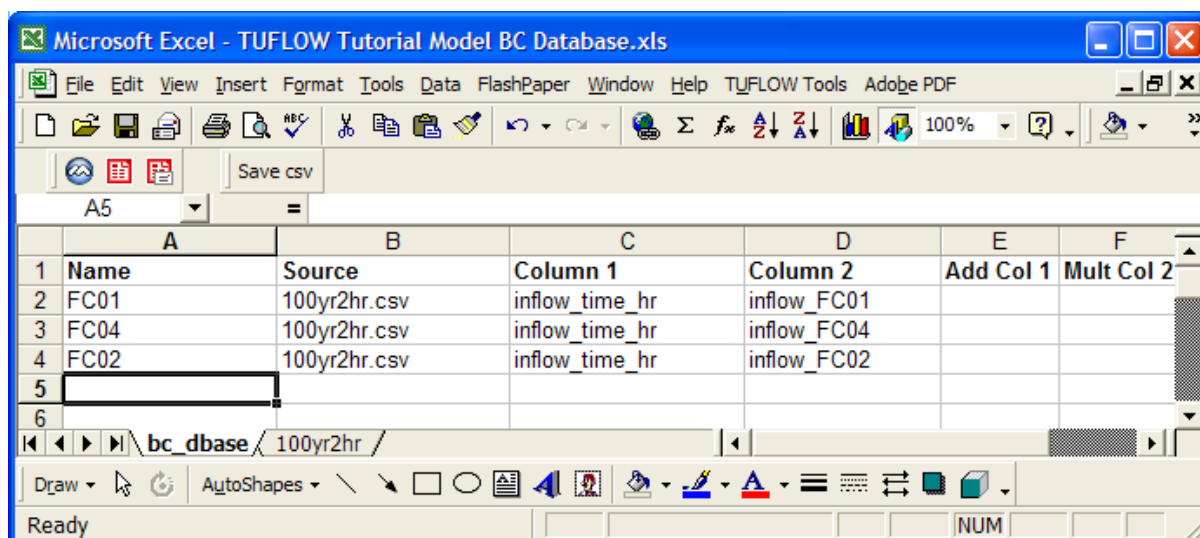
- (d) The resulting 3D water surface should extend beyond the _h_Max.mif layer as shown below.
- (e) Using Vertical Mapper's grid calculation tool subtract the original DTM grid from the 3D water surface grid to produce a 3D depth grid. To produce high quality flood extent and depth maps contour the 3D depth grid ensuring to start your contouring interval at zero (0) as negative depths will occur where the 3D flood surface has penetrated the original DTM. The zero contour will be the flood extent, the spatial quality of which will mimic the spatial resolution of the original DTM (rather than the 2D cell resolution which is usually coarser).

12.4 Microsoft Excel Tips

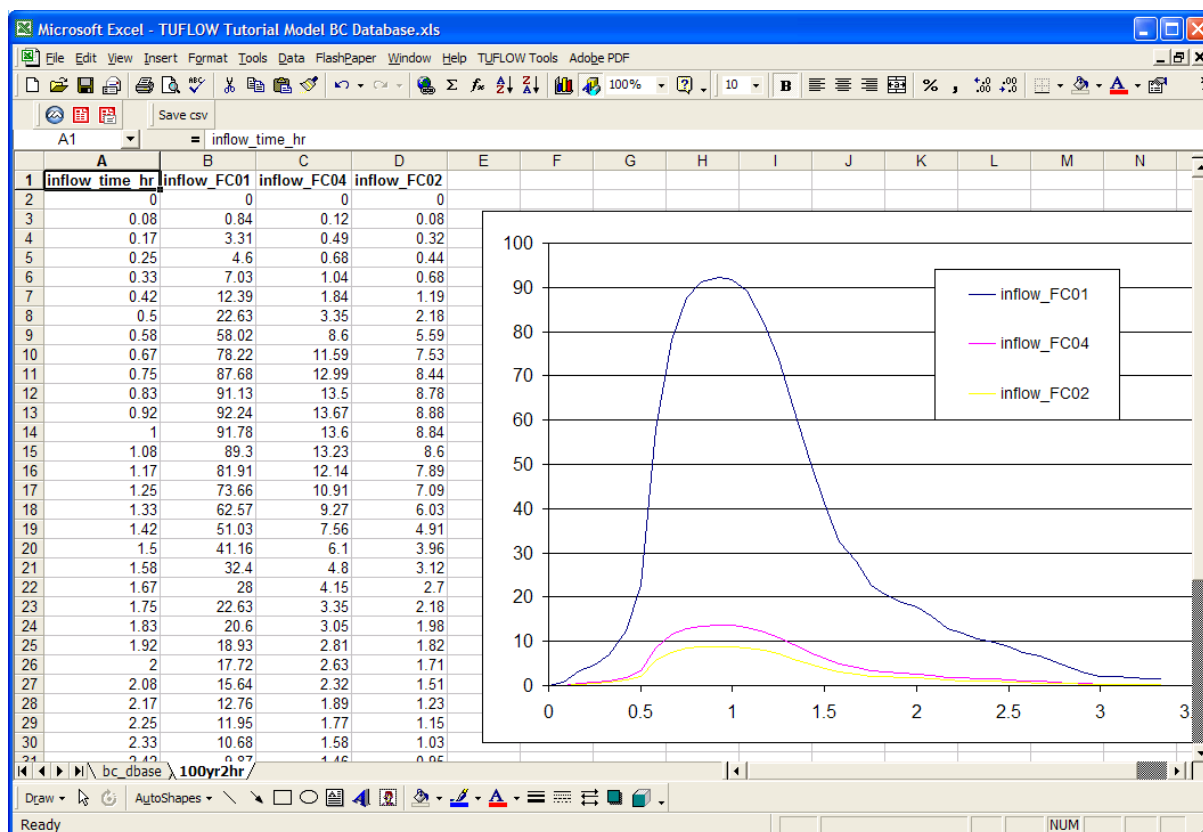
12.4.1 Save csv and Using .xls File to Manage Databases

The recommended approach for managing boundary, pit inlet, cross-section and other tabular databases is to manage all of the data in a Microsoft Excel .xls file and use the TUFLOW Tools.xls “Save csv” button to export the tabular data to the .csv format used by TUFLOW. To access and set up the Save csv macro see Section 11.9.

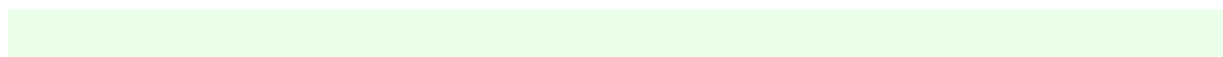
The image below shows the sheet used by the TUFLOW Tutorial Model to create the .csv file for setting the [BC Database](#).



The second sheet, which is shown below, contains the time series data sourced by the first sheet. To export this data to .csv format, click on the Save csv button. Charts or other graphics are not exported, but can be placed on the sheet to help view the data.



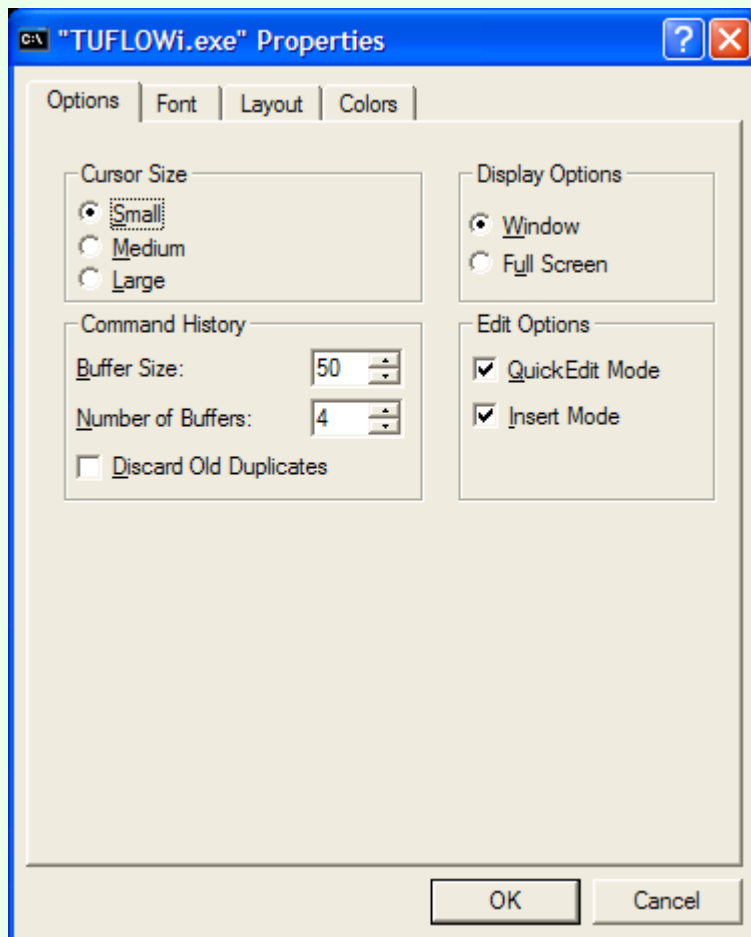
A similar example for a [Pit Inlet Database](#) is presented in Section 4.5.1.2.



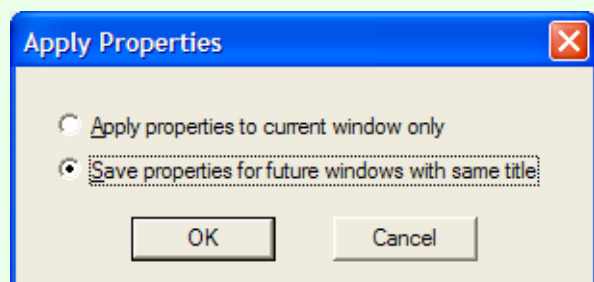
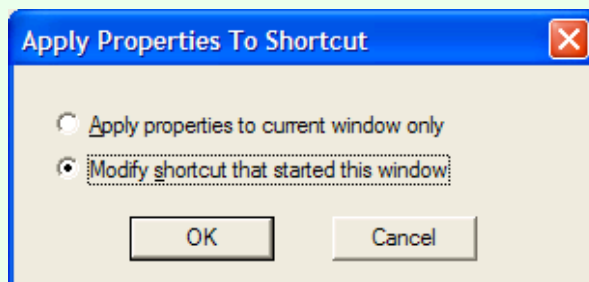
12.5 DOS Window Tips

12.5.1 Copying and Pasting in a Console (DOS) Window

To copy and paste into/from the TUFLOW Console Window the window must be set up to be in Quick Edit mode. To check or set this up right click on the title bar of the Console Window and select Properties to bring up the dialog below. Tick Quick Edit Mode and press OK.

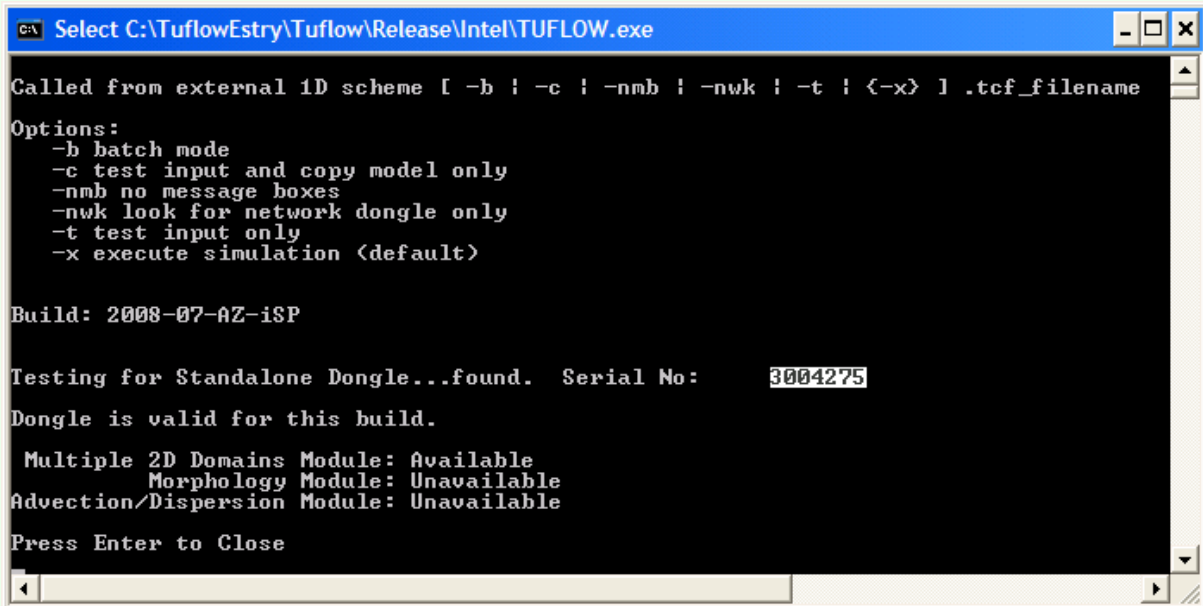


One of the dialogs below should appear. Select the bottom option in either case so that the Quick Edit setting is saved for future windows.



To copy text from the Console Window onto the Clipboard, select the text by dragging the cursor with the left mouse button held down. The image below shows the serial number being selected. Click the right mouse button and the text is placed on the Clipboard for use in other applications. To paste text

into a Console Window, copy the text to the Clipboard, go to the Console Window and right click the right mouse button and the text is pasted into the window.



```
C:\ Select C:\TuflowEstry\Tuflow\Release\Intel\TUFLOW.exe

Called from external 1D scheme [ -b ! -c ! -nmb ! -nw ! -t ! {-x} ] .tcf_filename

Options:
-b batch mode
-c test input and copy model only
-nmb no message boxes
-nw look for network dongle only
-t test input only
-x execute simulation (default)

Build: 2008-07-AZ-iSP

Testing for Standalone Dongle...found. Serial No: 3004275
Dongle is valid for this build.

Multiple 2D Domains Module: Available
Morphology Module: Unavailable
Advection/Dispersion Module: Unavailable

Press Enter to Close
```

13 References

Many of the references below can be downloaded from the Publications Downloads Page at www.tuflow.com.

Barton, C.L. (2001) *Flow Through an Abrupt Constriction – 2D Hydrodynamic Model Performance and Influence of Spatial Resolution* Thesis submitted as partial fulfilment for Master of Engineering Science, Environmental Engineering, Griffith University, July 2001.

Benham, S.A., Rogencamp, G.J. (2003) *Application of 2D Flood Models with 1D Drainage Elements* Flood Mitigation Conference, Forbes, 2003.

Charteris, A.B., Syme, W.J. (2001) *Urban Flood Modelling and Mapping – 2D or not 2D* Conference on Hydraulics in Civil Engineering, Hobart, November 2001.

Chow, V.T. (1959) *Open Channel Hydraulics* McGraw-Hill.

FHA (1973) U.S. Department of Transportation, Federal Highway Administration (US FHA 1973) *Hydraulics of Bridge Waterways* Hydraulic Design Series No. 1, Second Edition.

Henderson, F.M. (1966) *Open Channel Flow* Macmillan Publishing Co., Inc, 1966.

Huxley, C.D. (2004) *TUFLOW Testing and Validation* Undergraduate Thesis for Bachelor of Engineering in Environmental Engineering, School of Environmental Engineering, Griffith University, June 2004.

DHI (2009) *MOUSE Pipe Flow Reference Manual, 2009*. DHI Group. Agern Allé 5, DK-2970 Hørsholm, Denmark

Stelling, G.S. (1984) *On the Construction of Computational Methods for Shallow Water Flow Problems* Rijkswaterstaat Communications, No. 35/1984, The Hague, The Netherlands.

Syme W.J. (1989) *Computer Graphic Techniques - An Essential Tool for Interpreting and Analysing a Dynamic Flow Model* Watercomp '89 Melbourne, Australia, 1989.

Syme W.J., McColm G.A. (1990) *Integration of Numerical Flood Modelling into Geographic Information Systems* Conference on Hydraulics in Civil Engineering Sydney, Australia, 1990.

Syme W.J., Apelt C. (1990) *Linked 2-D/1-D Flow Modelling using the Shallow Water Equations* Conference on Hydraulics in Civil Engineering Sydney, Australia, 1990.

Syme W.J. (1990) *Practical 1-D and 2-D Computer Modelling of Flow in Coastal Waters and Estuaries* Technical Paper I.E. Aust. Qld Division, 1990.

Syme W.J. (1990) *Computer Modelling of Flow and Transport Processes. A Powerful Environmental Management Tool for Coastal Waters* Engineering in Coral Reef Regions Conference Townsville, Australia, 1990.

- Syme, W.J. (1991)** *Dynamically Linked Two-Dimensional / One-Dimensional Hydrodynamic Modelling Program for Rivers, Estuaries & Coastal Waters* William Syme, M.Eng.Sc (100% Research) Thesis, Dept of Civil Engineering, The University of Queensland, May 1991.
- Syme W.J., Barnett A.G., Turton G.B. (1992)** *GIS Floodplain Management* ITMG Conference New Zealand, 1992.
- Syme W.J., Paudyal G.N. (1994)** *Bangladesh Flood Management Model* 2nd International Conference on River Flood Hydraulics York, UK, 1994.
- Syme, W.J., Nielsen, C.F., Charteris, A.B. (1998)** *Comparison of Two-Dimensional Hydrodynamic Modelling Systems Part One - Flow Through a Constriction* International Conference on Hydraulics in Civil Engineering, Adelaide, September 1998.
- Syme W.J., Rogencamp G.J., Nielsen C.F. (1999)** *Two-Dimensional Modelling of Floodplains – A Powerful Floodplain Management Tool* NSW Flood Mitigation Conference, Tamworth, NSW, 1999.
- Syme W.J. (2000)** *Pros and Cons of One-dimensional and Two-Dimensional Modelling of Floodplains* Queensland Hydrology Symposium, Brisbane, Qld, 2000.
- Syme W.J. (2001a)** *TUFLOW – Two & one-dimensional Unsteady FLOW Software for Rivers, Estuaries and Coastal Waters* IEAust Water Panel Seminar and Workshop on 2D Flood Modelling, Guest Speaker, Sydney, February 2001.
- Syme, W.J. (2001b)** *Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme* Conference on Hydraulics in Civil Engineering, Hobart, November 2001.
- Syme W.J. (2006)** *Bruce Highway Eudlo Creek Hydraulic Investigations - A Turning Point* 30th Annual Conference of the Association of State Floodplain Managers, June 11–16, 2006, Albuquerque, New Mexico
- Syme W.J. (2008)** *Flooding in Urban Areas - 2D Modelling Approaches for Buildings and Fences* Engineers Australia, 9th National Conference on Hydraulics in Water Engineering, Darwin Convention Centre, Australia 23-26 September 2008.
- Syme W.J., Jones R, Arneson, L (2009)** *Two-Dimensional Flow Modeling of Hydraulic Structures in a 2D ADI Scheme* 33rd IAHR Congress, Vancouver, 2009
- Tullis B.P., Robinson, S.C. (2008)** *Quantifying Culvert Exit Loss* Journal of Irrigation and Drainage Engineering, 134(2):263-266, 2008, ISSN: 07339437.
- U.S. Department of Commerce, Bureau of Public Roads (US BPR 1965)** *Hydraulic Charts for the Selection of Highway Culverts and Capacity Charts for the Hydraulic Design of Highway Culverts* Hydraulic Engineering Circulars Nos. 5 and 10.

Appendix A .tcf File Commands

Adjust Head at Estry Interface	Extrapolate Heads at Flow Boundaries	Model TUFLOW Build
Apply Wave Radiation Stresses		Model TUFLOW Release
Apply Wind Stresses		
BC Control File	First Sweep Direction	Negative Depth Approach
BC Database	Free Overfall	Null Cell Checks
BC Event Name	Free Overfall Factor	Number Iterations
BC Event Source	Froude Check	Number 2D2D Link Iterations
BC Event Text	Froude Depth Adjustment	
BC Wet/Dry Method	Geometry Control File	Oblique Boundary Alignment
BC Zero Flow	Global FC Ch Factor	Oblique Boundary Method
Bed Resistance Cell Sides	Global Weir Factor	Output Drive
Bed Resistance Depth Interpolation		Output Folder
Bed Resistance Values	HQ Boundary Approach	Rainfall Boundaries
Boundary Cell Selection	HX ZC Check	Rainfall Gauges
		Recalculate Chezy Interval
Calibration Points MI File	If Scenario	Read File
Cell Wet/Dry Depth	Input Drive	Read Materials File
Cell Side Wet/Dry Depth	Inside Region	Read GIS Cyclone
Cell Size	Instability Water Level	Read GIS Hurricane
Change Zero Material Values to One	Latitude	Read GIS ISIS Network
Check Inside Grid	Line Cell Selection	Read GIS ISIS Nodes
Check MI Save Date	Link 2D2D Approach	Read GIS ISIS WLL
Check MI Save Ext	Log Folder	Read GIS ISIS WLL Points
CSV Time		Read GIS FC
	Map Cutoff Depth	Read GIS GLO
Defaults	Map Output Data Types	Read GIS IWL
Density of Air	Map Output Format	Read RowCol IWL
Density of Water	Map Output Interval	Read GIS LP
Depth/Ripple Height Factor Limit	Mass Balance Output	Read GIS PO
Display Water Level	Mass Balance Output Interval	Read GIS XP Network
Distribute HX Flows	Maximum Points	Read GIS XP Nodes
Double Precision	Maximum Velocity Cutoff Depth	Read GIS XP WLL
	Maximum Vertices	Read GIS XP WLL Points
End 1D Domain	Meshparts	Read Restart File
End 2D Domain	MI Projection	
End Time	MI Projection Check	Screen/Log Display Interval
ESTRY Control File	Model Events	
Event File	Model Platform	Set Route Cut Off Type
Excel Start Date	Model Precision	Set Route Cut Off Values
	Model Scenarios	Set IWL
		Snap Tolerance
		Shallow Depth Stability Factor

Shallow Depth Weir Factor Cut Off Depth	Time Series Output Interval	Water Level Checks
Shallow Depth Weir Factor Multiplier	Time Output Cutoff Depths	Wave Period
Simulations Log Folder	Timestep	Wetting and Drying
Start 1D Domain	Timestep During Warmup	Wind Output Interval
Start 2D Domain	UK Hazard Debris Factor	Wind/Wave Shallow Depths
Start Map Output	UK Hazard Formula	Write Check Files
Start Time	UK Hazard Land Use	Write Empty MI Files
Start Time Series Output	Unused HX and SX Connections	Write PO Online
Start Wind Output at Time	Verbose	Write Restart File at Time
Store Maximums and Minimums	Viscosity Coefficient	Write Restart File Interval
Supercritical	Viscosity Formulation	Write X1D Check Files
SX Head Adjustment	VG Z Adjustment	XF Files
SX ZC Check	Warmup Time	Zero Negative Depths in SMS

A.1 Geographic Reference Commands (.tcf)

Latitude == [{0} <value_in_degrees_from_equator>]	A-1
MI Projection == [<.mif file> <Projection_line_from_MIF_file>]	A-1
MI Projection Check == [{ERROR} WARNING]	A-2
Snap Tolerance == [{0.001} <value_in_metres>]	A-2

Latitude == [{0} | <value_in_degrees_from_equator>]
(Optional)

Sets the latitude used for calculating the Coriolis term in the shallow water equations. Negative value indicates south of the equator. A zero value disables the Coriolis term.

**MI Projection == [<.mif file> |
<Projection_line_from_MIF_file>]**
(Optional but recommended)

Sets the geographic projection for all GIS input and output. If this command is omitted, TUFLOW searches for a file “Header.mif” in each folder it opens GIS files, and extracts the projection from this file. The “Header.mif” file is any GIS layer in the correct projection exported in MIF/MID format. If no “Header.mif” file is found, non-earth coordinates are assumed.

Note: The projection must be Cartesian and in meters.

As of Build 2003-12-AD, a mif file can be specified and the projection line is extracted from this file, and is the referred approach than that described below.

To enter a projection line from a mif file, follow these steps:

- 1 In a GIS, create or open a layer in the Cartesian projection to be used for the model. For non-geographic models (eg. a test model), use the Non-Earth (meters) projection.
- 2 Export the layer in MIF/MID format.
- 3 Open the .mif file in a text editor, copy the whole line starting with “CoordSys” (usually the 4th or 5th line) and paste after “MI Projection ==” in the .tcf file.

Examples:

```
MI Projection == ..\model\mi\Model_Projection.mif
MI Projection == CoordSys Earth Projection 8, 13, "m", 153, 0,
0.9996, 500000, 10000000 Bounds (-7745874.38492, 1999.40969607)
(8745874.38492, 19998000.5903)
```

Note: All GIS layers read by TUFLOW MUST USE this projection. The projection must be a Cartesian based projection, not a spherical projection such as Latitude/Longitude.

MI Projection Check == [{ERROR} | WARNING]

(Optional)

Checks that the [MI Projection](#) setting is the same as the projection for all input layers. The Coordsys line check removes all spaces, tabs and quotes when making the comparison.

The check includes the “Bounds” values (as of Build 2007-07-AD), as having different bounds can affect the decimal precision used by GIS/CAD software when writing .mif files, which can affect the TUFLOW test for snapped (connected) objects. A bug was that caused MI Projection Check to sometimes fail was also fixed in Build 2007-07-AD.

If [Defaults](#) == Pre 2007-07-AA is set, the default setting is WARNING.

Snap Tolerance == [{0.001} | <value_in_metres>]

(Optional)

Sets the search distance in metres for detecting whether two GIS objects are connected (snapped). Introduced in Build 2005-05-AN.

A.2 File Management Commands (.tcf)

BC Control File == <.tbc_file>	A-3
Check MI Save Date == [{ERROR} WARNING OFF]	A-4
Check MI Save Ext == [{.tab} <ext>]	A-4
End 1D Domain	A-4
End 2D Domain	A-4
ESTRY Control File [{} AUTO] == <.ecf_file>	A-4
Geometry Control File == <.tgc_file>	A-5
If Scenario == <s1> <s2> <s3>... [Else If Scenario == <s1> <s2> <s3>...] [Else] End If	A-5
Log Folder == <folder>	A-5
Model Scenarios == <s1> <s2> <s3>...	A-5
Output Folder == <folder>	A-6
Read File == <file>	A-6
Simulations Log Folder == <folder>	A-6
Start 1D Domain	A-6
Start 2D Domain == [{} <2d_domain_name>]	A-7
Write Check Files == [<file_prefix> {OFF}]	A-7
Write Empty MI Files == [{} <folder>]	A-8
XF Files == [{ON} OFF]	A-8

BC Control File == <.tbc_file>

(Mandatory for carrying out a simulation - can be left out when developing the .tgc file)

Specifies the boundary control, .tbc, file (see Section D.1). There can only be one .tbc file.

Check MI Save Date == [{ERROR} | WARNING | OFF]

(Optional)

Checks that the save date of the .mid file is later than the save date of the GIS layer as defined by [Check MI Save Ext](#). The two files must be located in the same folder. This command is very useful for detecting the possibility that a GIS layer has been modified, but not exported as .mif/.mid files prior to the simulation.

For the ERROR option (the default), the simulation terminates and an error message is given.

For the WARNING option, a warning is written to the screen and log file, but the simulation proceeds without pausing. It remains the responsibility of the user to check for any warnings.

Prior to Build 2007-07-AA, setting to WARNING caused an infinite loop to occur (workaround is to set to OFF).

The OFF option disables all checks and no warnings are given.

Check MI Save Ext == [{.tab} | <ext>]

(Optional)

Sets the extension of the GIS file for which [Check MI Save Date](#) uses. The default extension is “.tab”; the MapInfo primary GIS table file.

End 1D Domain

(Optional)

Terminates a [Start 1D Domain](#) block of 1D (.ecf) commands in a .tcf file.

End 2D Domain

(Mandatory if more than one 2D domain)

Indicates the end of a block of commands that define a 2D domain. Must only occur after a [Start 2D Domain](#) command, otherwise an error occurs.

ESTRY Control File [{} | AUTO] == <.ecf_file>

(Mandatory if linking to an ESTRY 1D model)

Specifies the ESTRY control, .ecf, file (see Section 4.2.2). There can only be one .ecf file.

Note: As of Build 2008-08-AA, all 1D output filenames are now based on the .tcf filename, not the .ecf filename. This means that if the .ecf file does not change when setting up a new simulation based on a previous simulation, there is no need to make a copy of the .ecf file (ie. the .ecf file can be treated in a similar manner to the .tgc and .tbc files).

The AUTO option automatically sets the .ecf filename to the same as the .tcf file (except for the extension).

Note: Prior to Build 2008-08-AA, it is strongly recommended that the .tcf and .ecf files have the same name, which is enforced if using the AUTO option. If the AUTO option is used, <.ecf_file> is left blank if the .ecf and .tcf files are in the same folder, or is used to specify the folder in which the .ecf file is located.

Geometry Control File == <.tgc_file>

(Mandatory)

Specifies the geometry control, .tgc, file (see Section 4.3). There can only be one .tgc file.

A .tgf file (TUFLOW's original binary formatted geometry file) can also be specified for earlier version models.

**If Scenario == <s1> | <s2> | <s3>...
[Else If Scenario == <s1> | <s2> | <s3>...]
[Else]
End If**

(Optional)

Controls which commands to process for different scenarios as specified using the -s run option (see Table 5.2) or [Model Scenarios](#).

The "If Scenario" block must be terminated by "End If" otherwise an ERROR occurs.

Optional "Else If Scenario ==" and "Else" blocks can be embedded between "If Scenario" and "End If". The first block encountered that refers to a specified scenario is processed, and all remaining blocks within the "If Scenario" to "End If" construct are ignored. If an "Else" block is used it must occur as the last block (ie. must occur after any "Else If Scenario" blocks) and its commands are only processed if no previous blocks have been processed.

Log Folder == <folder>

(Optional)

Redirects the .tlf and _messages.mif file output to the specified folder. Typically used to write these files to a folder named log under the runs folder.

Model Scenarios == <s1> | <s2> | <s3>...

(Optional)

Alternative to using the -s option when running TUFLOW (see Table 5.2). Up to nine (9) separate scenarios that occur using [If Scenario](#) can be specified. Separate the scenarios names using vertical bars. The scenario names specified will be automatically appended to the output filenames if ~s~ or ~sX~ are not in the .tcf filename.

For example:

Model Scenarios == opA | opB

Note that using the `-s run` option will override this command.

Output Folder == <folder>

(Optional)

Redirects all TUFLOW output data except the .tlf file to another folder. Typically used to write output to your local C: or D: drive instead of filling up the network or to keep results separate to the input data. A URL can be used (eg. <\\wbmserv\Computer001\tuflow\results>); useful for running simulations on other computers, but with the output directed to your local drive or other location (your drive will need to be shared).

Read File == <file>

(Optional)

Directs input to another file. When finished reading <file>, TUFLOW returns to reading the .tcf file.

This command is particularly useful for projects with a large number of simulations. Repetitive commands are grouped and placed in another text file. If one of these commands changes, the command only has to be edited once, rather than in every .tcf file.

Also available in .tgc and .ecf files.

NOTE: As of Build 2002-03-AA, this command can now be used in redirected file(s) up to a maximum of ten levels.

Simulations Log Folder == <folder>

(Optional)

Sets the path to a folder path or URL on the LAN to log all simulations to a file “_ All TUFLOW Simulations.log” (for releases prior to 2010-10, or builds prior to the “DA” builds a slightly different filename was used).

This command takes precedence over the `-slp` option (see Table 5.2) and the same command in the “C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf” file. It is recommended that this command not be used in lieu of these other options as it is simulation specific.

If the keywords “DO NOT USE” occur within the folder path or URL, this feature is disabled.

Start 1D Domain

(Optional)

Indicates the start of a block of 1D or .ecf commands in the .tcf file. Block must be terminated by using [End 1D Domain](#).

Start 2D Domain == [{} | <2d_domain_name>]

(Mandatory if more than one 2D domain)

Indicates the start of a block of commands that define a 2D domain. If no 2d_domain_name is specified, the 2D domain is automatically assigned a name. The name is solely used for reporting in the .tlf file and elsewhere. Also see [End 2D Domain](#) and Section 4.4.13.

If there is only one 2D domain, this command is optional.

Write Check Files == [<file_prefix> | {OFF}]

(Optional)

Creates GIS check files in MIF/MID format and text .csv files for quality control checking of model input data. Prior to Build 2002-11-AA, “Write MI Check Files” was used and may continue to be used for backward compatibility. Some of files produced are noted below (see Section 7.2.5 for more details).

- Of the entire grid after all boundary conditions, flow constrictions, etc have been applied. The mif/mid files have a “_grd” appended to their names.
- Of all the Z-points after all geometry adjustments including flow constrictions, etc. The mif/mid files have a “_zpt” appended to their names.
- Plot output (“_PO”), longitudinal output (“_LP”) and flow constriction (“_FC”) details. These files can be modified and used as direct input (see Sections 4.7.3 and 4.5). If no PO, LP or FC’s exist an empty GIS layer is created.
- *_2d_bc_tables_check.csv file containing any tables read by 2d_bc layers.

If <file_prefix> is omitted or ends in a “\” to indicate a folder, the .tcf filename is used (without the .tcf extension). <file_prefix> can include a folder path that is normally set to the check folder. See the examples below for this subtle difference.

The OFF option deactivates any previously specified Write Check Files command - no check files will be created. If the command is never specified, the OFF option applies.

Examples:

Write Check Files == C:\jb9999\tuflow\check\2d ! writes check files to the folder “C:\jb9999\tuflow\check” and prefixes with “2d”

Write Check Files == C:\jb9999\tuflow\check\ ! writes check files to the folder “C:\jb9999\tuflow\check” and prefixes with the .tcf filename

Write Check Files == C:\jb9999\tuflow\check ! writes check files to the folder “C:\jb9999\tuflow” and prefixes with “check”

Write Empty MI Files == [{} | <folder>]

(Optional)

Creates empty 1D and 2D GIS files in MIF/MID format useful for setting up new GIS layers. Each layer as described in Table 2.3 is produced with the required attribute definitions pre-defined, but containing no geographic objects. Provided the [MI Projection](#) command has been previously specified, each layer has the correct GIS projection.

The layers are prefixed using the prefixes defined in Table 2.3 and are given a suffix of “_empty”. If <folder> is specified, the .mif/.mid files are located in the folder, which must already exist.

After writing the files, TUFLOW stops executing.

Example:

```
Write Empty MI Files == ..\model\mi\empty
```

XF Files == [{ON} | OFF]

(Optional)

Sets the global default for whether or not to automatically generate XF files. See Section 4.13.2.

A.3 Simulation Time Control Commands (.tcf)

End Time == <time_in_hours>	A-9
Number Iterations == [{2} <no_iterations>]	A-9
Number 2D2D Link Iterations == [{1} <no_iterations>]	A-10
Start Time == <time_in_hours>	A-10
Timestep == [{1.0} <timestep_in_seconds>]	A-10
Timestep During Warmup == [{timestep} <warmup_timestep_in_seconds>]	A-10
Warmup Time == [{0} <warmup_time_in_hours>]	A-10

End Time == <time_in_hours>

(Mandatory)

Specifies the finish time of the simulation in hours. Value must be greater than the start time and can be negative.

Number Iterations == [{2} | <no_iterations>]

(Optional)

Specifies the number of iterations per timestep (refer to Stelling (1984) or Syme (1991)). It is rare this value is changed from 2, the default. Doubling the number of iterations slows down the simulation by roughly a factor of two. If a value of less than 2 is specified, 2 is used.

The main exception to the default value of 2 is in the modelling of rapidly varying flood waves such as dambreak hydrographs. For simulations of this kind, improved convergence and reduced mass error can be achieved through increasing the number of iterations up to values as high as 10, although usually 4 should suffice. The recommended approach is if the simulation is demonstrating unacceptably high mass error (eg. >3%), carry out sensitivity tests that increase the number of iterations to ascertain whether a poor convergence using the default value of 2 is the cause of the problem. If increasing the number of iterations has no benefits keep the value at the default value of 2.

Number 2D2D Link Iterations == [{1} | <no_iterations>]

(Optional)

Specifies the number of iterations for setting water levels at control points along a 2D line in a 2d_bc layer for stitching 2D domains together. As of Build 2006-06-AA, this feature is still under development and is likely to be subject change.

Start Time == <time_in_hours>

(Mandatory)

Specifies the start time of the simulation in hours. Value can be negative and it is recommended that it be relative to midnight for historical events.

Timestep == [{1.0} | <timestep_in_seconds>]

(Mandatory)

Specifies the computation timestep of the simulation in seconds. Value must be greater than zero. Timesteps that divide equally into one minute are recommended. For example, 0.5, 1, 2, 3, 5, 6, 7.5, 10, 12, 15, 20, 30, 45, 60, etc. seconds.

As of Build 2005-05-AN, different timesteps can be specified for different 1D and 2D domains. If the command is specified outside a [Start 2D Domain](#) / [End 2D Domain](#) block, the timestep will apply to any 2D domain that is not given a timestep. If it is specified within a [Start 2D Domain](#) / [End 2D Domain](#) block it only applies to that 2D domain.

**Timestep During Warmup ==
[{timestep} | <warmup_timestep_in_seconds>]**

(Optional)

Note: This feature was deactivated in Build 2005-01-AA when different timesteps in different domains was introduced. Although rarely used, it is being considered for future releases.

Warmup timestep in seconds that is applied during the warmup time. Only used if the warmup time is greater than zero (see [Warmup Time](#) command). Default value is the computational timestep.

Warmup Time == [{0} | <warmup_time_in_hours>]

(Optional)

Note: This feature was deactivated in Build 2005-01-AA when different timesteps in different domains was introduced. Although rarely used, it is being considered for future releases.

Time in hours from the simulation start, during which, a different timestep is applied (see [Timestep During Warmup](#)). Rarely used.

A.4 Output Control and Format Commands (.tcf)

Calibration Points MI File == <gis_layer>	A-12
Display Water Level == <X>, <Y>	A-12
Input Drive == <drive_letter>	A-12
Map Cutoff Depth == [{0.0} <value_in_metres>]	A-12
Map Output Data Types == [AP Cr CWF d E F FLC {h} MB1 n q R SS t {V} Wl10 Z0 Z1 Z2 Z3 Z4 ZUK0 ZUK1 ZH]	A-13
Map Output Format == [{DAT} XMDF XMDF DAT] [{SMS} SMS HIGH RES SMS HIGH RES CORNERS ONLY]	A-15
Map Output Interval == <time_in_seconds>	A-15
Mass Balance Output == [{ON} OFF]	A-16
Mass Balance Output Interval == <time_in_seconds>	A-16
Maximum Velocity Cutoff Depth == [<y> {0.1}]	A-16
Meshparts == [ON {OFF}]	A-17
Output Drive == <drive_letter>	A-17
Read GIS GLO == <gis_layer >	A-17
Screen/Log Display Interval == [{1} <timesteps>]	A-18
Start Map Output == <time_in_hours>	A-18
Store Maximums and Minimums == [ON ON MAXIMUMS ONLY {OFF}]	A-18
Time Output Cutoff Depths == [{OFF} <y1, y2, ...>]	A-18
UK Hazard Debris Factor == [<DF> {1}]	A-19
UK Hazard Formula == [D*(V+1.5) {D*(V+0.5)+DF}]	A-19
UK Hazard Land Use == [PASTURE WOODLAND URBAN {CONSERVATIVE} NOT SET]	A-20
Verbose == [ON {OFF}]	A-20

Zero Negative Depths in SMS == [{ON} | OFF]**A-20**

Calibration Points MI File == <gis_layer>

(Optional)

Assigns the peak water level calculated during the simulation as an extra attribute to the .mif/.mid file. Useful for obtaining peak flood levels at calibration points and other locations as direct output from TUFLOW. Up to a maximum of ten (10) files can be specified.

As of Build 2010-10-AB, the GIS layer is opened and closed during the startup phase so the existence of the layer is checked (rather than at the end of the simulation as this causes issues if the layer does not exist or has a save date later than the .tab file), and also so that the layers are copied if using the -c or -ca run-time option.

Display Water Level == <X>, <Y>

(Optional)

Displays the water level on the screen for cell located at X,Y where X and Y are the geographic coordinates in meters.

Input Drive == <drive_letter>

(Optional)

Changes the drive letter of any input files with a full path specified. For example “Input Drive == K” changes any input filepaths that have a drive letter to the K drive. Also see [Output Drive](#).

The 2010-10 release fixes a bug with [Input Drive](#) and [Output Drive](#) that didn’t change the drive letters of some files such as the .tlf and .wor files.

Map Cutoff Depth == [{0.0} | <value_in_metres>]

(Optional)

If the value in metres is greater than zero, TUFLOW only outputs results for cells with depths above the cutoff depth. This feature is particularly useful for direct rainfall modelling where there is a need to differentiate between sheet flow and flooding.

Map Output Data Types == [AP Cr CWF d E F FLC {h} MB1 n q R SS t {V} WI10 Z0 Z1 Z2 Z3 Z4 ZUK0 ZUK1 ZH]

(Optional)

Defines data types to be output in map (SMS) format. The letters stand for:

AP	atmospheric pressure in hPa
BSS	Bed Shear Stress as given by the equation below where ρ is density, g gravity, V velocity, n Manning's n and y depth. $\tau_{bed} = \frac{\rho g V^2 n^2}{y^{1/3}}$
Cr	Courant Number (as of Build 2008-08-AA).
CWF	Cell flow width factor (as of Build 2008-08-AA). Useful if using Read GIS Layered FC Shape . Only available if Map Output Format == SMS HIGH RES.
d	water depth in m
E	energy level in m (water level plus dynamic head) (Build 2002-10-AH)
F	Froude Number (Build 2002-10-AH)
FLC	Form loss coefficient (as of Build 2008-08-AA). Useful if using Read GIS Layered FC Shape . Only available if Map Output Format == SMS HIGH RES.
h	water levels in m
MB1	As of 2010-10 a much improved output for identifying problem areas in 2D domains (not yet implemented for 1D domains) by tracking the convergence at each cell since the last output time. Prior to the 2010-10 release outputs the mass error % occurring at 2D cells at that instant in time.
MB2	New for the 2010-10 release is the cumulative of MB1 up to that time in the simulation.
n	Manning's n values
q	unit flow vectors or flood hazard (product of depth and velocity) in m^2/s
R	flow regime value: 0 (zero) for normal (sub-critical flow with momentum); greater than zero (0) for upstream controlled friction flow (eg. supercritical flow); -1.5 for broad-crested weir flow; and -1 for submerged flow through a flow constriction. (Build 2002-10-AH).
SP	Stream Power as given by the equation below where τ_{bed} is bed shear stress (see BSS above) and V is velocity. $StreamPower = V \tau_{bed}$
SS	Sink/source flows in m^3/s – values at centre of cells are output at top right corner of cells.

t	Eddy viscosity coefficient (“e” prior to Build 2002-10-AH). (Note: Prior to Build 2008-08-AC if Viscosity Formulation == CONSTANT the coefficient was output as zero values irrespective of the coefficient set using Viscosity Coefficient , and the constant component introduced in Build 2008-08-AA was not included.)
V	Velocity vectors (water speed vectors) in m/s
WI10	Wind vectors in m/s
Z0	Australian flood hazard value, ie. VxD
Z1	Flood Hazard categories based on the Australian NSW Floodplain Management Manual (see Table 7.2 for details).
Z2	Flood Hazard categories as developed for the Herbert River Flood Study, Australia (see Table 7.2 for details).
Z3	Flood Hazard categories as developed for the Tweed River Flood Study, Australia (see Table 7.2 for details).
Z4	Flood hazard categories based on the Australian guidelines (see Table 7.2 for details).
ZMW1	Flood hazard categories based on Melbourne Water guidelines (see Table 7.2 for details).
ZMW2	Flood hazard categories based on Melbourne Water guidelines (see Table 7.2 for details).
ZUK0	UK flood hazard value based on the settings using UK Hazard Formula , UK Hazard Debris Factor and UK Hazard Land Use .
ZUK1	UK flood hazard category based on the settings using UK Hazard Formula , UK Hazard Debris Factor and UK Hazard Land Use .
ZH	ZH elevations over time (primarily used for morphological or breach (VG) modelling)

The letters can occur in any order or combination and are not case sensitive. Spaces between letters are optional. For example to output heads, velocities and unit flow enter the following line:

```
Map Output Data Types == h V q
```

The default is:

```
Map Output Data Types == hV
```

For further discussion on map output, see Section 7.3.1.

As of Build 2007-07-AF, ME was changed to MB1. This build also includes a bug fix that incorrectly calculated the mass error for MB1 output data along 2D H boundaries and links (eg. HX links; HT boundaries).

Map Output Format

```
== [ {DAT} | XMDF | XMDF DAT ]  
   [ {SMS} | SMS HIGH RES | SMS HIGH RES CORNERS ONLY ]  
(Optional)
```

Sets the format for TUFLOW map output. The default is SMS generic binary .dat file format excluding cells that were designated as “land”, ie. permanently inactive cells.

For the 2010-10 release there is now the option of using the .xmdf file format instead of the .dat format by specifying XMDF. If XMDF DAT is used, both sets of files are output!

The .xmdf format has the following advantages:

- All map output is in one file (ie. all .dat files are replaced by one .xmdf file).
- It is very fast to load up into SMS as it contains an data index header at the top of the file.
- Uses a folder structure within the file to manage data and more easily view and access data using an explorer style tree structure when using SMS.
- The format is recognised by other software such as MatLab.

The main reason for not using XMDF is the TUFLOW utilities such as TUFLOW_to_GIS and dat_to_dat need to be updated to recognise the .xmdf format, however, this is planned for early 2011 and will be announced via the TUFLOW Forum.

Note: The SMS HIGH RES and SMS HIGH RES CORNERS ONLY options are now ready for use (prior to Build 2008-08-AA, they were still being tested and reviewed).

The SMS HIGH RES option outputs ground elevations and results at the cell centres, mid-sides and corners, and takes into account where upstream controlled flow regimes occur by setting the water level accordingly. This provides a high-resolution output that is excellent for models with a lot of detail such as urban models with fences, roads, houses, etc, or with a significant amounts of upstream controlled flow regimes. Some of the utility programs such as TUFLOW_to_GIS.exe and dat_to_dat.exe do not yet recognise this format. It also requires extra RAM and the SMS .dat files are roughly four times the size.

The SMS HIGH RES CORNERS ONLY option uses the SMS HIGH RES approach, but only outputs at the cell corners as per the default SMS output. This provides a higher quality output without an increase in .dat file sizes, and the output will work with the TUFLOW_to_GIS.exe and dat_to_dat.exe utility programs.

For further discussion on map output, see Section 7.3.1.

Map Output Interval == <time_in_seconds>

(Optional)

The output interval in seconds for map based output. If the command is omitted, output is at every computational timestep.

Mass Balance Output == [{ON} | OFF]

(Optional)

If set to ON (the default as of Build 2006-06-AA), outputs the following:

- `_MB.csv` and `_MB2D.csv` files in the .tcf [Output Folder](#) and `_MB1D.csv` in the .ecf [Output Folder](#). These files contain mass balance calculations for the overall model, 1D domains and 2D domains at each time a line is displayed to the Console Window.
- `_MB1.dat` map output for 2D domains only and if specified by [Map Output Data Types](#).
- `_TSMB.mif` and `_TSMB1d2d.mif` GIS layers in the .ecf [Output Folder](#). These files contain summary and time based information on mass errors occurring at all 1D nodes and at 1D nodes connected to 2D HX links.

For the 2010-10 release the mass balance output has been reworked so that the display on the screen and the `_MB.csv` output files can be at different time intervals (prior to the 2010-10 release the interval in the `_MB.csv` files was by default every timestep unless changed by [Screen/Log Display Interval](#)). The new tcf command [Mass Balance Output Interval](#) is used for setting the interval in the `_MB.csv` files. There is also now a summary at the end of the simulation on the display console and .tlf file of key model performance indicators (see Section 7.6).

See Section 7.1.1 for information displayed to the Console Window and Section 7.5 for descriptions of the data in the output files.

Builds prior to Build 2007-07-AA only output 2D mass balance calculations if set to ON.

Setting to OFF does not produce any mass balance information to the Console Window or data files, and will reduce run-times.

Mass Balance Output Interval == <time_in_seconds>

(Optional)

The output interval in seconds for the `_MB.csv` output files. If the command is omitted, output is at the lesser of [Map Output Interval](#) and [Time Series Output Interval](#).

Maximum Velocity Cutoff Depth == [<y> | {0.1}]

(Optional)

Incorporated into the 2009-07 release (with a default value of 99999.), the command sets 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 a large number that exceeds the greatest depth (eg. 99999.), the maximum velocity will be as in releases prior to 2009-07, 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 which is the default for the 2010-10 release) is recommended as high velocities can occur at very shallow depths during the wetting and drying process.

Note, as a consequence of the velocities now being tracked independently of the maximum water level (ie. the maximum water level and maximum velocity can occur at different times), the maximum unit flow (q) and energy (E) were disabled for the 2010-10 release.

Meshparts == [ON | {OFF}]

(Optional)

If set to ON, the SMS mesh .2dm file is split up into different meshparts. Meshparts are determined as follows:

- Each 2D domain constitutes one meshpart.
- Each 1d_nwk layer constitutes one meshpart.

The benefits of using meshparts are that for future versions of SMS, while viewing results different meshparts can be switched on and off. Can also be useful when using the TUFLOW_to_GIS.exe utility (see Section 11.2) to remove meshparts from the output. An example is to remove the pipe network layer WLLs from the output.

The name allocated to the meshpart is either the 2D Domain Name (see [Start 2D Domain](#)) or the 1d_nwk layer name. The meshpart names can be clarified by opening the .2dm file in a text editor and searching for the “MESHPART” keyword.

Output Drive == <drive_letter>

(Optional)

Changes the drive letter of any output files with a full path specified. For example “Output Drive == K” changes any output filepaths that have a drive letter to the K drive. Also see [Input Drive](#).

The 2010-10 release fixes a bug with [Input Drive](#) and [Output Drive](#) that didn’t change the drive letters of some files such as the .tlf and .wor files.

Read GIS GLO == <gis_layer >

(Optional)

Opens .mif and .mid files containing details on gauge level output (GLO) locations. GLO controls map output based on the height of the water at a specified location – this is useful for producing a series of output based on gauge heights for flood warning purposes. It can also be used to display the height of the water at the gauge location to the screen.

As at Build 2003-01-AI, GLO works correctly for levels specified in a text file or by using the start, end and increment attributes. At this build, a buffer was also incorporated so that GLO only repeats at a level once the water level has moved at least 0.1m from the gauge level (this stops repetitive output if the model is “hovering” or “bouncing” around a gauge level.

Only the last occurrence of this command is used.

Screen/Log Display Interval == [{1} | <timesteps>]

(Optional)

Sets the frequency for display of output to the computer screen and log (.tlf) file. If omitted, every timestep is shown. A value of zero is treated the same as for a value of 1. A value of -2 suppresses the display except for any negative depth warnings. A value of -3 suppresses all timestep displays.

Start Map Output == <time_in_hours>

(Optional)

The simulation time in hours when map output commences. If the command is omitted, the simulation start time is used.

**Store Maximums and Minimums ==
[ON | ON MAXIMUMS ONLY | {OFF}]**

(Optional)

If set to “ON”, the highest and lowest values of selected map output data are tracked during the simulation, and included in the map output. The maximum and minimum values are those of every timestep, not those just at the output data times.

The maximum values are given the time 99999.0 in SMS .dat files and minimum values -99999.0

The ON MAXIMUMS ONLY option will output the maximums and not the minimums (this is the preferred option for flood modelling).

Note that maximums and minimums for velocities and unit flows are those that occur at the maximum/minimum water level or depth. Maximums and minimums are not at an instant in time.

For water level output (_h.dat) the time of the maximum water level in hours is also included in the .dat file and is assigned a time of 99999.1.

Time Output Cutoff Depths == [{OFF} | <y1, y2, ...>]

(Optional)

If one or more comma or space delimited depth values are specified, the _Times.dat file will contain additional output on the following time data for each depth:

- 1 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.
- 2 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.

Incorporated into the 2009-07 release.

Note: For 1D WLL output, the above _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.

UK Hazard Debris Factor == [<DF> | {1}]

(Optional)

Sets the debris factor (DF) value for calculating the flood hazard output for options ZUK0 and ZUK1 for [UK Hazard Formula](#) == $D*(V+0.5)+DF$. If a [UK Hazard Land Use](#) is specified, DF is determined from the debris factor land use table (see [UK Hazard Land Use](#)).

The default value is 1.0.

UK Hazard Formula == [D*(V+1.5) | {D*(V+0.5)+DF}]

(Optional)

Sets the formula to be used for calculating the flood hazard output for options ZUK0 and ZUK1, where D is depth, V velocity and DF Debris Factor (see [UK Hazard Debris Factor](#)). If a [UK Hazard Land Use](#) is set, the $D*(V+0.5)+DF$ option is used irrespective of this command.

ZUK0 produces a .dat file containing the actual value from applying the UK Hazard Formula, while ZUK1 outputs a .dat file containing an integer as per:

0 = No Hazard ($H \leq 0$)

1 = Low Hazard ($H \leq 0.75$)

2 = Moderate Hazard ($H \leq 1.25$)

3 = Significant Hazard ($H \leq 2.5$)

4 = Extreme Hazard ($H > 2.5$)

Formulae based on the UK publication DEFRA R&D Outputs: Flood Risks to People Phase Two Draft FD2321/TR1 and TR2.

UK Hazard Land Use

== [PASTURE | WOODLAND | URBAN | {CONSERVATIVE} | NOT SET]

(Optional)

Sets the land use category for varying debris factors with depth and velocity as specified in DEFRA R&D Outputs: Flood Risks to People Phase Two Draft FD2321/ TR2, Table 3.1 as shown below. Use NOT SET to ignore the land use setting and allow use of [UK Hazard Debris Factor](#) and [UK Hazard Formula](#) commands.

In the table below, the $V > 2\text{m/s}$ criteria in the last row is applied at all depths greater than 0.1m. Occasionally, as a 2D cell wets, a high velocity may occur, hence the 0.1m cut-off for applying the $V > 2\text{m/s}$ criteria.

Guidance on debris factors (DF) for different flood depths, velocities and dominant land uses

Depths	Pasture/Arable	Woodland	Urban	Conservative ¹
0 to 0.25 m	0	0	0	0.5
0.25 to 0.75 m	0	0.5	1	1
$d > 0.75\text{ m}$ and/or $v > 2$	0.5	1	1	1

Ref: FD2321/TR1 Table 3.1

1 Conservative option added to table

Verbose == [ON | {OFF}]

(Optional)

Available as of Build 2008-08-AA. If set to ON, outputs to the console window and .tlf file even more information! With future builds, information displayed to the screen may be reduced. To see all information, set to ON.

Zero Negative Depths in SMS == [{ON} | OFF]

(Optional)

Setting to ON zeroes depths in SMS map output if negative. The negative depth arises from interpolating the water level at the cell corners from the surrounding cell centres (this is necessary to convert TUFLOW output into SMS compatible format. Occasionally, due to the ZH Zpt being higher than the interpolated water level, the depth appears as negative.

Setting to OFF disables this command.

A.5 Bed Resistance Commands (.tcf)

Bed Resistance Cell Sides == [AVERAGE M AVERAGE n MAXIMUM n {INTERROGATE}]	A-21
Bed Resistance Depth Interpolation == [LINEAR M {SPLINE n} LINEAR n]	A-21
Bed Resistance Values == [{MANNING N} MANNING M CHEZY]	A-22
Change Zero Material Values to One == [ON {OFF}]	A-22
Depth/Ripple Height Factor Limit == [{10} <value>]	A-22
Read Materials File == <file>	A-22
Recalculate Chezy Interval == [{0} <timesteps>]	A-23

Bed Resistance Cell Sides == [AVERAGE M | AVERAGE n | MAXIMUM n | {INTERROGATE}]
(Optional)

Defines how the bed resistance value at a 2D cell's mid-side (ie. that used in the momentum equation) is calculated. The approach prior to Build 2007-07-AA was AVERAGE M, which takes the average Manning's M (1/Manning's n) value of the two adjoining cell centre values.

The AVERAGE n option takes the average Manning's n values of the cell centres. MAXIMUM n takes the maximum n values of the cell centres either side of the cell side.

INTERROGATE (the default) applies the exact value from the material polygons using [Read GIS Mat](#). The INTERROGATE option provides a higher resolution sampling of material values compared with just sampling at the cell centres. This higher resolution sampling is particularly useful in modelling urban areas where frequent and large changes in Manning's n occurs.

Note the [Read RowCol Mat](#) command is incompatible with the INTERROGATE option. If using [Read RowCol Mat](#), use AVERAGE M or AVERAGE n.

Bed Resistance Depth Interpolation
== [LINEAR M | {SPLINE n} | LINEAR n]
(Optional)

Controls how the Manning's n value is interpolated between the two depths if using the varying Manning's n with depth (see [Read Materials File](#)). As of Build 2008-08-AA, the default value is

SPLINE n which uses a curved fit so that the n values transition gradually. Prior to Build 2008-08-AA, the default was LINEAR M. LINEAR M and LINEAR n both use a linear interpolation of the M (1/n) and n value respectively.

Also as of Build 2008-08-AA, the depth taken to interpolate Manning's n values that vary with depth is taken as the minimum of the depths at the cell mid-side and the neighbouring cell centres (previously the depth used was that at the cell mid-side).

Bed Resistance Values == [{MANNING N} | MANNING M | CHEZY]

(Optional)

Sets the bed resistance formula to use. The default value is MANNING N.

Change Zero Material Values to One == [ON | {OFF}]

(Optional)

As of Build 2007-07-AA, the default material value is now zero (was previously one), which means that every cell must now be assigned a material value (ie. use [Set Mat](#) as the first materials command in the .tgc file). For backward compatibility set to ON.

Depth/Ripple Height Factor Limit == [{10} | <value>]

(Optional. Only used if bed resistance values are set to CHEZY)

Sets an upper limit on the ratio of the water depth over the ripple height in the formula for calculating Chezy values based on water depth. The value must be greater than 1/12, and if less than 1/12 is set to the default value of 10.

Read Materials File == <file>

(Optional. Presently only available for [Bed Resistance Values](#) == MANNING N based bed resistance values.

Can be extended to CHEZY and MANNING M upon request.)

Reads a text file containing Manning's n values for different materials (land-use types).

As of the 2009-07 Release, two file formats (.tmf and .csv) are available. The .tmf format is described in Section 4.4.12.2 and the .csv format in Section 4.4.12.3. The 2009-07 Release also allows multiple materials files to be specified (see Section 4.4.12.1).

Prior to the 2009-07 Release only the .tmf format is supported and only one .tmf file is used.

Recalculate Chezy Interval == [{0} | <timesteps>]

(Optional)

Warning: This command overwrites any previous use of [Bed Resistance Values](#) by setting CHEZY.

Sets the number of timesteps between recalculation of Chezy values based on the ripple height. The default value of zero indicates Chezy values are not recalculated (ie. remain constant throughout the simulation).

A.6 Flow Constriction (FC) Commands (.tcf)

Global FC Ch Factor == [{0.8} | <Ch>]

A-24

Read GIS FC == <gis_layer>

A-24

Global FC Ch Factor == [{0.8} | <Ch>]

(Optional)

The global C_h factor applied to flow constrictions when the flow upstream is submerged and the flow downstream is unsubmerged using the pressure flow equation for upstream controlled flow.

Read GIS FC == <gis_layer>

(Optional)

Opens .mif and .mid files containing details on flow constrictions to model bridges, box culverts, etc. (see Section 4.7.2). This command may be used any number of times.

As of Build 2008-08-AA, it is recommended to use the .tgc command [Read GIS FC Shape](#) in preference to this command.

A.7 Time-Series Output (PO & LP) Commands (.tcf)

CSV Time == [{DAYS} HOURS]	A-25
Excel Start Date == <days_since_1900>	A-25
Read GIS LP == <gis_layer>	A-25
Read GIS PO == <gis_layer>	A-25
Start Time Series Output == <time_in_hours>	A-26
Time Series Output Interval == <time_in_seconds>	A-26
Write PO Online == [ON {OFF}]	A-26

CSV Time == [{DAYS} | HOURS]

(1D & 2D/1D. Optional)

If set to HOURS, writes out time values in hours rather than days. Will also apply to 1D .csv output.

Excel Start Date == <days_since_1900>

(Optional)

Adjusts the time column of time series output by the amount specified. The amount is in days from the year 1900 as used by Microsoft Excel to manage its date fields. To determine this value, enter the date corresponding to time zero in the TUFLOW simulation as a date field in Excel. Change the format of the Excel cell to “Number”, and the number of days since 1900 is shown. Paste this number into the .tcf file for <days_since_1900>.

Read GIS LP == <gis_layer>

(Optional)

Opens .mif and .mid files containing details on longitudinal profile output (LP) locations (see Section 4.7.3). This command may be used any number of times.

Read GIS PO == <gis_layer>

(Optional)

Opens .mif and .mid files containing details on plot output (PO) locations (see Section 4.8.1). This command may be used any number of times.

Start Time Series Output == <time_in_hours>

(Optional)

The simulation time in hours when time series (PO and LP) output commences. If the command is omitted, the simulation start time is used. Also see [Time Series Output Interval](#).

Time Series Output Interval == <time_in_seconds>

(Optional)

The output interval in seconds for time series based output (PO and LP). If the command is omitted, PO and LP output is at every computational timestep. Also see [Start Time Series Output](#).

Write PO Online == [ON | {OFF}]

(Optional)

If set to ON writes the 1D and 2D time-series data files as the simulation progresses. The TS.mif file is only written if there is a 1D domain in the model. The files are closed off so that they can be opened in Excel or other software for viewing during a simulation, however, opening the files in some software (eg. Excel) may cause TUFLOW to pause at the next output time until the files are closed. If set to OFF the files are not written until the simulation finishes.

A.8 Initial Water Level (IWL) Commands (.tcf)

Read GIS IWL == <gis_layer>	A-27
Read RowCol IWL == <.mid_file>	A-27
Set IWL == <value>	A-27

Read GIS IWL == <gis_layer>

(Optional)

Opens .mif and .mid files defining the water level at the start of the simulation. This option allows the water level to vary spatially in height, for example, to set water levels of lakes. This command may be used any number of times. Note that if the water level of a cell is specified more than once, the last occurrence prevails.

Note: This command overwrites any IWL values set in the .tgc file for the same 2D cells.

For details see Section 4.9. This command was incorporated into Build 2004-03-AA.

Read RowCol IWL == <.mid_file>

(Optional)

Opens a .mid file defining the water level in each cell at the start of the simulation. This option allows the water level to vary spatially in height, for example, to set water levels of lakes. This command may be used any number of times. Note that if the water level of a cell is specified more than once, the last occurrence prevails.

Note: This command overwrites any IWL values set in the .tgc file for the same 2D cells.

For details see Section 4.9.

Set IWL == <value>

(Optional)

Sets the initial water level for all cells in the 2D domain to the value. Initial water levels for individual cells can be overwritten using [Read GIS IWL](#) or [Read RowCol IWL](#).

Note: This command overwrites any IWL values set in the .tgc file for the same 2D domain.

A.9 Restart File Commands (.tcf)

Read Restart File == <.trf_file> A-28

Write Restart File at Time == <time_in_hours> A-28

Write Restart File Interval == [{0} | <interval_in_hours>] A-28

Read Restart File == <.trf_file>

(Optional)

Reads a restart file written from a previous simulation (see [Write Restart File at Time](#) and [Write Restart File Interval](#)). The file must have the .trf extension, and if a 2D/1D model there must be a corresponding .erf file.

The water levels, velocities, wetting and drying status and other information saved in the restart file are used as the initial conditions for the simulation.

Note that the simulation start time needs to be changed to be the same as the time of the restart file.

Write Restart File at Time == <time_in_hours>

(Optional)

Sets when to write the restart file in hours. If the time is before the simulation start, the start time is used. Only the last occurrence of this command is used.

The restart file is given the extension .trf, and if there is 2D/1D dynamic linking, a .erf file is written for the 1D components. The .trf file is a binary file and not readable by a text editor. The .erf file is a text file and is readable by a text editor. The first line of the .erf file shows the time when the restart files were written. The time(s) when the restart files are written are displayed in the log file(s).

Write Restart File Interval == [{0} | <interval_in_hours>]

(Optional)

Sets the interval in hours between writing the restart file. The restart file is overwritten every <interval_in_hours> after the first restart file write. This is useful if a simulation is going unstable. A restart file is written prior to the instability, and is used to restart the simulation after modification of the topography to control the instability – thereby saving time in reaching the time of instability.

If set to zero, the default, or is negative, the restart file is written only once at the write restart file time. Only the last occurrence of this command is used.

A.10 Wetting and Drying Commands (.tcf)

Cell Side Wet/Dry Depth == [{0.001} <depth_in_m>]	A-30
Cell Wet/Dry Depth == [{0.002} <depth_in_m>]	A-30
HX ZC Check == [{ON} OFF]	A-30
SX Head Adjustment == [ON {OFF}]	A-31
SX ZC Check == [{ON} OFF]	A-31
Wetting and Drying == [{ON} ON NO SIDE CHECKS OFF]	A-31

Cell Side Wet/Dry Depth == [{0.001} | <depth_in_m>]

(Optional)

As of Build 2008-08-AA, this command has been disabled and the cell side wet/dry depth is always set to half the [Cell Wet/Dry Depth](#) value. This was implemented to prevent mass errors that sometimes occurred due to inappropriate cell side wet/dry depths being used. Does not apply if [Defaults](#) == PRE 2008-08. The default value of 0.001m remains unchanged.

Sets the wet/dry depth for determining when cell sides wet and dry. The default is 0.001m (1mm), but prior to Build 2006-03-AB was 0.03m. The depth should be selected according to the magnitude of flooding depths. The cell side wet/dry depth should be half to two-thirds of the [Cell Wet/Dry Depth](#).

For multiple 2D domains, this command is domain dependent.

Note: Ensure that [Cell Wet/Dry Depth](#) is also changed if this command is used.

Cell Wet/Dry Depth == [{0.002} | <depth_in_m>]

(Optional)

Sets the wet/dry depth for determining when a cell wets and dries. The default is 0.002m (2mm), but prior to Build 2006-03-AB was 0.05m. The depth should be selected according to the magnitude of flooding depths. For broad-scale models with large cell sizes values of up to 0.05m have typically been used, while for models using the direct rainfall approach, or that have a high proportion of steep flow, a wet/dry depth of less than a mm (eg. 0.0002m) may be required due to the substantial amount of shallow sheet flow.

As of Build 2008-08-AA, the cell wet/dry depth cannot be set to be below 0.0002m (0.2mm).

For multiple 2D domains, this command is domain dependent.

Note: Ensure that [Cell Side Wet/Dry Depth](#) is also changed if this command is used (not applicable as of Build 2008-08-AA provided [Defaults](#) == PRE 2008-08 is not set).

HX ZC Check == [{ON} | OFF]

(Optional)

If ON (the default), checks whether the minimum ZC elevation at or along a HX object (see Table 4.32 and Table 4.33) is above the 1D bed level interpolated between connected 1D nodes. This is necessary to ensure that there is water in the nodes when the 2D HX cells start to wet. If the ZC elevation is lower than the 1D bed level, unexpected flows or a surge of water may occur into the 2D domain.

Using the “Z” flag (see HX in Table 4.33), the ZC elevation is automatically raised at each 2D HX cell to slightly above the 1D node bed level. Only ZC elevations that are below the 1D bed are raised.

The checks and any automatic raising of ZC points includes the [Cell Wet/Dry Depth](#) value so that the ZC elevation is above the node bed plus the cell wet/dry depth.

If this option is set to OFF, lower ZC elevations are allowed and no automatic raising of ZC elevations occurs.

This option was incorporated in Build 2004-01-AE and is provided for backward compatibility.

SX Head Adjustment == [ON | {OFF}]

(Optional)

If OFF (the default), makes no adjustment for energy compatibility (ie. the 1D water level and 2D water level are set as equal to each other at the 2D SX link).

SX ZC Check == [{ON} | OFF]

(Optional)

If ON (the default), checks whether the minimum ZC elevation at or along a SX object (see Table 4.32 and Table 4.33) is below the connected or snapped 1D node bed level. This is necessary to ensure that the channels connected to the node only start flowing once the 2D SX cell is wet and the water level in the cell is above the lowest channel bed. If the ZC elevation is higher than the lowest channel, unexpected flows or a surge of water may occur in the 1D channels.

Using the “Z” flag (see SX in Table 4.33), the ZC elevation is automatically lowered at each cell associated with a SX object to below the connected or snapped 1D node bed level. Only ZC elevations that are above the node are lowered.

The checks and any automatic lowering of ZC points includes the [Cell Wet/Dry Depth](#) value so that the ZC elevation is below the node bed less the cell wet/dry depth.

If this option is set to OFF, higher ZC elevations are allowed and no automatic lowering of ZC elevations occurs.

This option was incorporated in Build 2002-08-AI and is provided for backward compatibility.

Wetting and Drying == [{ON} | ON NO SIDE CHECKS | OFF]

(Optional)

Controls the wetting and drying method.

The default ON dries cells once the cell water depth falls below the cell wet/dry depth (see [Cell Wet/Dry Depth](#) and [Cell Side Wet/Dry Depth](#)). A cell becomes wet once an adjoining cell’s water level is higher than the cell’s wet/dry depth. This method only considers adjoining wet cells that share a common cell side that is wet.

The ON NO SIDE CHECKS option is as described above, except that drying at the cell sides is not considered. All four adjoining cells are always considered.

The OFF option disables wetting and drying. This should only be used for models that have no cells likely to wet and/or dry.

A.11 Supercritical and Weir Flow Commands (.tcf)

Free Overfall == [{ON} ON WITHOUT WEIRS OFF]	A-32
Free Overfall Factor == [{0.6} <value_0.0_to_1.0>]	A-33
Froude Check == [{1} <froude_no>]	A-33
Froude Depth Adjustment == [{ON} OFF]	A-33
Global Weir Factor == [{1.0} <value>]	A-33
Shallow Depth Stability Factor == <SF>	A-33
Shallow Depth Weir Factor Cut Off Depth == [{0} <value_in_m>]	A-34
Shallow Depth Weir Factor Multiplier == [{1} <value>]	A-34
Supercritical == [{ON} OFF PRE 2002-11-AD]	A-34

Free Overfall == [{ON} | ON WITHOUT WEIRS | OFF]

(Optional)

The default ON option activates the free-overfall method described in Syme (1991). The method offers better stability; particularly where major wetting and drying occurs. It also allows large tidal flats to continue to drain without being cut-off at their edges. This option also activates the automatic broad-crested weir flow switch between upstream and downstream controlled flow. Use this option where weir flow occurs over levees and embankments. This option increases the computation time, typically by 10 to 30%, depending on the degree of wetting, drying and weir flow.

Upstream controlled flow is determined by comparison of the upstream and downstream energy levels. If upstream controlled, the broad-crested weir formula is used to define the flow across the cell-side. With the development of the [Supercritical](#) flow switch, the automatic weir flow algorithm was enhanced and only applies to cell-sides that have an adverse slope (ie. the bed slope from the ZC to ZU/ZV point is of opposite sign to the water surface slope) - see Section 4.7.3.

The ON WITHOUT WEIRS option activates the free-overfall method without the automatic weir flow switching. Mainly used for models developed prior to 1999, which is when the weir flow option became available.

The OFF option deactivates the free-overfall method. Used for models with little or no wetting and drying, and no upstream controlled weir flow.

Free Overfall Factor == [{0.6} | <value_0.0_to_1.0>]

(Optional)

Sets the free-overfall factor (see Syme 1991).

For the 2010-10 release the default was reduced from 0.8 to 0.6 as this was found to provide some minor enhancements to the propagation of steep flood waves. In most models the effect is minor and not of any great significance. The value should be less than 1.0 and greater than 0.0.

Froude Check == [{1} | <froude_no>]

(Optional)

Sets the minimum Froude Number that upstream controlled friction flow may occur. Only applies if [Supercritical](#) is set to ON, otherwise it is not used. Improved stability may occur in steeply flowing areas if <froude_no> is less than 1. <froude_no> cannot be below zero and would normally not exceed 1.

Froude Depth Adjustment == [{ON} | OFF]

(Optional)

Switches on or off an additional upstream controlled friction flow check incorporated in Build 2003-01-AF (See Section 4.7.3). Set to OFF for backward compatibility for models run prior to Build 2003-01-AF that use the upstream controlled friction feature (ie. see [Supercritical](#)).

Global Weir Factor == [{1.0} | <value>]

(Optional)

Factor that adjusts the broad-crested weir formula (see Section 4.7.3). Testing has shown that a value of 1.0 to 1.1 is needed to reproduce upstream controlled weir flow (Syme 2001). This factor is applied globally, although spatial variation of the factor can be specified through a GIS layer read by the geometry control file (see [Read GIS](#) or [Read RowCol](#) with the WrF option). Note that the global value and the spatially varying value are multiplied together (ie. one does not replace the other).

Shallow Depth Stability Factor == <SF>

(Optional)

Applies a higher roughness for several timesteps (depending on the magnitude of SF) when a cell side becomes wet. This gives improved performance for direct rainfall models (where large numbers of cells suddenly become wet). However, it can cause undesirable results for the simulation of rapidly propagating waves such as a dambreak flood wave (this is apparent by a “lurching” of the flood wave as it progresses down a dry valley). For the majority of models, this feature does not have a major effect on performance.

The default for models without direct rainfall is an SF value of zero (0), which disables this feature. SF equal to zero (0) is that generally recommended unless improved stability (and satisfactory performance) is gained through using a non-zero SF value. Note for Builds 2008-08-AA and 2008-08-

AB the default for models without direct rainfall was SF equal to three (3) - for backward compatibility to these builds use [Shallow Depth Stability Factor](#) == 3. For direct rainfall models the default is six (6). SF must be in the range from 0 to 10.

```
Shallow Depth Weir Factor Cut Off Depth ==  
[ {0} | <value_in_m> ]  
(Optional)
```

Note: This command is not recommended for use since the advent of the [Supercritical algorithm](#).

Sets the depth in metres for applying the shallow depth weir factor algorithm. This value typically varies from zero (preferable) to up to 0.2 for major river systems. See [Shallow Depth Weir Factor Multiplier](#) on how it is applied. A value less than the [Cell Side Wet/Dry Depth](#) has no effect.

```
Shallow Depth Weir Factor Multiplier == [ {1} | <value> ]  
(Optional)
```

Note: This command is not recommended for use since the advent of the [Supercritical algorithm](#).

Stability was achieved in old TUFLOW models by adjusting the Manning's n value at shallow depths and invoking a weir equation. The multiplier typically has a value in the range of 5 to 20, and is linearly reduced from its full value at zero depth to a value of 1 at the [Shallow Depth Weir Factor Cut Off Depth](#). A value of 1 has no effect on the weir calculations and is that recommended.

```
Supercritical == [ {ON} | OFF | PRE 2002-11-AD ]  
(Optional)
```

Sets the supercritical flow mode. If set to ON (the default), flow automatically switches into upstream controlled friction flow, allowing the supercritical flow conditions on steep slopes to be modelled. See Section 4.7.3 and [Froude Check](#) for more details.

If set to OFF, and [Free Overfall](#) is set to ON, the broad-crested weir formula applies where flow conditions are predicted to be upstream controlled. For simulations prior to Build 2002-11-AC, this flag may need to be set to OFF for backward compatibility.

Setting to PRE 2002-11-AD provides backward compatibility for simulations carried out using supercritical flow prior to Build 2002-11-AD. In Build 2002-11-AD, additional checks using the Froude Number specified by [Froude Check](#) were incorporated in addition to the downstream/upstream controlled flow check comparison. This may produce different results in some flow conditions. The ON option is to be used in preference to the PRE 2002-11-AD option.

A.12 Eddy Viscosity Commands (.tcf)

Viscosity Coefficients == [{0.2, 0.1} | <smag, cons>] A-35

Viscosity Formulation == [CONSTANT | {SMAGORINSKY}] A-35

Viscosity Coefficients == [{0.2, 0.1} | <smag, cons>]

(Optional)

Sets the viscosity coefficient(s) (see Section 3.7).

As of Build 2008-08-AA, if [Viscosity Formulation](#) == SMAGORINSKY (the default), two values can now be specified, the first being the Smagorinsky coefficient component, and the second a constant coefficient component. The default values are 0.2 and 0.1 respectively. The Smagorinsky coefficient component is subject to variation depending on the spatial variation of the velocity magnitude and direction (see Section 3.7). The constant component remains unchanged. If a second value is not specified, the constant component is zero.

Prior to Build 2006-03-AB, the default was 1.0 to match the default [Viscosity Formulation](#) of CONSTANT. From Build 2006-03-AB to Build 2008-08-AA, the SMAGORINSKY formulation was the default, and the default value was set to 0.2 accordingly.

If using the [Viscosity Formulation](#) == SMAGORINSKY, the coefficient is typically between 0.06 to 1.0.

Viscosity Formulation == [CONSTANT | {SMAGORINSKY}]

(Optional)

Sets the viscosity formulation (see Section 3.7). Options are:

- “CONSTANT” – the viscosity coefficient remains constant
- “SMAGORINSKY” – applies the Smagorinsky formula

As of Build 2006-03-AB, the default was changed from CONSTANT to SMAGORINSKY.

A.13 Miscellaneous Commands (.tcf)

Cell Size == <value_in_meters>	A-36
Defaults == [PRE 2010-10 PRE 2008-08 PRE 2007-07 PRE 2006-06]	A-36
Double Precision == [ON {OFF}]	A-39
First Sweep Direction == [AUTOMATIC {POSITIVE} NEGATIVE]	A-39
Line Cell Selection == [METHOD A METHOD C {METHOD D}]	A-40
Inside Region == [METHOD A {METHOD B}]	A-40
Maximum Points == [<mp> {500000}]	A-41
Maximum Vertices == [<mp> {100000}]	A-41
Model Platform == [w32 w64]	A-41
Model Precision == [SINGLE DOUBLE]	A-41
Model TUFLOW Build == <build>	A-42
Model TUFLOW Release == [2006-06 2007-07 2008-08 2009-07 2010-09 2010-10]	A-42
Set Route Cut Off Type == [{Depth}]	A-42
Set Route Cut Off Values == <y1, y2, ...>	A-43

Cell Size == <value_in_meters>

(Mandatory if not specified in .tgc file)

Sets the grid cell size in meters. Rarely used; normally specified in the .tgc file (see Section 4.5).

Defaults

== [PRE 2010-10 | PRE 2008-08 | PRE 2007-07 | PRE 2006-06]

(Optional)

PRE 2010-10 sets the default settings for both 1D and 2D domains to those used in the 2008-08 release (Builds 2008-08-XX) – there was no change in the default settings between the 2008-08 and 2009-07 releases. Also see Section 10.

The settings/changes invoked by **PRE 2010-10** are:

2D Domains (.tcf file):

[Free Overfall Factor](#) == 0.8

[HQ Boundary Approach](#) == METHOD A

[Rainfall Boundaries](#) == SMOOTHED TIME CENTRED

[Maximum Velocity Cutoff Depth](#) == 99999.

[Negative Depth Approach](#) == METHOD A

[Link 2D2D Approach](#) == METHOD A

[XF Files](#) == OFF

2D Domains (.tgc file):

[TIN Angles](#) == 95, 30 (Note, not fully backward compatible – see [TIN Angles](#).)

1D Domains (.ecf file):

[S Channel Approach](#) == METHOD A

[Structure Losses](#) == ADJUST

[Structure Losses Approach](#) == METHOD A

[Manholes at All Culvert Junctions](#) == OFF

PRE 2008-08 sets the default settings for both 1D and 2D domains to those used in the previous release (Builds 2007-07-XX). Also see Appendix F.1.

Note: If using the Compaq version (see Section 5.2) the PRE 2008-08 option is automatically set.

The settings/changes invoked by **PRE 2008-08** are:

2D Domains (.tcf file):

[Bed Resistance Depth Interpolation](#) == LINEAR M

[Cell Side Wet/Dry Depth](#) command not disabled.

[Viscosity Coefficient](#) == 0.2 (does not assign a constant component of 0.1)

The enhanced treatment of the viscosity term at dry boundaries, cell-centred application of Smagorinsky viscosity formula, and application directly to cell mid-sides (rather than whole of cell) are not applied - see Section 3.7.

[Verbose](#) == ON (displays all information to the console window and .tlf file)

[Zero Z Point](#) == WARNING

1D Domains (.ecf file):

[Weir Flow](#) == Method A

[Culvert Flow](#) == Method D

[Interpolate Cross-Sections](#) == OFF

[Interpolate Culvert Inverts](#) == OFF

The _cnch.tmp file is still created and used.

WLLs are duplicated if overriding a channel that has WLLs.

Allocation of only one 2D SX cell to pits (see Section 4.5.1.3).

New ERROR messages are reported as WARNING messages.

PRE 2007-07 sets the default settings for both 1D and 2D domains to those used in the previous release (Builds 2006-06-XX) (for more information see Appendix F.2).

The settings invoked by **PRE 2007-07** are:

2D Domains (.tcf file):

[Change Zero Material Values to One](#) == ON

[Inside Region](#) == Method A

[Line Cell Selection](#) == Method C

[VG Z Adjustment](#) == ZC

[Bed Resistance Cell Sides](#) == AVERAGE M

[MI Projection Check](#) == WARNING

Reinstates the 2D FC BD and FD bug that did not correctly apply the Mannings_n attribute.

Reinstates the bug that incorrectly set the water levels on dried VG cells.

1D Domains (.ecf file):

[Culvert Flow](#) == Method C

[Culvert Critical H/D](#) == 1.5

PRE 2006-06 sets the default settings for both 1D and 2D domains to those used in the previous release (Builds 2005-05-XX). It also automatically invokes the PRE 2007-07-AA default settings above (except where one of the settings below prevails).

The **PRE 2006-06** settings are:

2D Domains (.tcf file):

[Cell Wet/Dry Depth](#) == 0.05

[Cell Side Wet/Dry Depth](#) == 0.03

[Adjust Head at Estry Interface](#) == ON

[Boundary Cell Selection](#) == Method A

[Line Cell Selection](#) == Method A

[Viscosity Formulation](#) == CONSTANT

[Viscosity Coefficient](#) == 1.0

[Oblique Boundary Method](#) == ON PRE 2005-10-AH

[BC Wet/Dry Method](#) == PRE 2005-11-AF

[SX Head Adjustment](#) == ON

1D Domains (.ecf file):

[Structure Losses](#) == FIX

[Storage Above Structure Obvert](#) == CHANNEL WIDTH

[Depth Limit Factor](#) == 1

[Culvert Flow](#) == Method B

[Culvert Add Dynamic Head](#) == OFF

[Bridge Flow](#) == Method A

[WLL Approach](#) == Method A

[Apply All Inverts](#) == OFF

[Conveyance Calculation](#) == CHANGE IN RESISTANCE

[Flow Calculation](#) == Method A

Double Precision == [ON | {OFF}]

(Optional)

As of Build 2008-08-AA, this command is no longer supported, even if [Defaults](#) == PRE 2008-08 is specified. For models requiring double precision use the double precision compiled version as discussed in Section 5.2. Also see [Model Precision](#).

First Sweep Direction == [AUTOMATIC | {POSITIVE} | NEGATIVE]

(Optional)

Build 2004-05-AD reworked and tested part of the Stelling scheme that can vary the sweep direction depending on the flow regime at the time. In rare situations, this may cause very slight difference in results between two models (eg. before and after cases) in areas where there should be no difference at all. This was as a result of the unpredictable sweep direction in one part of the scheme. Testing on a number of models showed that by fixing the sweep directions, there was virtually no difference in results. This also solved the rare situation where two models were showing a slight difference in areas they should not have been.

This command is provided for backward compatibility, although it is not considered that this will be necessary in most models. To use the approach prior to Build 2004-05-AD use the AUTOMATIC option.

Line Cell Selection == [METHOD A | METHOD C | {METHOD D}]
(Optional)

Sets the method for selecting 2D cells along lines in GIS layers (for lines in 2d_bc layers see [Boundary Cell Selection](#)). METHOD A was that used prior to Build 2006-06-AA, and is provided for backward compatibility.

METHOD D (the default) and METHOD C (the default up until Build 2007-04-AC), use the cell “cross-hair” approach where a cell is only selected if the line intersects imaginary “cross-hairs” that extend from cell mid-sides to cell mid-sides. METHOD D differs in that it uses a more advanced approach for assigning interpolation weightings of 1D node water levels based on the perpendicular intersection of the 2D cell centre with the boundary line (similar to that used for Z Lines). This provides a “smoother” water surface profile along HX lines, offers better stability along 1D/2D HX interfaces, and is the recommended approach.

As of Build 2007-04-AC, this command also sets the [Boundary Cell Selection](#) setting making [Boundary Cell Selection](#) redundant. Prior to Build 2007-04-AC, [Boundary Cell Selection](#) should also be set to METHOD C.

METHOD D and METHOD C are particularly useful along HX lines that follow, for example, the top of a levee or flood defence wall, as it ensures that the same cells selected along the 1D/2D interface are those raised by the Z line.

Inside Region == [METHOD A | {METHOD B}]
(Optional)

METHOD B offers much faster processing for assigning values to 2D cells or cell mid-sides that fall within a polygon using commands that process polygons from a .mif file (eg. [Read GIS Mat](#)).

METHOD A uses the previous, much slower, method, and is provided in case of backward compatibility issues. Testing thus far has shown the two methods yield identical results although it is possible that if a 2D cell centre or mid-side point lies exactly on a polygon boundary different results may occur.

To appreciate the increase in startup time this feature offers, testing on two large models reduced the startup time from 20 minutes to 3 minutes for one model, and from 40 minutes to 5 minutes for the other. The faster startup time occurs for any polygon layers being accessed from the .tgc and .tbc files, particularly those containing large number of vertices.

This command was introduced for Build 2007-07-AA.

Maximum Points == [<mp> | {500000}]

(Optional)

Controls the maximum number of elevation points that can exist in a GIS layer referenced by commands such as [Create TIN Zpts](#), [Read GIS Z Line](#), [Read GIS Z HX Line](#), [Read GIS Z Shape](#), [Read GIS Variable Z Shape](#), [Read GIS FC Shape](#) and [Read GIS Layered FC Shape](#). If the number of points exceeds the amount specified, an ERROR occurs and this command needs to be used to increase the maximum number of points. This value can also be lowered to reduce RAM requirements.

Maximum Vertices == [<mp> | {100000}]

(Optional)

Controls the maximum number of vertices that can exist in a single polyline or polygon in a GIS layer referenced by commands such as [Create TIN Zpts](#), [Read GIS Z Line](#), [Read GIS Z HX Line](#), [Read GIS Z Shape](#), [Read GIS Variable Z Shape](#), [Read GIS FC Shape](#) and [Read GIS Layered FC Shape](#). If the number of vertices exceeds the amount specified, an ERROR occurs and this command needs to be used to increase the maximum number of vertices. This value can also be lowered to reduce RAM requirements.

Model Platform == [w32 | w64]

(Optional)

Forces the .tcf file to only be run using either a 32-bit (w32) or 64-bit (w64) version.

This feature was incorporated into the 2010-10 release.

Model Precision == [SINGLE | DOUBLE]

(Optional)

Used to force a model to use either the single or double precision version of TUFLOW. If the command is not specified, the model may be run in either version. If SINGLE is specified, and the double precision version of TUFLOW is used, the simulation stops with an error. Similarly, if DOUBLE is specified, the simulation stops if the single precision version is used. For further discussion see Section 5.2.

This command is useful for ensuring that the same (single or double) version of TUFLOW is always used.

Using double precision can be needed for some models, especially when using direct rainfall or where the ground elevations are large (eg. several hundred metres above sea level) relative to zero. The additional precision is needed when adding a very small rainfall depth (a tiny fraction of a metre) or inflow to a high elevation (hundreds of metres). If single precision is used for these models, unacceptable arithmetic errors and associated mass errors may occur.

Model TUFLOW Build == <build>

(Optional)

Forces the .tcf file to only be run using the specified build of TUFLOW. <build> must be identical to that displayed in the .tlf file or when double clicking TUFLOW.exe.

This feature was incorporated into the 2010-10 release and also for the DB or later builds of the 2006-06, 2007-07, 2008-08 and 2009-07 releases.

For example:

- Model TUFLOW Build == 2010-10-AA-iSP-w64
- Model TUFLOW Build == 2007-07-DB

Model TUFLOW Release ==

[2006-06 | 2007-07 | 2008-08 | 2009-07 | 2010-09 | 2010-10]

(Optional)

Forces the .tcf file to only be run using a build from a specified release of TUFLOW. The specified release must be as per one of the options above.

This feature was incorporated into the 2010-10 release and also for the DB or later builds of the 2006-06, 2007-07, 2008-08 and 2009-07 releases.

Set Route Cut Off Type == [{Depth}]

(Optional)

Sets the cutoff value type for evacuation routes if the Cut_Off_Type attribute in the [Read GIS Z Shape Route](#) layer is blank. As of the 2009-07 release only Depths can be used. Other options such as Velocity and Hazard are planned for future releases. The cutoff values are set using [Set Route Cut Off Values](#) in the .tcf and/or .tgc file, and evacuation routes are described in Section 4.8.2 and set using the .tgc file command [Read GIS Z Shape Route](#).

This command may be used in either the .tcf and/or .tgc file. If used in the .tcf it is the global default setting when the .tgc file is processed. If used in the .tgc file its location in the file is important in that it only applies to subsequent evacuation route commands. It may be used any number of times in the .tgc file so as to change the evacuation route settings at different points within the .tgc file.

Incorporated into the 2009-07 release.

Set Route Cut Off Values == <y1, y2, ...>

(Optional)

Sets the cutoff values for the evacuation route feature (see Section 4.8.2) if the Cut_Off_Values attribute in the [Read GIS Z Shape Route](#) layer is blank. The type of cutoff values is set using [Set Route Cut Off Type](#) and evacuation routes are set using the .tgc file command [Read GIS Z Shape Route](#).

This command may be used in either the .tcf and/or .tgc file. If used in the .tcf it is the global default setting when the .tgc file is processed. If used in the .tgc file its location in the file is important in that it only applies to subsequent evacuation route commands. It may be used any number of times in the .tgc file so as to change the evacuation route settings at different points within the .tgc file.

Incorporated into the 2009-07 release.

A.14 Water Level Instability Detection Commands (.tcf)

Instability Water Level == [{see_below} | <value_in_meters>] A-44

Negative Depth Approach == [Method A | {Method B}] A-44

Water Level Checks == [{ON} | OFF] A-44

Instability Water Level == [{see_below} | <value_in_meters>]
(Optional)

As of Build 2006-06-AA, the default water level used to detect instabilities is ten metres higher than the highest cell elevation of all cells (whether wet, dry or permanently dry). Any unassigned elevations (which are given a value of 99999, are not included). Prior to Build 2006-06-AA, the level was set a 1m higher than the highest level, and would include any 99999 values, thereby allowing an instability to oscillate within a huge range, and in some cases cause TUFLOW to crash without notification.

Alternatively, this command is used to set the instability water level manually.

Negative Depth Approach == [Method A | {Method B}]
(Optional)

Method B provides an improved performance in handling negative depths (which are a consequence of a non-convergence of the solution). It is only applied if a cell experiences a depth below zero at its centre. If the flow across any of the four cell sides is extracting water from the cell, a high friction is temporarily applied to that side and the high friction is reduced back to normal within several timesteps. This tends to stop a cell repeatedly experiencing a negative depth, which can cause mass errors. The negative depth problem can be particularly acute where there is very steep (supercritical) flow with high velocities, combined with some cell sides attempting to switch into weir flow.

Method A represents the case of no special treatment of negative depths.

Introduced in Build 2009-07-AB with Method A as the default. Method B is the default for the 2010-10 release.

Water Level Checks == [{ON} | OFF]
(Optional)

The default ON option carries out checks on water levels to detect any significant instabilities. Instabilities are triggered when a water level exceeds the [Instability Water Level](#) (see below) or falls

below the negative of the [Instability Water Level](#). Switching this option off reduces the computation time very slightly.

A.15 Boundary Condition Commands (.tcf)

BC Database == <.csv_file>	A-46
BC Event Name == <bc_event_name>	A-46
BC Event Source == <bc_event_text> <bc_event_name>	A-47
BC Event Text == <bc_event_text>	A-47
BC Zero Flow == [{OFF} START END START and END]	A-47
Boundary Cell Selection == Method [A B {C}]	A-47
Event File == <.tef_file>	A-48
Model Events == <e1> <e2> <e3>...	A-48
VG Z Adjustment == [{MAX ZC} ZC ZH]	A-48

BC Database == <.csv_file>

(Mandatory)

Sets the active BC Database file as described in Sections 4.10.1 and 4.10.2. The file is usually created using spreadsheet software such as Microsoft Excel.

If the BC Database is specified in the TUFLOW .tcf file, it is set as the active database for both 2D and 1D models. However, the active database can be changed at any stage in the .tbc and .ecf files by repeating the command with the new database set as the <.csv_file>.

A BC Database must be specified before any of the other BC commands are used.

BC Event Name == <bc_event_name>

(Optional)

Sets the active BC name to be substituted wherever [BC Event Text](#) values occurs in the BC Database. See Section 4.10.3 for a description of how the BC event commands operate.

If specified in the .tcf file, <bc_event_name> also applies to any 1D models.

The <bc_event_name> value can be changed at any stage by repeating this command in the .tbc and .ecf files. For example, it may be set to “Q100” to read in the 100 year catchment inflows, then set as “H010” to read in the 10 year ocean levels for the downstream boundary. Note that, in this case, the

locations of the catchment inflows and downstream boundaries would have to be placed in two separate GIS layers.

BC Event Source == <bc_event_text> | <bc_event_name>

(Optional)

Combines [BC Event Name](#) and [BC Event Text](#) into one command, and can occur up to 100 times to allow multiple events within the one simulation. Cannot be used in conjunction with [BC Event Text](#).

Ideally used within [Define Event](#) blocks in a .tef file (TUFLOW Event File) – see Section 4.12.1, but can be used one or more times in a .tcf file.

An ERROR 2313 occurs if <bc_event_text> is not unique for a simulation.

BC Event Text == <bc_event_text>

(Optional)

Sets the text in the BC Database that is to be substituted by the [BC Event Name](#) value. See Section 4.10.3 for a description of how the BC event commands operate.

If specified in the .tcf file, <bc_event_text> also applies to any 1D models. The <bc_event_text> value can be changed at any stage by repeating this command in the .tbc and .ecf files, although it is strongly recommended that the <bc_event_text> value is standardised across all models and the command is specified only once.

BC Zero Flow == [{OFF} | START | END | START and END]

(Optional)

If set to START, END or START and END, zeros the start and/or end of 1D and 2D flow hydrographs (QT, ST, SA) as the option implies. The hydrograph is modified by adding another row at the start/end of the hydrograph with a flow value of zero.

The benefit is that should a simulation start before or finish after the start/end of a hydrograph, the flow from this hydrograph into the model will be zero. (TUFLOW, by default, extends the first value of all boundary conditions backwards in time indefinitely, and the last value forwards in time indefinitely.)

Only applies to hydrographs sourced via the [BC Database](#) file.

This command was introduced for Build 2007-07-AA.

Boundary Cell Selection == Method [A | B | {C}]

(Optional)

As of Build 2007-07-AA, this command is now redundant other than for backward compatibility to previous builds.

Sets the method for selecting 2D cells along a boundary or interface line. Method A was that used prior to Build 2006-06-AA, and is provided for backward compatibility. Method B was a pre-cursor to Method C, and is also provided for backward compatibility for the few models that it was used for.

Method C uses the cell “cross-hair” approach where a cell is only selected if the line intersects imaginary “cross-hairs” that extend from cell mid-sides to cell mid-sides. This approach is the same as now used for [Read GIS Z Line](#) with the THICK option. This is particularly useful along HX lines that follow, for example, the top of a levee or flood defence wall, as it ensures that the same cells selected along the 1D/2D interface are those raised by the Z line. See also [Line Cell Selection](#).

Event File == <.tcf_file>

(Optional)

Sets the active event file. See Section 4.12.1.

Whilst this command maybe be repeated to change the active event file it is recommended that only one event file be created for all simulations.

Model Events == <e1> | <e2> | <e3>...

(Optional)

Alternative to using the `-e` option when running TUFLOW (see Table 5.2). Up to nine (9) separate events that have been defined using [Define Event](#) can be specified. Separate the event names using vertical bars. The event names specified will be automatically appended to the output filenames if `~e~` or `~eX~` are not in the .tcf filename.

For example:

Model Events == Q100 | 02h

Note that using the `-e` run option will override this command.

VG Z Adjustment == [{MAX ZC} | ZC | ZH]

(Optional)

Introduced for Build 2006-06-AA. The ZH option provides backward compatibility for models using the original VG adjustment of Zpts based on changing the ZH values.

MAX ZC was introduced for Build 2007-07-AA and was set as the new default. Can now specify MAX before or after ZC option to force the adjusted ZU/ZV and ZH points to be set to the maximum ZC value rather than an interpolated ZC value. This option provides significant enhancements in some situations to the stability of the flow over the breach.

A.16 Boundary Treatment Commands (.tcf)

Adjust Head at Estry Interface == [ON ON VARIABLE {OFF}]	A-49
BC Wet/Dry Method == [PRE 2005-11-AF]	A-50
Check Inside Grid == [{ERROR} WARNING OFF]	A-50
Distribute HX Flows == [ON {OFF}]	A-50
Extrapolate Heads at Flow Boundaries == [ON {OFF}]	A-50
HQ Boundary Approach == [METHOD A {METHOD B}]	A-50
Link 2D2D Approach == [METHOD A {METHOD B}]	A-51
Null Cell Checks == [ON {OFF}]	A-51
Oblique Boundary Alignment == [{NEAREST TO LINE} CENTRE TO CENTRE]	A-51
Oblique Boundary Method == [{ON} ON PRE 2005-10-AH ON METHOD 2 OFF]	A-52
Rainfall Boundaries == [{STEPPED} SMOOTHED SMOOTHED TIME CENTRED]	A-52
Rainfall Gauges == [One per Cell {Unlimited per Cell}]	A-52
Unused HX and SX Connections == [{ERROR} WARNING]	A-53

Adjust Head at Estry Interface == [ON | ON VARIABLE | {OFF}]
(Optional)

As of Build 2006-03-AB the default is OFF, and this command's main use is to provide backward compatibility for older models using the previous default of ON. If set to ON, TUFLOW lowers the 1D water level sent to the 2D cells along HX lines by an average of the dynamic head based on the 2D velocities, unless the S Flag is specified for a HX line (see Table 4.33). This can be useful where the 1D water level is more representative of a static water level (1D schemes roughly approximate the variation in water level across a flowpath due to the dynamic head). Based on numerous and wide ranging application of HX lines, it is recommended as of Build 2006-03-AB that this command use the default OFF setting.

The ON VARIABLE option, adjusts the water level on a cell-by-cell basis and is presently not recommended other than for research reasons.

BC Wet/Dry Method == [PRE 2005-11-AF]

(Optional)

As of Build 2005-11-AF, water levels at HX cells are set to be not less than the ZC plus [Cell Wet/Dry Depth](#) value for when the 1D water level falls below the HX cell. This enhances stability in some situations. For backward compatibility use the PRE 2005-11-AF option.

Check Inside Grid == [{ERROR} | WARNING | OFF]

(Optional. Incorporated Build 2005-05-AN)

By default, some layers, such as the 2d_bc and 2d_po layers, must have all of their objects fall within the 2D domain they are associated with, otherwise an ERROR is issued and the simulation stops. Should it be required that this check be switched off, set to either WARNING (a WARNING is issued and will be included in the _messages.mif file) or OFF (no checks are made). The treatment of objects that fall partly inside a 2D domain should be cross-checked viewing the check files and results as to how they were treated.

Distribute HX Flows == [ON | {OFF}]

(Optional)

Offers an alternative option for distributing the flow across HX lines to/from the 1D nodes. The distribution is based on a linear interpolation based on the distance of the HX cell from the 1D node. This option, on some models, has improved model performance if the 1D/2D interface is being problematic. The feature is still under trial and should be benchmarked before adopting. It is not available for the ISIS 1D link as incorrect results presently occur. It has not been tested with the XP-SWMM 1D link.

Extrapolate Heads at Flow Boundaries == [ON | {OFF}]

(Optional)

Undocumented feature.

HQ Boundary Approach == [METHOD A | {METHOD B}]

(Optional)

Sets the approach to be used for automatically generated 2D HQ boundaries. Prior to the 2010-10 release METHOD A was the default approach. METHOD B was introduced for the 2009-07 release and is recommended over Method A.

METHOD A extends the HQ curve by ten metres on its last point which can cause a sudden change in the slope of the curve at this elevation. This method also does not issue WARNING 2365 if the top of the curve is exceeded during a simulation.

METHOD B sets the top level in HQ curve to be that of the highest cell elevation, and issues WARNING 2365 if the top of the curve is exceeded during the simulation. If the elevation range of the curve is less than one metre, the top elevation is raised to 1 metre above the lowest 2D cell.

Alternatively the 2d_bc d attribute can be used to specify the max depth to be used for generating the curve (if less than 1m it is set to 1m).

Link 2D2D Approach == [METHOD A | {METHOD B}]

(Optional)

METHOD B is an improvement over METHOD A which was prone to generating flows where the 2D2D link was located along dry boundaries with “bumpy” topography.

METHOD A is the default for the 2009-07 release, however, METHOD B is the default for the 2010-10 release and is preferred over METHOD A. This command and METHOD B was also incorporated into the 2009-07 release.

Null Cell Checks == [ON | {OFF}]

(Optional. Incorporated Build 2001-08-AE)

Switches on and off the checks that ensure null cells occur on one side of an external boundary. A TUFLOW simulation prior to Build 2001-08-AE will not proceed unless a null cell occurs on one side of an external boundary cell (this was used to indicate the inactive side of the boundary line). Setting this to OFF (the default) allows ESTRY models to be inserted through the 2D domain with no need to specify null cells (eg. a 1D creek flowing through a 2D floodplain). It also allows land cells, instead of null cells, to be specified against a boundary on the inactive side.

Note: Prior to Build 2001-08-AE models were checked for the null cells along boundaries. For models prior to this build, you may need to set this flag.

Oblique Boundary Alignment == [{NEAREST TO LINE} | CENTRE TO CENTRE]

(Optional)

As of Build 2006-06-AA, this command is considered redundant (other than for backward compatibility), as the Method C or D ([Boundary Cell Selection](#) and [Line Cell Selection](#)) ensures that the same cells are selected along both HX lines and Z lines.

Build 2004-01-AA changed the method used for assigning boundary cells (Code 2) along a 2d_bc line to be commensurate with the method used for 3D breaklines (see [Read GIS Z Line](#)). Prior to this build, a 2d_bc line segment was shifted so that each end of the line segment was located at the centre of the nearest 2D cell. This approach is a hangover from earlier versions of TUFLOW when data was entered in on a cell reference basis. This approach will not necessarily choose the cell nearest to the line, and would choose different cells than that using [Read GIS Z Line](#). For example, this was of a particular nuisance where 2D HX lines are used to link 1D and 2D domains along a levee defined by a 3D breakline. The CC option in [Read GIS Z Line](#) was introduced to minimise this issue.

The default from Build 2004-01-AA onwards is to choose boundary cells nearest the digitised line. For backward compatibility, the original approach can be used by specifying this command with the CENTRE TO CENTRE option.

Oblique Boundary Method**== [{ON} | ON PRE 2005-10-AH | ON METHOD 2 | OFF]**

(Optional)

If set to ON, applies Oblique Boundary Method 1 as documented in Syme (1991). This offers substantial improvements in model stability along boundaries not parallel or at 45° to the X and Y axes of the grid. Using this method, boundaries and 2D/1D interfaces can be orientated at any angle. Further improvements to this method were made in Build 2005-10-AH. The ON PRE 2005-10-AH option provides backward compatibility if required.

ON METHOD 2 activates Oblique Boundary Method 2 in Syme (1991). This method does not perform as well as Method 1 and is not recommended.

The OFF option disables any Oblique Boundary Methods.

Rainfall Boundaries**== [{STEPPED} | SMOOTHED | SMOOTHED TIME CENTRED]**

(Optional)

SMOOTHED TIME CENTRED is the default for the 2009-07 release and is the same approach used in the 2008-08 and earlier releases. This option “smooths” the rainfall histogram by converting it into a “hydrograph” shaped curve rather than a histogram shape as is usually adopted for rainfall hyetographs. If TIME CENTRED is omitted (ie. SMOOTHED option), 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.

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).

The STEPPED option is the default for the 2010-10 release, but is available as an option for the 2009-07 release.

Rainfall Gauges == [One per Cell | {Unlimited per Cell}]

(Optional)

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 2070 is issued if a 2D cell is assigned rainfall from more than one RF input (eg. from overlapping RF polygons) as this 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.

Incorporated into the 2009-07 release.

Unused HX and SX Connections == [{ERROR} | WARNING]

(Optional)

If set to “ERROR”, the default, any unconnected or redundant CN objects in 2d_bc layers are treated as an ERROR. This error is typically due to a CN object not being snapped to a HX or SX object in the same 2d_bc layer, or the use of two CN objects at either end of a SX line (only one CN object is required to connect a SX line, thereby making the other one redundant). This error check was incorporated in Build 2003-06-AC. For backward compatibility, set to “WARNING” so that TUFLOW continues to run, but only issues a WARNING. It is not recommended that the WARNING option be used other than for backward compatibility.

A.17 External 1D Schemes Commands (.tcf)

Read GIS ISIS Network == <gis_layer> Read GIS XP Network == <gis_layer>	A-54
Read GIS ISIS Nodes == <gis_layer> Read GIS XP Nodes == <gis_layer>	A-54
Read GIS ISIS WLL == <gis_layer> Read GIS XP WLL == <gis_layer>	A-55
Read GIS ISIS WLL Points == <gis_layer> Read GIS XP WLL Points == <gis_layer>	A-55
Write X1D Check Files == [ON {OFF}]	A-55

Read GIS ISIS Network == <gis_layer>

Read GIS XP Network == <gis_layer>

(Optional)

Reads the location of ISIS or XP-SWMM 1D nodes and channels (Units in ISIS; Links in XP-SWMM). The nodes are required for linking ISIS or XP-SWMM to TUFLOW. The channels are required if using [Read GIS ISIS WLL](#) to integrate 1D and 2D results in the map output. The only attribute required is the ID of the ISIS node/unit or XP-SWMM node/link. See Section 4.10.5.2 for details on linking ISIS or XP-SWMM to TUFLOW, and Section 4.11 for integrating 1D and 2D results.

For linking ISIS or XP-SWMM with TUFLOW, the linked nodes must occur in this layer or in a [Read GIS ISIS Nodes](#) or [Read GIS XP Nodes](#) layer. The nodes and channels (links) can be placed in separate layers.

Note: As of Build 2007-07-AA, ISIS IDs are case sensitive (because ISIS is case sensitive), therefore, the ID in ISIS and the IDs in the 1d_x1d layer(s) must be identical (including case).

Read GIS ISIS Nodes == <gis_layer>

Read GIS XP Nodes == <gis_layer>

(Optional)

Reads the location of ISIS or XP-SWMM 1D nodes. The only attribute required is the ISIS/XP-SWMM ID of the node, which must be identical to that in the ISIS or XP-SWMM model. See Section 4.10.5.2 for details on linking ISIS or XP-SWMM to TUFLOW. This command is now somewhat redundant as the nodes can be placed in a [Read GIS ISIS Network](#) or [Read GIS XP Network](#) layer, although this command can be used if only linking with TUFLOW and not using the WLL feature.

Note: As of Build 2007-07-AA, ISIS IDs are case sensitive (because ISIS is case sensitive), therefore, the ID in ISIS and the IDs in the 1d_x1d layer(s) must be identical (including case).

```
Read GIS ISIS WLL == <gis_layer>
```

```
Read GIS XP WLL == <gis_layer>
```

(Optional)

Reads the location of ISIS or XP-SWMM 1D WLLs. See Section 4.11 for integrating 1D and 2D results. The GIS layer is identical to that used for [Read GIS WLL](#).

```
Read GIS ISIS WLL Points == <gis_layer>
```

```
Read GIS XP WLL Points == <gis_layer>
```

(Optional)

Reads the location of ISIS or XP-SWMM 1D WLL points for setting elevations and materials at points along WLLs. See Section 4.11 for integrating 1D and 2D results for more details. The GIS layer is identical to that used for [Read GIS WLL Points](#).

```
Write X1D Check Files == [ ON | {OFF} ]
```

(Optional)

If set to ON, writes out check_x1D_H_to_2D.csv and check_2D_Q_to_x1D.csv files that contain the water levels sent to the 2D and the flows sent by the 2D to/from an external 1D scheme such as ISIS or XP-SWMM.

Note that [Write Check Files](#) must be also be specified (and not set to OFF).

A.18 Cyclone/Hurricane and Wind Commands (.tcf)

Apply Wind Stresses == [ON ON VARIABLE {OFF}]	A-56
Density of Air == [{1.25} <value>]	A-56
Density of Water == [{1025} <value>]	A-56
Read GIS Cyclone [{} NO PRESSURE] [{} NO WIND] Read GIS Hurricane [{} NO PRESSURE] [{} NO WIND] == <gis_layer>	A-57
Start Wind Output at Time == <time_in_hours>	A-57
Wind Output Interval == <time_in_seconds>	A-58
Wind/Wave Shallow Depths == [{0.2, 1.0} <y1, y2>]	A-58

Apply Wind Stresses == [ON | ON VARIABLE | {OFF}]
(Optional)

Unsupported feature – yet to be set up and tested on PC version.

Density of Air == [{1.25} | <value>]
(Optional)

Sets the density of air in kg/m³. If a cyclone/hurricane track is used, the density of air can be varied along the track.

Density of Water == [{1025} | <value>]
(Optional)

Sets the density of water in kg/m³.

```
Read GIS Cyclone [ {} | NO PRESSURE ] [ {} | NO WIND ]
Read GIS Hurricane [ {} | NO PRESSURE ] [ {} | NO WIND ]
== <gis_layer>
(Optional)
```

Reads a cyclone or hurricane track. The attributes of the GIS layer are listed in the table below and are all float values. Only the first polyline in the layer is read and used for the track. Points are snapped to the line wherever attribute data are to be assigned. It is not necessary to have points snapped to every vertex of the line – values will be interpolated between the digitised points. There must be points with attribute data snapped to the start and end of the polyline track.

The optional NO PRESSURE and NO WIND options deactivate the pressure/wind fields respectively.

Attribute	Description	Type
Time	Time in hours	Float
p0	Pressure at the eye (hPa)	Float
pn	Pressure of surrounds (hPa)	Float
R	Radius to maximum winds (m)	Float
B	See reference below	Float
rho_air	Density of Air (kg/m ³). If zero, Density of Air is used.	Float
km	See reference below. If zero, the formula based on wind speed in the reference below is used.	Float
ThetaMax	See reference below	Float
DeltaFM	See reference below	Float
bw_speed	Background wind speed in m/s (ignored if less than or equal to zero)	Float
bw_dirn	Background wind direction in degrees relative to East (0°), North (90°), etc.	Float

The generation of the wind and pressure fields are based on Appendix C of “Queensland Climate Change and Community Vulnerability to Tropical Cyclones – Ocean Hazards Assessment – Stage 1” Queensland Government, March 2001. The wind and pressure fields can be output using the W110 and AP options for [Map Output Data Types](#).

The background wind attributes were added for Build 2007-01-AB and need to be added to any existing GIS layers if this build or later is to be used. The background wind is applied outside R (the radius of maximum winds), if it exceeds the cyclone/hurricane wind speed.

Note that [Latitude](#) must be specified so as to distinguish between southern and northern hemispheres.

```
Start Wind Output at Time == <time_in_hours>
(Optional)
```

Unsupported feature – yet to be set up and tested on PC version.

Wind Output Interval == <time_in_seconds>

(Optional)

Unsupported feature – yet to be set up and tested on PC version.

Wind/Wave Shallow Depths == [{0.2, 1.0} | <y1, y2>]

(Optional)

Sets the depths of water when the wind and/or wave stress is reduced to zero. This command is necessary to avoid a divide by zero, and model instabilities when high wind/wave stresses are applied to zero or very shallow depths. Below y1, the stress is set to zero, above y2 the full stress is applied, and between y1 and y2 the stress is interpolated. y1 and y2 are in metres.

A.19 Wave Radiation Stress Commands (.tcf)

Apply Wave Radiation Stresses == [ON | {OFF}]

A-59

Wave Period == <period_in_seconds>

A-59

Apply Wave Radiation Stresses == [ON | {OFF}]

(Optional)

Unsupported feature – yet to be set up and tested on PC version.

Wave Period == <period_in_seconds>

(Optional)

Unsupported feature – yet to be set up and tested on PC version.

Appendix B .ecf File Commands

[Apply All Inverts](#)

[BC Database](#)

[BC Event Name](#)

[BC Event Text](#)

[BG](#)

[BG Data](#)

[Bridge Flow](#)

[Check MI Save Date](#)

[Check MI Save Ext](#)

[Create Nodes](#)

[CS](#)

[CS Data](#)

[CSV Format](#)

[CSV Time](#)

[Conveyance Calculation](#)

[Culvert Add Dynamic Head](#)

[Culvert Critical H/D](#)

[Culvert Flow](#)

[Defaults](#)

[Depth Limit Factor](#)

[EB Data](#)

[End Time](#)

[Flow Area](#)

[Flow Calculation](#)

[Froude Check](#)

[Froude Depth Adjustment](#)

[Head Rate Creep Factor](#)

[Head Rate Limit](#)

[Head Rate Limit Minimum](#)

[Interpolate Cross-Sections](#)

[Interpolate Culvert Inverts](#)

[Log Folder](#)

[M11 Network](#)

[MI Projection](#)

[Manhole Default Loss](#)

[Approach](#)

[Manhole Default C Exit](#)

[Coefficient](#)

[Manhole Default R Exit](#)

[Coefficient](#)

[Manhole Default Side](#)

[Clearance](#)

[Manhole Default Type](#)

[Manhole K Maximum](#)

[Bend/Drop](#)

[Manhole Minimum](#)

[Dimension](#)

[Manholes at All Culvert](#)

[Junctions](#)

[Minimum Channel](#)

[Conveyance Length](#)

[Minimum Channel Storage](#)

[Length](#)

[Minimum NA](#)

[Minimum NA Pit](#)

[Momentum Equation](#)

[NA](#)

[NA Data](#)

[Order Output](#)

[Output Folder](#)

[Output Interval](#)

[Output Times Same as 2D](#)

[Pit Channel Offset](#)

[Pit Default 2D Connection](#)

[Pit Inlet Database](#)

[Pit Search Distance](#)

[Read File](#)

[Read Materials File](#)

[Read GIS BC](#)

[Read GIS IWL](#)

[Read GIS Manhole](#)

[Read GIS Network](#)

[Read GIS Table Links](#)

[Read GIS WLL](#)

[Read GIS WLL Points](#)

[Relative Resistance](#)

[S Channel Approach](#)

[Set IWL](#)

[Snap Tolerance](#)

[Start Output](#)

[Start Time](#)

[Storage Above Structure](#)

[Obvert](#)

[Structure Losses](#)

[Structure Losses Approach](#)

[Taper Closed NA Table](#)

[Timestep](#)

[Trim XZ Profiles](#)

[Vel Rate Creep Factor](#)

[Vel Rate Limit](#)

[Vel Rate Limit Minimum](#)

[VG](#)

[VG Data](#)

[Weir Flow](#)

[WLL Additional Points](#)

[WLL Adjust XS Width](#)

[WLL Approach](#)

[WLL Automatic](#)

[WLL No Weirs](#)

[WLL Vertical Offset](#)

[WLLp Interpolate Bed](#)

[Write CSV Online](#)

[Write Check Files](#)

[Write Empty MI Files](#)

[XS Database](#)

[Zero Culvert Coefficients](#)

B.1 Geographic Reference Commands (.ecf)

MI Projection == [<.mif file> | <Projection_line_from_MIF_file>] **B-2**

Snap Tolerance == [{0.001} | <value_in_metres>] **B-2**

**MI Projection == [<.mif file> |
<Projection_line_from_MIF_file>]**
(1D Only. Optional but recommended)

Same as for the .tcf file – see [MI Projection](#).

Snap Tolerance == [{0.001} | <value_in_metres>]
(1D Only. Optional)

Same as for the .tcf file – see [Snap Tolerance](#).

B.2 File Management Commands (.ecf)

Check MI Save Date == [{ERROR} WARNING OFF]	B-2
Check MI Save Ext == [{.tab} <ext>]	B-2
CSV Format == [HORIZONTAL {VERTICAL}]	B-2
CSV Time == [{DAYS} HOURS]	B-3
Log Folder == <folder>	B-3
Output Folder == <folder>	B-3
Read File == <file>	B-3
Write CSV Online == [ON {OFF}]	B-3
Write Empty MI Files == [{} <folder>]	B-4
Write Check Files == [<file_prefix> {OFF}]	B-4

Check MI Save Date == [{ERROR} | WARNING | OFF]

(1D Only. Optional)

Same as for TUFLOW – see [Check MI Save Date](#).

Check MI Save Ext == [{.tab} | <ext>]

(1D Only. Optional)

Same as for TUFLOW – [Check MI Save Ext](#).

CSV Format == [HORIZONTAL | {VERTICAL}]

(1D & 2D/1D. Optional)

If set to HORIZONTAL, writes the 1D .csv output file with the head/flow/velocity values for a node/channel in rows. The default is to write the values in columns.

CSV Time == [{DAYS} | HOURS]

(1D & 2D/1D. Optional)

If set to HOURS, writes out time values in hours rather than days. For 2D/1D models specifying this command in the .tcf file will also apply to the 1D .csv output – see [CSV Time](#).

Log Folder == <folder>

(1D Only. Optional)

Redirects the .elf and _messages.mif file output to the specified folder. Typically used to write these files to a folder named log under the runs folder.

Output Folder == <folder>

(1D & 2D. Optional)

Redirects all ESTRY output data except the .elf file to another folder. Typically used to write output to your local C: or D: drive instead of filling up the network or to keep results separate to the input data. A URL path can be used (eg. [\\wbmserv\Computer001\tuflow\results](#)); useful for running simulations on other computers, but having the output directed to your local drive or other location (your drive will need to be shared).

As of Build 2004-05-AF, the default location for 1D output is that specified using [Output Folder](#) in the .tcf file for 2D/1D models.

Read File == <file>

(1D & 2D/1D. Optional)

Directs input to another file. When finished reading <file>, ESTRY returns to reading the .ecf file.

This command is particularly useful for projects with a large number of simulations. Repetitive commands are grouped and placed in another text file. If one of these commands changes, the command only has to be edited once, rather than in every .ecf file.

NOTE: As of Build 2002-03-AA, this command can now be used in redirected file(s) up to a maximum of ten levels.

Write CSV Online == [ON | {OFF}]

(1D Only. Optional)

For 1D only models, if set to ON, writes the 1D .csv (and also _TS.mif and _mm_.mif) output files at each output time allowing monitoring of the 1D results during the simulation. The default is OFF.

For 2D/1D models use [Write PO Online](#) in the .tcf file.

Write Empty MI Files == [{} | <folder>]

(1D Only. Optional)

Creates empty 1D GIS files in MIF/MID format useful for setting up new GIS layers. Each 1D layer as described in Table 2.3 is produced with the required attribute definitions pre-defined, but containing no geographic objects. Provided the [MI Projection](#) command has been previously specified, each layer has the correct GIS projection.

The layers are prefixed using the prefixes defined in Table 2.3 and are given a suffix of “_empty”. If <folder> is specified, the .mif/.mid files are located in the folder, which must already exist.

After writing the files, ESTRY stops executing.

Example:

```
Write Empty MI Files == ..\model\mi\empty
```

Write Check Files == [<file_prefix> | {OFF}]

(1D & 2D/1D. Optional)

Creates GIS check files in MIF/MID format and text .csv files for quality control checking of model input data. Prior to Build 2002-11-AA, “Write MI Check Files” was used and may continue to be used for backward compatibility. Also see [Write Check Files](#) for further discussion.

Some of the files produced are:

- Of the entire network after all network input files have been read. The mif/mid files have a “_nw” appended to <file_prefix>.
- Of all boundary conditions. The mif/mid files have a “_bc” appended to <file_prefix>.
- Of the initial water levels. The mif/mid files have a “_iwl” appended to <file_prefix>.
- *_1d_bc_tables_check.csv file containing any tables read by 1d_bc layers.
- *_1d_ta_tables_check.csv file containing any tables read by 1d_tab layers.

<file_prefix> can include a folder path which is normally set to the check folder. If <file_prefix> is omitted, the .ecf filename is used (without the .ecf extension).

The OFF option deactivates any previously specified Write Check Files command - no check files will be created. If the command is never specified, the OFF option applies.

Examples:

```
Write Check Files == ..\check\1d ! writes check files to the folder  
“..\check” and prefixes with “1d”
```

```
Write Check Files == ..\check\ ! writes check files to the folder  
“..\check” and prefixes with the .ecf filename
```

```
Write Check Files == ..\check ! writes check files to the folder  
one level up and prefixes with “check”
```

B.3 Simulation Control Commands (.ecf)

Bridge Flow == [Method A {Method B}]	B-6
Culvert Add Dynamic Head == [ON {OFF}]	B-6
Culvert Critical H/D == [{OFF} <critical_h/d>] Culvert Critical H/D == [{OFF} <v1>, <v2>, <v3>]	B-6
Culvert Flow == Method [A B C D {E}]	B-7
Defaults == [PRE 2006-06-AA PRE 2007-07-AA]	B-7
End Time == <time_in_hours>	B-7
Flow Calculation == [Method A {Method B}]	B-7
Froude Check == [{1} <froude_no>]	B-7
Froude Depth Adjustment == [{ON} OFF]	B-8
Head Rate Creep Factor == [<value> {1.2}]	B-8
Head Rate Limit == [ON {OFF} <hrl>]	B-8
Head Rate Limit Minimum == [<hrlmin> {0.001}]	B-8
Start Time == <time_in_hours>	B-8
Structure Losses == [{ADJUST {EXCEPT BG TABLE}} FIX]	B-9
Structure Losses Approach == [METHOD A {METHOD B}]	B-9
Timestep == <timestep_in_seconds>	B-9
Vel Rate Creep Factor == [<value> {1.2}]	B-9
Vel Rate Limit == [<vrl> {0.2}]	B-10
Vel Rate Limit Minimum == [<vrlmin> {0.0001}]	B-10
Weir Flow == [Method A {Method B}]	B-10

Bridge Flow == [Method A | {Method B}]

(1D & 2D/1D. Optional)

Controls the method for calculating bridge flows. Method A uses the original ESTRY bridge routine. Method B, introduced at Build 2005-05-AN, is based on Method A, but provides improved stability particularly when the bridge is flowing at shallow depths or is wetting and drying. Method B also does not force the loss coefficient to be a minimum of 1.5625 once the bridge obvert is surcharged (Method B uses the value as specified in the BG table).

Testing of Method B indicates that it is the preferred approach, and was changed to the default in Build 2006-03-AB. Use Method A for backward compatibility.

Culvert Add Dynamic Head == [ON | {OFF}]

(1D & 2D/1D. Optional)

For [Culvert Flow](#) == Method C, includes an improved method for allowance of the dynamic head upstream and downstream of culverts, so that there is better transitioning between the different culvert flow regimes. Introduced as the default in Build 2006-03-AB.

For [Culvert Flow](#) == Method D, the above benefits became redundant and the default setting for this command was set to OFF as introduced for Build 2007-07-AC. Substantial improvements in culvert stability are achieved using [Culvert Flow](#) == Method D and [Culvert Add Dynamic Head](#) == OFF.

Culvert Critical H/D == [{OFF} | <critical_h/d>]**Culvert Critical H/D == [{OFF} | <v1>, <v2>, <v3>]**

(1D & 2D/1D. Optional)

Sets the H/D value to be used for determining whether outlet control Regimes E and F take preference over the inlet control Regimes B or L. H is the upstream head above the culvert sill and D is the culvert height. If H/D exceeds <critical_h/d> Regime E or F is used, otherwise the regime with the lower discharge (along with other tests) is used.

For Build 2007-07-AA, the default is OFF (ie. infinitely large H/D). In previous builds a value of 1.5 was hard-wired into the code, therefore, use [Culvert Critical H/D](#) == 1.5 for backward compatibility.

As of Build 2008-08-AA, can optionally now have three values as follows:

- v1 = Critical H/D value at culvert inlets connected to at least one open channel.
- v2 = Critical H/D value at culvert inlets connected to a pit channel (and not connected to any open channels).
- v3 = Critical H/D value at culvert inlets at culvert only junctions (no open channels or pits).
- Default values are 99999. (ie. no critical H/D values applied). If only one value is specified, this is applied to all three. Specifying OFF applies 99999 to all three values.

Culvert Flow == Method [A | B | C | D | {E}]

(1D & 2D/1D. Optional)

Controls the method for calculating culvert flows. Method A is the original ESTRY culvert routines. Method B, first incorporated at Build 2002-07-AC, is an adaptation of Method A to include new regimes K and L (see Section 4.7.4.3). Method B also offers improved stability, smoother transitions between flow regimes and corrects very occasional mass conservation errors under certain flow regimes. Method C, introduced for Build 2006-03-AB, is a slight improvement on Method B for flow Regime C (see Figure 4-7).

As of Build 2008-08-AA, Method E is the default. Method E incorporates minor improvements for transitioning between Regimes A and B, and between inlet and outlet controlled regimes, for circular culverts. If [Defaults](#) == PRE 2008-08 is specified, Method D is the default setting. Methods A, B, C and D are provided for backward compatibility.

For further discussion see Section 4.7.4.3.

Defaults == [PRE 2006-06-AA | PRE 2007-07-AA]

(Optional)

Sets the default settings to those prior to Build 2006-06-AA and Build 2007-07-AA for 1D only models – see [Defaults](#) for details.

End Time == <time_in_hours>

(1D Only. Mandatory)

Specifies the finish time of the simulation in hours. Value must be greater than the start time and can be negative.

Flow Calculation == [Method A | {Method B}]

(Optional)

Method B corrects an anomaly that would sometimes incorrectly output 1D flow values where the channel flow regimes are oscillating every half timestep (for example, between super and sub-critical flow regimes). Where the channel is switching flow regimes between timesteps (nearly always the case), the correct flow is calculated. This fix also affects the flow in/out of 2D SX connections if the connected 1D channel is oscillating every half timestep. The fix does not change 1D water level and velocity results, unless they are influenced by changes due to any effects on SX flows. Use Method A only for backward compatibility.

Froude Check == [{1} | <froude_no>]

(Optional)

Sets the minimum Froude Number that upstream controlled friction flow may occur in “S” channels. Improved stability may occur in steeply flowing areas if <froude_no> is less than 1. <froude_no> cannot be below zero and would normally not exceed 1.

Froude Depth Adjustment == [{ON} | OFF]

(Optional)

Switches on or off an additional upstream controlled friction flow check for S channels incorporated in Build 2003-01-AF (a similar check is used for 2D domains – see Section 4.7.3). Set to OFF for backward compatibility for models run prior to Build 2003-01-AF that use the upstream controlled friction feature (ie. “S” channels).

Head Rate Creep Factor == [<value> | {1.2}]

(1D & 2D/1D Only. Optional)

Specifies rate at which the [Head Rate Limit](#) value changes.

Head Rate Limit == [ON | {OFF} | <hrl>]

(1D & 2D/1D Only. Optional)

Specifies the head rate limit applied to nodes. This feature can be used to stabilise problematic 1D nodes, but should be used with caution and mass balance checks must be made to ensure there is no significant mass loss or gain. It is particularly useful where a node “bounces” temporarily and is prevented from becoming unstable. The maximum amount the water level can change in half a timestep is the <hrl> value after any adjustment by the [Head Rate Creep Factor](#). The <hrl> is adjusted up and down depending on the stability of the node in a similar approach used for [Vel Rate Limit](#). If the ON option is used, the <hrl> value is set to 0.1.

Head Rate Limit Minimum == [<hrlmin> | {0.001}]

(1D & 2D/1D Only. Optional)

Specifies the minimum head rate limit that can occur. See [Head Rate Limit](#).

Start Time == <time_in_hours>

(1D Only. Mandatory)

Specifies the start time of the simulation in hours. Value can be negative and it is recommended that it be relative to midnight for historical events.

Structure Losses == [{ADJUST {EXCEPT BG TABLE}} | FIX]

(1D & 2D/1D Only. Optional)

If set to ADJUST, the entrance and exit losses of culverts and the bridge loss coefficients are adjusted according to the approach and departure velocities upstream and downstream of the structure. See Section 4.7.4.1 for details. Introduced in Build 2005-05-AN. As of Build 2006-03-AB, the default was changed from FIX to ADJUST (set to FIX for backward compatibility).

As of Build 2007-07-AA, this setting can be overridden using an A (adjust) or F (fix) flag for B, C and R channels (see Table 4.14).

Prior to Build 2007-07-AA, if set to ADJUST, this may cause excessively high flows for flow Regime E (see Section 4.7.4.3).

New option “EXCEPT BG TABLE” and default for the 2010-10 release is to only adjust losses for culverts and automatically generated BG tables noting that in the latter it is only the deck loss component that is adjusted as the component specified via the 1d_nwk Form_Loss attribute is always treated as fixed. Any manually specified BG tables are not adjusted unless the “A” flag is used in the 1d_nwk Type attribute or “Structure Losses == ADJUST” is used without specifying “EXCEPT BG TABLE”.

Structure Losses Approach == [METHOD A | {METHOD B}]

(1D & 2D/1D Only. Optional)

METHOD B was incorporated into the 2010-10 release and is an enhancement over the previous approach covered by METHOD A. METHOD B is the same as METHOD A, except that if reverse flow occurs in the approach or departure primary channels (reverse flow is when flow is in the opposite direction to the channel’s digitised direction), the adjustment of entrance and exit losses is based on the flow direction, not the digitised direction.

Timestep == <timestep_in_seconds>

(1D & 2D/1D Only. Mandatory for 1D only.)

Specifies the computation timestep of the simulation in seconds. Value must be greater than zero. Timesteps that divide equally into one minute are recommended. For example, 0.5, 1, 2, 3, 5, 6, 7.5, 10, 12, 15, 20, 30, 45, 60, etc. seconds.

As of Build 2005-05-AN, this command can be specified for 2D/1D models and can be different to the timesteps of the 2D domains, but must not be greater than the smallest timestep of the 2D domains. If the 1D timestep is not equally divisible into the smallest 2D timestep, the 1D timestep is reduced automatically so that it is equally divisible. If this command is not specified in the .ecf file, the smallest 2D timestep is used.

Vel Rate Creep Factor == [<value> | {1.2}]

(1D & 2D/1D Only. Optional)

Specifies rate at which the [Vel Rate Limit](#) value changes. This value is rarely changed from its default value of 1.2. See [Vel Rate Limit](#) for further discussion.

Note, in early versions of TUFLOW, this command was referred to as just Creep Factor.

Vel Rate Limit == [<vrl> | {0.2}]

(1D & 2D/1D Only. Optional)

Specifies the velocity rate limit applied to non-inertial channels (structures). This value is rarely changed from its default value of 0.2. During a computation this value is adjusted downwards if a structure becomes unstable and upwards if stable using the [Creep Factor](#) value. In Build 2003-07-AF, an “L” is shown in the second space after velocity and flow time output in the .ecf file, and also in the _TSF.mif output, indicating if the velocity rate limit algorithm was applied (previous builds used an asterisk (“*”). If a structure frequently has the velocity rate limit applied to it, checks should be made on structure configuration and on the results at the structure. For example, check the flow through the structure based on the upstream and downstream water levels is similar to that using desktop calculations or other software.

Vel Rate Limit Minimum == [<vrlmin> | {0.0001}]

(1D & 2D/1D Only. Optional)

Specifies the minimum velocity rate limit that can occur. See [Vel Rate Limit](#). Prior to Build 2003-08-AD, the velocity rate limit could, in rare cases, reach zero and “freeze” the structure, giving unrealistic results.

Weir Flow == [Method A | {Method B}]

(1D & 2D/1D. Optional)

Method B provides improved stability of weir flows when the head drop across a weir is very small. Method A is the original approach and is that used if [Defaults](#) == PRE 2008-08.

B.4 Output Control and Format Commands (.ecf)

Order Output == [{ON} OFF]	B-11
Output Interval [{} (s)] == <time>	B-11
Output Times Same as 2D == [{ON} OFF]	B-12
Read GIS WLL == <gis_layer>	B-12
Read GIS WLL Points == <gis_layer>	B-12
Start Output == <time_in_hours>	B-12
WLL Additional Points == [{0} <value>]	B-12
WLL Adjust XS Width == [{ON} OFF]	B-13
WLL Approach == [Method A {Method B}]	B-13
WLL Automatic == [CULVERTS {OFF}]	B-13
WLL No Weirs == [ON {OFF}]	B-13
WLL Vertical Offset == [<dZ> {10}]	B-13
WLLp Interpolate Bed == [{ON} OFF]	B-14
WLLp Interpolate Bed == [{ON} OFF]	B-14

Order Output == [{ON} | OFF]

(1D & 2D/1D. Optional)

Alphanumerically orders 1D output according to the node and channel IDs. The exception is the boundary condition data in the .ecf file.

Output Interval [{} | (s)] == <time>

(1D & 2D/1D. Optional)

The output interval for ESTRY output. The default units are hours, however, seconds may be used if the “(s)” option is specified. If the command is omitted, output is at every computational timestep.

Output Times Same as 2D == [{ON} | OFF]

(2D/1D Only)

For 2D/1D models, as of Build 2003-06-AA, the times for 1D output are, by default, the same as that of the 2D domain(s) time series output (see [Start Time Series Output](#) and [Time Series Output Interval](#)), unless the no 2D time series output (2d_po layers) has been specified, in which case [Start Output](#) and [Output Interval](#) are used. For backward compatibility or to use different times for 1D time series output, set to OFF.

This change was made so that both 1D and 2D time series data could be output to the _TS.mif file, allowing graphing of 1D and 2D time series data within a GIS (see Section 7.4.4).

This command is ignored for 1D only (ESTRY) models.

Read GIS WLL == <gis_layer>

(2D/1D. Optional)

Reads water level lines (WLL) for defining 1D map output for viewing in SMS and a GIS. See Section 4.11 for further information.

Read GIS WLL Points == <gis_layer>

(2D/1D. Optional)

For [WLL Approach](#) Method B, reads elevation and material points generated from the WLLs. This allows more accurate velocity and flood hazard mapping. See Section 4.11.2 for further information.

Start Output == <time_in_hours>

(1D & 2D/1D. Optional)

The simulation time in hours when output commences. If the command is omitted, the simulation start time is used.

WLL Additional Points == [{0} | <value>]

(2D/1D. Optional)

[WLL Approach](#) Method A only. Sets the number of additional points to be used in creating SMS mesh elements for 1D map output. The number of additional points is twice the value specified, as the additional points are placed on both sides of the WLL mid-point. See Section 4.11 for further information.

WLL Adjust XS Width == [{ON} | OFF]

(2D/1D. Optional)

[WLL Approach](#) Method A only. If set to ON (the default), the location of additional WLL points (see [WLL Additional Points](#)) is adjusted proportionally according to the length of the WLL side. If set to OFF, the location of additional points is based on the true width of flow as determined from the channel hydraulic properties table (as produced in the .ecf file). Setting this option to OFF is useful if the true width of flow (based on the channel cross-section) is to be viewed in SMS for quality control checks.

WLL Approach == [Method A | {Method B}]

(2D/1D. Optional)

If set to Method A uses the simpler approach for incorporating 1D output into SMS and GIS map output. Method B allows the use of elevation points and material values to more accurately map and animate 1D results. See Section 4.11 for details.

Method B was made the default in Build 2006-03-AB.

WLL Automatic == [CULVERTS | {OFF}]

(2D/1D. Optional)

If set to CULVERTS, automatically generates 1D WLLs along culverts (C and R channels). The WLL will have the same width as the culvert width, and can save a lot of digitising for large pipe models! WLLs are placed a short distance from each end of the culvert channel, and also at each vertice along the channel line.

WLL No Weirs == [ON | {OFF}]

(2D/1D. Optional)

If set to ON, TUFLOW will not assign any WLLs to 1D weir channels. This is useful where weir channels modelling flow over a bridge or culvert (especially those using the BW, CW or RW channel type) is in parallel to a B, C or R channel. In this instance, it is not known whether the B, C or R channel, or the W channel will be selected for assigning results to WLLs. To guarantee that the B, C or R channel is selected use this command with the ON option.

WLL Vertical Offset == [<dZ> | {10}]

(2D/1D. Optional)

Sets the vertical adjustment of WLL elevations in the SMS .2dm file. The value of 10 generally means that the 1D WLL output sits above the 2D cell output and is more visible in SMS (which presents data in order of increasing height). However, in 3D the 1D appears perched on top of the 2D. To show the WLL mesh at its correct height for 3D displays specify a dZ value of 0.0.

This command only effects the .2dm file, so can be applied to a temporary .tcf file used solely to generate an alternative .2dm file (ie. there is no need to carry out the hydraulic calculations).

WLLp Interpolate Bed == [{ON} | OFF]

(2D/1D. Optional)

If set to ON (the default), sets the centre WLL point to the channel bed based on the processed data (rather than use any value from a WLLp layer). This forces the bed profiles in longitudinal profile plots using the -lp switch in TUFLOW_to_GIS to be based on that modelled, rather than that a DTM using WLLp values (which may sometimes occur above the water surface!). Also helps show where the WLLp elevations are inconsistent with the channel bed when viewing in SMS or using TUFLOW_to_GIS.

Doesn't apply to culverts and bridges which use this approach regardless, and only applies to [WLL Approach](#) == Method B.

Introduced for Build 2007-07-AA.

WLLp Interpolate Bed == [{ON} | OFF]

(2D/1D. Optional)

If set to ON (the default), sets the centre WLL point to the channel bed based on the processed data (rather than use any value from a WLLp layer). This forces the bed profiles in longitudinal profile plots using the -lp switch in TUFLOW_to_GIS to be based on that modelled, rather than that a DTM using WLLp values (which may sometimes occur above the water surface!). Also helps show where the WLLp elevations are inconsistent with the channel bed when viewing in SMS or using TUFLOW_to_GIS.

Doesn't apply to culverts and bridges which use this approach regardless, and only applies to [WLL Approach](#) == Method B.

Introduced for Build 2007-07-AA.

B.5 Model Network and Topography Commands (.ecf)

Apply All Inverts == [{ON} OFF]	B-16
Conveyance Calculation == [CHANGE IN RESISTANCE {ALL PARALLEL}]	B-16
Create Nodes == [{ON} OFF]	B-16
Depth Limit Factor == [{10} <value>]	B-17
Flow Area == [{EFFECTIVE} TOTAL]	B-17
Interpolate Cross-Sections == [{ON} OFF]	B-18
Interpolate Culvert Inverts == [{ON} OFF]	B-18
M11 Network == <.nwk11_file>	B-18
Minimum Channel Conveyance Length == [{0} <length_m>]	B-18
Minimum Channel Storage Length == [{0} <length_m>]	B-18
Minimum NA [{} (%)] == [{1} <value>]	B-19
Minimum NA Pit == [{1} <value>]	B-19
Momentum Equation == [PRE 2003-08-AD]	B-19
Pit Channel Offset == [{10} <length_m>]	B-19
Pit Inlet Database == [<pit_inlet_dbase.csv>]	B-20
Pit Search Distance == [{0.0} <psd_m>]	B-20
Read GIS Network == <gis_layer>	B-20
Read GIS Table Links == <gis_layer>	B-20
Read Materials File == <file>	B-20
Relative Resistance == [{RELATIVE} MATERIAL]	B-21
S Channel Approach == [PRE 2004-06-AA METHOD A {METHOD B}]	B-21
Storage Above Structure Obvert [{(%) }] == [CHANNEL WIDTH <value> {5}]	B-21
Taper Closed NA Table == [ON {OFF}]	B-22

Trim XZ Profiles == [ON | {OFF}] B-22

XS Database == <file> B-22

Zero Culvert Coefficients == [DEFAULT | {OFF} | v1, v2, v3, v4, v5] B-23

Apply All Inverts == [{ON} | OFF]

(1D & 2D/1D. Optional)

If set to ON, applies the upstream and downstream inverts specified in the 1d_nwk layer to all channels (ie. also for B, Blank and W channels), except where a value of -99999 is specified. Introduced for Build 2005-05-AN and made the default in Build 2006-03-AB.

Note: If upgrading a model used prior to Build 2006-03-AB, any inverts for B, Blank and W channels need to be set to -99999 to ensure the inverts remain unchanged. Otherwise, the inverts for these channels are likely to be set to zero, as this is the default value set by MapInfo for the invert attributes.

Conveyance Calculation == [CHANGE IN RESISTANCE | {ALL PARALLEL}]

(1D & 2D/1D. Optional)

If set to CHANGE IN RESISTANCE, the parallel channel analysis splits the cross-section into separate parallel channels based on wherever there is a change in resistance (due to different relative resistance, material type or Manning's n values).

If set to ALL PARALLEL, a parallel channel is created for every X (distance across section) value. This approach does not cause conveyance reducing with height warnings.

Introduced in Build 2005-05-AN. ALL PARALLEL was made the default as of Build 2006-03-AB.

Create Nodes == [{ON} | OFF]

(1D & 2D/1D. Optional)

If no node is found snapped to the end of a channel a new node is automatically created. The ID of the node is the first ten characters of the channel ID with a ".1" or ".2" extension. ".1" is used if the node is at the start of the channel and ".2" if at the end. If more than one channel is connected to the created node, the channel ID that occurs first alphanumerically is used.

The automatic creation of nodes can be switched off using the OFF option. This option may be desirable for models developed prior to Build 2002-08-AC when nodes were mandatory.

Depth Limit Factor == [{10} | <value>]

(1D & 2D/1D. Optional)

Sets the depth limit for detecting instabilities. Prior to Build 2006-03-AB, the default value was 1, so if the water level exceeded the highest elevation in a CS or NA table, this was regarded as an instability, and the simulation would stop. The default as of Build 2006-03-AB is set to 10, therefore the water level must exceed ten times the depth of the CS or NA table before an instability is triggered.

Specifying a value greater than one extends the cross-section hydraulic properties and nodal storages above the highest elevation. For example, if a value of 2 is specified, this will allow water levels to reach twice the depth where depth is the difference between the highest and lowest elevations in the table.

Cross-section hydraulic properties above the highest elevation are calculated based on the flow width remaining constant at the width of the highest elevation in the table. If the hydraulic properties are calculated from a cross-section profile, it uses the effective flow width as shown in the .eof file (it does not use the storage width) – this preserves the effect of any variation in relative roughness across the cross-section. All other hydraulic property sources use the storage width, and any relative roughness effects are ignored once the water level exceeds the highest elevation. Also note that the wetted perimeter remains constant above the highest elevation; ie. it is not increased on the vertical as the flood level rises. Cross-section properties of bridge channels are not affected by this command.

Nodal storage properties extend upwards by keeping the surface area constant above the highest elevation in the table.

Flow Area == [{EFFECTIVE} | TOTAL]

(1D & 2D/1D. Optional)

Sets the default method for calculating flow area at a channel cross-section when ESTRY calculates the hydraulic properties from a cross-section XZ profile table. The default is effective area, which means that the flow area is the sum of the areas divided by the relative resistance factor. Total area ignores the relative resistance factor when calculating area, but uses it to set the wetted perimeter and hydraulic radius values. Either method gives the same channel conveyance. If the relative resistance across the profile is not specified or constant at a value of one, effective and total area are the same.

The effective area method produces a velocity that applies to the main channel (where the relative resistance is set to one). The total area approach produces a velocity depth and width averaged, and typically underestimates the main channel velocity. The recommended approach is to use effective area.

See Section 4.6.7 for a more detailed discussion.

Interpolate Cross-Sections == [{ON} | OFF]

(1D & 2D/1D. Optional)

If set to ON (the default), any channels that don't have a cross-section have their cross-section properties interpolated using the nearest cross-sections attached to other channels. See Section 4.6.5 for details on the process used. Set to OFF to disable this feature and force every channel to have a cross-section. Incorporated into Build 2008-08-AA.

Interpolate Culvert Inverts == [{ON} | OFF]

(1D & 2D/1D. Optional)

If set to ON (the default), any culverts that don't have an invert (ie. a -99999 value was assigned to the invert attribute(s) and there were no inverts assigned via any pits or nodes snapped to the culvert ends) have their invert(s) interpolated using the nearest assigned inverts of connected culverts/pipes. Set to OFF to disable this feature. Incorporated into Build 2008-08-AA.

M11 Network == <.nwk11_file>

(1D & 2D/1D. Optional)

Sets the active MIKE 11 network file as <.nwk11_file>. The file is used to extract link cross-section and other information using the Branch, Topo_ID and XSect_ID_or_Chainage attributes as discussed in Table 4.14. Topo_ID must be set to "\$Link".

This command must be specified before the relevant [Read GIS Network](#) command. The command maybe used at any point to change the active MIKE 11 network file.

Minimum Channel Conveyance Length == [{0} | <length_m>]

(1D & 2D/1D. Optional)

Automatically increases the conveyance (and storage) length to <length_m> if the channel's length is less than <length_m>. The default setting is zero (ie. no change to any channel's length). Does not apply to pit channels.

Minimum Channel Storage Length == [{0} | <length_m>]

(1D & 2D/1D. Optional)

Introduced for Build 2007-07-AA. If a channel's length is less than <length_m>, then <length_m> is used for calculating any storage contributions from the channel widths. It does not affect the channels bed resistance, conveyance or slope. Can be useful to add additional storage for stability reasons to nodes at the ends of very short channels. If using this command, care must be taken not to excessively add additional storage to the model that causes the model results to be distorted. Generally, adding an appropriate amount of storage for stability reasons does not distort results, however, it is strongly recommended that sensitivity tests are carried out to cross-check the effect of any additional storage, and that any adverse effects are corrected.

Minimum NA [{} | (%)] == [{1} | <value>]

(1D & 2D/1D. Optional)

Sets the minimum surface area (m^2) in all NA tables (except, as of Build 2007-07-AA, for the upstream (ground) nodes of pit channels). The default value is one (1m^2). This command is useful for stabilising 1D nodes that have very small storages, particularly at shallow depths. If using this command, care must be taken not to excessively add additional storage to the model that causes the model results to be distorted. Generally, adding an appropriate amount of storage for stability reasons does not distort results, however, it is strongly recommended that sensitivity tests are carried out to cross-check the effect of any additional storage, and that any adverse effects are corrected.

For Build 2008-08-AA, the (%) option is provided which sets the minimum NA value for each node based on a percentage of the maximum nodal area value for the node.

Minimum NA Pit == [{1} | <value>]

(1D & 2D/1D. Optional)

Sets the minimum surface area (m^2) of the upstream (ground) nodes of all pit channels. The default value is one (1m^2). This command was introduced for Build 2007-07-AA to differentiate upstream pit channel nodes from the [Minimum NA](#) setting. If the pit channel is connected to a 2D domain, this storage has no influence on the hydraulic computations, and increasing the value has no stability benefits.

Momentum Equation == [PRE 2003-08-AD]

(1D & 2D/1D. Optional)

Sets the treatment of the effective flow width above the top of a cross-section to that prior to Build 2003-08-AD to provide backward compatibility. After this build, the effective flow width at the top of a cross-section is stored and used to extend the effective flow area above the highest point in the cross-section. Prior to this build, the top storage width was used for the effective flow width for flows above the top of the cross-section. This may only affect results where relative resistance varies across a cross-section, and flow occurs above the top of the cross-section, and effective flow area is being used.

Pit Channel Offset == [{10} | <length_m>]

(1D & 2D/1D. Optional)

Sets the display, not actual, length in metres of pit channels in 1D output and the 1d_nwk check files. The channel is displayed on a north to south alignment.

Pit Default 2D Connection == [{} | <Conn_2D>]

(1D & 2D/1D. Optional)

Sets the default value for the 1d_nwk Conn_2D attribute (see Table 4.14). For example, if set to SX, then if the 1d_nwk Conn_2D attribute for a pit is empty, SX will be used, saving the user to specify SX at pits. To disconnect a pit NO can be used for the Conn_2D attribute. Also, it is now possible to

have the L and Z options at the one pit (eg. SXLZ) - previously only a maximum of three characters was allowed.

Pit Inlet Database == [<pit_inlet_dbase.csv>]

(1D & 2D/1D. Mandatory if any Q pit channels.)

Specifies the pit inlet database file that references the depth-discharge curves for Q pit channels. Mandatory if the model has any Q pit channels.

See Section 4.5.1.2 for more information.

Pit Search Distance == [{0.0} | <psd_m>]

(1D & 2D/1D. Optional)

Set the distance (radius) in metres to search for the closest node to automatically connect pits into the 1D network where pits are not snapped to channel ends. Pits connected via this feature are displayed in the 1d_nwk_C_check layer as occurring from the location of the pit to the node, ie. [Pit Channel Offset](#) is not used to display the pit channels created. The pits are also displayed in the 1d_nwk_N_check layer using a different colour to those pits that are snapped directly to a channel end.

This command may be used several times in the .ecf file with the most recent occurrence applying at the time a pit is processed. A value of 0.0 (the default), disables the pit search feature.

This command was introduced for Build 2010-10-AB.

Read GIS Network == <gis_layer>

(1D & 2D/1D. Mandatory)

Reads node and channel locations and attributes from a GIS 1d_nwk layer as described in Section 4.5. Any number of 1d_nwk layers may be read by repeating this command. If accessing external cross-section databases such as MIKE 11 .txt file, the [XS Database](#) command must be specified before this command to set the active cross-section database.

Read GIS Table Links == <gis_layer>

(1D & 2D/1D. Optional)

Reads links to tabular input of cross-section profiles, cross-section hydraulic parameters, nodal surface areas and bridge loss coefficients. The first attribute is the filename (can include a file path) of the .csv or similar file containing the table. This attribute can, for example in MapInfo, be setup as a hotlink allowing the file to be opened in a spreadsheet via the GIS.

See Section 4.6.3 for details.

Read Materials File == <file>

(1D Only. Optional)

Reads a text file containing Manning's n values for different materials. Same format as that used for 2D domains – see [Read Materials File](#).

Relative Resistance == [{RELATIVE} | MATERIAL]

(1D & 2D/1D. Optional)

REDUNDANT as of Build 2003-03-AA and will cause an unrecognisable command error as of Build 2003-07-AA.

Prior to Build 2003-03-AA, sets the default for treating the optional third column of 1d_tab XZ data. The default is to use relative resistance factor. If set to MATERIAL, any values in the third column are treated as a material value, which must occur in the 2D materials file (see [Read Materials File](#)). This setting can be overruled for individual cross-sections using the “R” and “M” flags (see Table 4.17). Refer to discussion in Table 4.17 for more information.

S Channel Approach

== [PRE 2004-06-AA | METHOD A | {METHOD B}]

(1D & 2D/1D. Optional)

PRE 2004-06-AA provides for backward compatibility of S channel types for old models.

METHOD A improved the S channel algorithm after Build 2004-06-AA. The new approach utilises the approach used by G channels for handling situations when the downstream end of a channel is dry or maybe free-overfalling.

METHOD B was adopted as the default for the 2010-10 release after rigorous testing on steep, fast flowing channels showed improved performance over METHOD A. METHOD B only applies the G channel approach in adverse flow conditions (ie. when the water surface gradient is of opposite slope to the channel bed slope), whereas METHOD A tests for and may apply the G channel algorithm on any S channel, and was found in extreme situations to cause some minor choking of the flow down the channel.

Storage Above Structure Obvert [| {(%) }]

== [CHANNEL WIDTH | <value> | {5}]

(1D & 2D/1D. Optional)

Defines how the surface area is to be contributed to the NA table above the obvert of B, C and R channels. As of Build 2006-03-AB, the default is to apply 5% of the maximum surface area. Prior to this build, the default was CHANNEL WIDTH, which uses the top width of B and R channels and the diameter for C channels (see Section 4.6.2.3). Older models that used CHANNEL WIDTH (the default prior to 2006-03-AB) and require it for stability, will most likely need to adopt a smaller 1D [Timestep](#). Alternatively, use of CHANNEL WIDTH is acceptable provided the additional storage that it adds to the model is relatively minor, or it can be demonstrated to not significantly influence results.

If a value is specified, the channels width by half the channel length is applied (provided the UCS attribute is left blank or set to “Y” (yes) or “T” (true) between the invert and obvert, with <value> applied above the obvert. If the (%) option is specified (the default as of Build 2006-03-AB), the

value applied above the obvert is the percentage of the structure's maximum surface area. The default setting is to use 5% of the structure's maximum surface area above the obvert. If the (%) option is not specified, the value is in m² and is applied as a constant above the obvert.

For C channels, the correct flow width in the section is applied (rather than the diameter), and for C and R channels, the No_of_Culverts attribute in the 1d_nwk layer is also used. Use this option where the storage contributed by B, C and/or R channels is significant (eg. pipe model). Note, the reason a storage value of zero is not automatically used above the obvert is that a node cannot have zero storage. A value of zero can be used provided storages at the nodes is contributed by other channels, or a pit storage is applied or commands such as [Minimum NA](#) are used. If the only channels connected to a node are B, C and R channels, the NA table is extended vertically by 5m above the highest obvert. Should water levels exceed this height, use [Depth Limit Factor](#) to extend the table further.

Taper Closed NA Table == [ON | {OFF}]

(1D & 2D/1D. Optional)

Reduces the second last surface area value gradually over 3 additional levels for nodes connected to only closed channels such as bridges and culverts. Also, for B and R channels, starts to reduce storage a 20% of the structures total depth below the obvert, to prevent a sudden change in surface area. This command is still to be further tested, but may offer additional stability in urban models with many closed structures.

Trim XZ Profiles == [ON | {OFF}]

(1D & 2D/1D. Optional)

Trims the XZ profile extracted from ISIS .dat files so that the treatment at the ends of the cross-section profile is similar to that used by ISIS. If set to OFF the whole XZ profile is stored with the sections of the profile before and after the left and right markers disabled. However, the active end of the cross-section profile will extend to midway between the first/last disabled point and the last/first active point at either end of the profile. If set to ON, the points before and after the left and right markers are not stored, and the cross-section extent is not extended to midway to the first/last points nearest the left and right markers.

To have similar compatibility with ISIS, this command should be set to ON.

XS Database == <file>

(1D & 2D/1D. Optional)

Sets the active cross-section database as <file>. The extension of the file determines its format as follows:

- .txt indicates a MIKE 11 .txt processed data import/export file. The file must contain processed cross-section data; any raw data is ignored.
- .dat indicates an ISIS data file containing XZ cross-section profiles – also see [Trim XZ Profiles](#).

- .pro indicates an ISIS processed cross-section data file.
- other file formats including a generic .csv format are planned to be incorporated.

The assignment of cross-sections is carried out using the Branch, Topo_ID and XSect_ID_or_Chainage attributes as discussed in Table 4.14.

This command must be specified before a [Read GIS Network](#) command. The command maybe used at any point to change the active cross-section database.

```
Zero Culvert Coefficients ==  
[ DEFAULT | {OFF} | v1, v2, v3, v4, v5 ]
```

(1D & 2D/1D. Optional)

Sets the default culvert coefficients if they are set to zero (0) in the 1d_nwk layer(s). Incorporated into Build 2008-08-AA. Note, if this command has not been specified a coefficient with a value of zero (the default in MapInfo) is interpreted as having a value of zero and is treated as such.

The five values are as per the following 1d_nwk attributes (see end of Table 4.14).

- v1 Circular culvert inlet controlled contraction coefficient (Height_Cont attribute).
- v2 Rectangular and irregular culvert height contraction coefficient (Height_Cont attribute).
- v3 Rectangular and irregular culvert width contraction coefficient (Width_Cont attribute).
- v4 Culvert inlet loss coefficient (Entry_Loss attribute).
- v5 Culvert outlet loss coefficient (Exit_Loss attribute).

For example, Zero Culvert Coefficients == 1.0, 0.6, 0.9, 0.5, 1.0 sets any zero (0) coefficient values in 1d_nwk layers to the corresponding value from this command. Any non-zero values remain unchanged.

DEFAULT sets zero coefficients to the default values of 1.0, 0.6, 0.9, 0.5 and 1.0. OFF doesn't apply this feature.

B.6 Accessing Fixed Field Data Commands (.ecf)

[BG CS NA VG].... ! Fixed Field Flags	B-24
BG Data == <file>	B-24
CS Data == <file>	B-24
NA Data == <file>	B-25
VG Data == <file>	B-25

[BG | CS | NA | VG].... ! Fixed Field Flags
(1D & 2D/1D. Optional)

Tables in ESTRY's fixed field format (ie. BG, CS, NA and VG tables – see manuals prior to 2007 available from www.tuflow.com) maybe specified at any point in the .ecf file or in another file and accessed using the commands below.

BG Data == <file>
(1D & 2D/1D. Optional)

Reads bridge structure properties tables in ESTRY's fixed field format (ie. BG tables – see manuals prior to 2007 available from www.tuflow.com). The bridge channel must exist in a 1d_nwk layer. The command can be used any number of times to access more than one file, and the file may contain other information besides BG tables.

CS Data == <file>
(1D & 2D/1D. Optional)

Reads channel cross-section properties tables in ESTRY's fixed field format (ie. CS tables – see manuals prior to 2007 available from www.tuflow.com). The channel must exist in a 1d_nwk layer, otherwise an error message occurs. The command can be used any number of times to access more than one file, and the file may contain other information besides CS tables. If channel cross-section properties have been previously specified for a channel (for example, from an external source or in a previous CS table), the last table read for that channel prevails.

NA Data == <file>

(1D & 2D/1D. Optional)

Reads node storage properties tables in ESTRY's fixed field format (ie. NA tables – see manuals prior to 2007 available from www.tuflow.com). The node must exist in a 1d_nwk layer, otherwise an error message occurs. The command can be used any number of times to access more than one file, and the file may contain other information besides NA tables. If node storage properties have been previously specified for a node (for example, using the UCS attribute or from a previous NA table), the last table read for that node prevails.

VG Data == <file>

(1D & 2D/1D. Optional)

Reads variable geometry channel properties tables in ESTRY's fixed field format (ie. VG tables – see manuals prior to 2007 available from www.tuflow.com). The variable geometry channel must exist in a 1d_nwk layer. The command can be used any number of times to access more than one file, and the file may contain other information besides VG tables.

B.7 Initial Water Level (IWL) Commands (.ecf)

Read GIS IWL == <gis_layer>	B-26

Set IWL == <IWL>	B-26

Read GIS IWL == <gis_layer>

(1D & 2D/1D. Optional)

Reads initial water level elevations at nodes from a 1d_iwl GIS layer. The 1d_iwl layer contains points snapped to nodes in the 1d_nwk layer(s). The first attribute of the layer must be the initial water level as a number (float or decimal). The layer can define any number of the nodes (it does not need to define all the nodes). The command can be used any number of times to access more than one 1d_iwl layer.

Set IWL == <IWL>

(1D & 2D/1D. Optional)

Sets the initial water level at all nodes to <IWL>. Initial water levels different to <IWL>, for example in a lake, can be set using the “Read GIS IWL” command.

B.8 Boundary Condition Commands (.ecf)

BC Database == <.csv_file>	B-27
BC Event Name == <bc_event_name>	B-27
BC Event Text == <bc_event_text>	B-28
EB Data == <file>	B-28
Read GIS BC == <gis_layer>	B-28

BC Database == <.csv_file>

(Mandatory)

Sets the active BC Database file as described in Sections 4.10.1 and 4.10.2. The file is usually created using spreadsheet software such as Microsoft Excel.

If the BC Database is specified in the TUFLOW .tcf file, it is set as the active database for both 2D and 1D models. However, the active database can be changed at any stage in the .tbc and .ecf files by repeating the command with the new database set as the <.csv_file>.

A BC Database must be specified before any of the other BC commands are used.

BC Event Name == <bc_event_name>

(Optional)

Sets the active BC name to be substituted where <bc_event_text> (see [BC Event Text](#)) occurs in the BC Database. See Section 4.10.3 for a description of how the BC event commands operate.

This command is normally specified in the .tcf file, and only used in the .tbc file if the event boundaries vary by event within the model. For example, it may be set to “Q100” to read in the 100 year catchment inflows, then set as “H010” to read in the 10 year ocean levels for the downstream boundary. Note that, in this case, the locations of the catchment inflows and downstream boundaries would have to be placed in two separate GIS layers, with each layer read using [Read GIS BC](#) after the relevant [BC Event Name](#) command as shown below:

```
BC Event Name == H010
Read GIS BC == mi\1d_bc_head_boundaries.mif
BC Event Name == Q100
Read GIS BC == mi\1d_bc_flow_boundaries.mif
```

BC Event Text == <bc_event_text>

(Optional)

Sets the text in the BC Database that is to be substituted by the [BC Event Name](#) command value. See Section 4.10.3 for a description of how the BC event commands operate.

For 2D/1D models this command only needs to be specified in the .tcf file. It would be only used in the .ecf file for 1D only models or if for some reason the <bc_event_text> value needs to change prior to reading the 1D BCs. Also see [BC Event Text](#) for the .tcf file if the model is 2D/1D.

The <bc_event_text> value can be changed at any stage by repeating this command in the .ecf file, although it is strongly recommended that the <bc_event_text> value is standardised across all models and the command is specified only once.

EB Data == <file>

(1D & 2D/1D. Optional)

Read boundary condition tables from a file in ESTRY's fixed field format (see manuals prior to 2007 available from www.tuflow.com). The command can be used any number of times to access more than one file, and the file may contain other information besides boundary condition data tables.

Read GIS BC == <gis_layer>

(Mandatory if not using fixed field text entry)

Reads the location and attributes of 1D model boundary conditions as described in Section 4.10.

B.9 Manhole Commands (.ecf)

Manhole Default Loss Approach == [NONE {ENGELHUND} FIXED]	B-29
Manhole Default C Exit Coefficient == [<K _{CE} > {0.25}]	B-29
Manhole Default R Exit Coefficient == [<K _{RE} > {0.5}]	B-29
Manhole Default Side Clearance == [<value_m> {0.3}]	B-30
Manhole Default Type == [C J R CR {CJR}]	B-30
Manhole K Maximum Bend/Drop == [<max_K_bd {4.0}]	B-31
Manhole Minimum Dimension == [<min_width_m {1.05}]	B-31
Manholes at All Culvert Junctions == [{ON} OFF]	B-31
Read GIS Manhole == <gis_layer>	B-31

Manhole Default Loss Approach

== [NONE | {ENGELHUND} | FIXED]

(Optional)

Sets the default approach to be used at automatically generated manholes, and manholes where no loss approach is specified. The default is ENGELHUND (see Section 4.5.2 for a description on the different approaches).

Manhole Default C Exit Coefficient == [<K_{CE}> | {0.25}]

(Optional)

For C (circular) manholes, sets the default K coefficient for flow out of the manhole and into the out flowing culvert(s). The default value is 0.25.

Manhole Default R Exit Coefficient == [<K_{RE}> | {0.5}]

(Optional)

For R (rectangular) manholes, sets the default K coefficient for flow out of the manhole and into the out flowing culvert(s). The default value is 0.5.

Manhole Default Side Clearance == [<value_m> | {0.3}]

(Optional)

For C (circular) manholes, sets the default side clearance in metres from the side of the largest culvert to the side of the manhole chamber (ie. if the diameter of the chamber is not specified, the diameter is set to the width/diameter of the largest culvert plus twice the side clearance).

For R (rectangular) manholes, sets the default side clearance from the side of the culvert(s) to the side of the manhole chamber. If the width of the chamber is not specified, the width is set to the greatest incoming or outgoing width, where the width is calculated as the sum of the incoming/outgoing culvert widths/diameters, plus twice the side clearance for the sides, plus twice the side clearance for the space between each incoming/outgoing culvert if more than one incoming or outgoing culvert exists.

The default side clearance value can be overridden by specifying a negative width for a manhole in a 1d_mh layer. The side clearance applied will be the absolute value of the 1d_mh Width attribute, and the above approach will be used to determine the manhole width/diameter.

Manhole Default Type == [C | J | R | CR | {CJR}]

(Optional)

Sets the default type of manhole to be used at automatically generated manholes, and manholes where no type approach is specified.

- C is for circular manholes;
- R for rectangular manholes;
- J for junctions (ie. no chamber); and
- CR and CJR are for a combination of C, R and J depending on the size/configuration of the connecting culverts (see Section 4.5.2 for further details). The default is CJR.

CR assigns an R type if one or more of the below are true, otherwise a C type is assigned.

- The number of barrels of any culvert is greater than one.
- There are any parallel culverts (ie. two or more culverts with the same nodes and the culverts are digitised in the same direction).
- The diameter of at least one circular culvert (C channel) exceeds 1.2m.
- The width of at least one rectangular culvert (R channel) exceeds 0.45m.

CJR uses the same logic as described above for CR, however, it will assign a J type instead of a C or R if all of the below are true. The J type in this case is to cover the situation where access is possible via the culverts, rather than via a manhole.

- The number of barrels of all connected culverts must be one, and there are no parallel culverts (ie. two or more culverts with the same nodes and the culverts are digitised in the same direction).
- There is only one inlet and one outlet culvert.

- The diameter of at least one circular culvert exceeds 1.8m, or the width and height of at least one rectangular culvert exceeds 1.2m and 1.8m.

Manhole K Maximum Bend/Drop == [<max_K_bd | {4.0}]

(Optional)

Sets the maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach.

Manhole Minimum Dimension == [<min_width_m | {1.05}]

(Optional)

Sets the minimum diameter for C manholes and minimum width and length for R manholes. The value is usually controlled by the minimum dimensions needed for access to manhole chambers, not by any hydraulic efficiency requirements.

Manholes at All Culvert Junctions == [{ON} | OFF]

(Optional)

If set to ON (the default), manholes are automatically created at all culvert junctions (see Section 4.5.2 for more information).

Read GIS Manhole == <gis_layer>

(1D & 2D/1D. Mandatory)

Reads manhole locations and attributes from a GIS 1d_mh layer as described in Section 4.5.2. Any number of 1d_mh layers may be read by repeating this command. Manholes that occur in the same location will override each other with the last manhole processed prevailing. Manholes processed using this command will overwrite any automatically generated manholes. Automatically generated manholes may be individually disabled by digitising points and specifying “NO” for the Loss_Approach attribute – see Table 4.12.

Appendix C .tgc File Commands

[Allow Dangling Z Lines](#)

[Cell Size](#)

[Create TIN Zpts](#)

[Default Land Z](#)

[External Bndy](#)

[Grid Size \(N,M\)](#)

[Grid Size \(X,Y\)](#)

[Interpolate ZC](#)

[Interpolate ZHC](#)

[Interpolate ZUV](#)

[Interpolate ZUVC](#)

[Interpolate ZUVH](#)

[Orientation](#)

[Orientation Angle](#)

[Origin](#)

[Pause When Polyline Does Not Find Zpt](#)

[Read File](#)

[Read GIS](#)

[Read GIS Code](#)

[Read GIS FC Shape](#)

[Read GIS Layered FC Shape](#)

[Read GIS Location](#)

[Read GIS \[Mat | IWL | CnM | Fric | WrF |
FLC \]](#)

[Read GIS Variable Z Shape](#)

[Read GIS Z HX Line](#)

[Read GIS Z Line](#)

[Read GIS Z Shape](#)

[Read GIS Z Shape Route](#)

[Read GIS Zpts](#)

[Read RowCol \[Code | Mat | IWL | CnM | Fric
| WrF \]](#)

[Read RowCol Grid](#)

[Read RowCol Zpts](#)

[Read TGF](#)

[Read TIN Zpts](#)

[Set \[Code | Mat | IWL | CnM | Fric | WrF \]](#)

[Set Route Cut Off Type](#)

[Set Route Cut Off Values](#)

[Set Zpt](#)

[Stop](#)

[TIN Angles](#)

[TIN Coincident Point Distance](#)

[Write MI Domain](#)

[Write MI Grid](#)

[Write MI Zpts](#)

[ZC = MIN\(ZU,ZV\)](#)

[Zero Z Point](#)

C.1 Grid Size, Location and Orientation Commands (.tgc)

Cell Size == <value_in_meters>	C-2
Grid Size (N,M) == <number_rows>, <number_columns>	C-2
Grid Size (X,Y) == <X_length_in_meters>, <Y_length_in_meters>	C-2
Orientation == <XX_in_meters>, <YX_in_meters>	C-2
Orientation Angle == <angle_in_degrees_relative_to_east>	C-3
Origin == <OX_in_meters>, <OY_in_meters>	C-3
Read GIS Location == <gis_layer>	C-3

Cell Size == <value_in_meters>

(Mandatory if not specifying in .tcf file)

Sets the grid's cell size in meters. This overrides any value specified in the .tcf file. The cell size must be specified either using this command or in the .tcf file.

Grid Size (N,M) == <number_rows>, <number_columns>

(One GRID SIZE command is mandatory)

Sets the dimensions of the grid based on the number of rows and columns. Must be integer values.

Grid Size (X,Y) == <X_length_in_meters>, <Y_length_in_meters>

(One GRID SIZE command is mandatory)

Sets the dimensions of the grid using a distance along the grid's X-axis (<X>) and Y-axis (<Y>). The number of columns and rows is rounded to the nearest integer, therefore, <X> and <Y> do not have to be an exact multiple of the cell size.

Orientation == <XX_in_meters>, <YX_in_meters>

(One ORIENTATION command is mandatory if Read GIS Location not used)

Sets the geographical orientation of the grid using another point along the bottom (X-axis) of the grid with coordinates <XX>, <YX>. To set the grid's origin see [Origin](#).

Orientation Angle == <angle_in_degrees_relative_to_east>

(One ORIENTATION command is mandatory if Read GIS Location not used)

Sets the geographical orientation of the grid using an angle. The angle is in degrees relative to east (eg. X-axis directly north would be 90°).

Origin == <OX_in_meters>, <OY_in_meters>

(Mandatory if Read GIS Location not used)

Sets the geographical origin of the grid, the origin being the lower left corner of the lower left cell. <OX> is the X-coordinate in meters and <OY> the Y-coordinate.

Read GIS Location == <gis_layer>

(Mandatory if Origin and Orientation commands not used)

Sets the geographical origin and orientation of the grid based on the first line, polyline or region found in `gis_layer`. The orientation is based on the first point in the line or region being located at the bottom left corner of the grid.

If a line or polyline is used the second point is located anywhere along the bottom of the grid to set the orientation of the grid – it does not determine the length of the grid along the X-axis (use [Grid Size \(N,M\)](#) or [Grid Size \(X,Y\)](#) to set the size of the grid). If using a polyline it must only have two points (vertices) otherwise TUFLOW stops with an error.

If a region is used it must have four sides digitised clockwise. The second vertex is located at or close to the top left corner of the 2D grid. The distance from the first to second vertices determines the length of the grid's Y-axis. The third vertex is not used. The fourth vertex is located at the bottom right corner of the grid. The distance from the first to fourth vertices determines the length of the grid's X-axis. The grid's orientation is determined from the line passing from the first vertex to the fourth vertex. The region approach was incorporated at Build 2002-10-AG.

C.2 Reading External Formats (.tgc)

Read TGA == <.tga_file>

C-4

Read TGF == <.tgf_file>

C-4

Read TGA == <.tga_file>

(Optional)

(.tga files ASCII version of .tgf files (see [Read TGF](#)) and are created using a conversion program written for UNIX platforms. The .tga format is required when transferring .tgf files from UNIX.)

Undocumented feature.

Read TGF == <.tgf_file>

(Optional)

(.tgf files were those used in previous versions of TUFLOW instead of the .tgc file. They are binary formatted.)

Reads a .tgf file. If any of the [Read GIS Location](#), [Grid Size \(N,M\)](#), [Grid Size \(X,Y\)](#), [Origin](#), [Orientation](#), or [Orientation Angle](#) commands above occur after this command, they will overwrite the relevant values from the .tgf file and vice versa. This is useful if a .tgf file is not geographically located or needs to be relocated.

Note: The grid dimensions should not be modified from those in the .tgf file otherwise unexpected results may occur.

Note: This command reads in all the bathymetry and other data in the .tgf file. If the elevation data are to be used, then the [Interpolate ZUVC](#) with the ALL option must be specified after this command as .tgf files only contain ZH Zpts. This creates the ZU, ZV and ZC elevations based on a linear interpolation from the ZH values. Note this occurs prior to any adjustments by flow constrictions (see [Read GIS FC](#) and Section 4.7.2), whereas if the .tgf file is used directly as the geometry file using [Geometry Control File](#) in the .tcf file, the interpolation of ZU, ZV and ZC values occurs after elevation adjustments by flow constrictions.

Once the .tgf data has been read in, other commands can be used to further modify the data. The [Write MI Grid](#) and [Write MI ZPTS](#) commands can be used to export the grid and elevations into .mif/.mid format if desired.

C.3 Model Grid Commands (.tgc)

External Bndy == <n1>, <m1>, <n2>, <m2>	C-5
Read GIS [MAT IWL CnM FRIC WrF FLC] [{} INVERT] == <mif/mid_file>	C-5
Read GIS Code [{} BC] [{} INVERT] == <mif/mid_file>	C-5
Read RowCol [CODE MAT IWL CnM FRIC WrF FLC] == <mid_file>	C-6
Read RowCol Grid == <mid_file>	C-7
Set [Code Mat IWL CnM Fric WrF] == <value> Set Code [{} ZERO ABOVE ZC] == <ZC>	C-7
Write MI Domain == <gis_layer>	C-8
Write MI Grid == <gis_layer>	C-8

External Bndy == <n1>, <m1>, <n2>, <m2>

(Optional)

Now a largely redundant command as TUFLOW automatically assigns boundary codes to cells based on the GIS boundary condition layer (see Section 4.10.7). The command assigns external boundary codes (code = 2) from cell (n1,m1) to cell (n2,m2). Note n is the nth row and m is the mth column from the lower left hand corner.

Read GIS [MAT | IWL | CnM | FRIC | WrF | FLC] [{} | INVERT] == <mif/mid_file>

Read GIS Code [{} | BC] [{} | INVERT] == <mif/mid_file>

(Optional)

Reads the code, material, IWL, CnM, FRIC, WrF or FLC values from a GIS layer exported as .mif/.mid files. Except for the “Read GIS Code BC” combination, the first attribute (column) in the file must be the value of the cell code (CODE), material ID (MAT), initial water level (IWL), bed resistance value (CnM), ripple height (FRIC), weir factor (WrF) or form loss coefficient (FLC) values attached to the GIS objects. Any other attribute columns are ignored. CnM is a Chezy C, Manning’s n or Manning’s M value as set by [Bed Resistance Values](#).

For “Read GIS Code BC”, code values are extracted from objects in a 2d_bc layer that have a Type attribute of “CD”. The code value is taken from the 2d_bc f attribute. See Table 4.32 in Section 4.10.7.

As of Build 2007-07-BD, if INVERT is specified (eg. Read GIS Code Invert or Read GIS Code BC Invert), the active/inactive status of any Code polygons are reversed, ie. a Code of 1 becomes 0, and a Code of -1 or 0 becomes 1. This means that the same layer can be used by both .tgc and .tbc files when linking 2D domains.

Any cell falling within/on an object is assigned the object’s value. The object may be a region (polygon), line or point. For CODE, MAT, IWL, CnM and FRIC the cell centre must fall within the region, or if the object is a point, the point must fall within the cell. For WrF and FLC the mid-sides of the cell are used rather than the cell centre.

If [Bed Resistance Cell Sides](#) == INTERROGATE (the default as of Build 2007-07-AA), the material values are also directly sampled at the cell mid-sides. This gives a higher resolution definition of the materials data, thereby giving improved flow patterns where Manning’s n values vary significantly such as in an urban environment. See Section 4.4.12.

Note: IWLs can also be set in the .tcf file (see [Read GIS IWL](#)). This is preferable if the initial water levels vary from simulation to simulation as it removes the necessity to create a new .tgc file each time the initial water levels change. Any IWL values set in the .tcf file override those specified in the .tgc file for the same cells.

WrF values can vary throughout the model. A value of zero (0) turns the weir function OFF at BOTH the u and v points of a cell (ie. right and upper sides). The WrF values are multiplied by the [Global Weir Factor](#) specified in the .tcf file.

This command is similar to the [Read RowCol](#) command, but is preferred as the GIS layer is read directly, offering better efficiency and quality control.

Note, as of Build 2007-07-AA, that any [Read GIS Fric](#) commands in old models used to set the material values must now be renamed [Read GIS Mat](#).

```
Read RowCol [ CODE | MAT | IWL | CnM | FRIC | WrF | FLC ] ==  
<mid_file>  
(Optional)
```

Reads the code, material, IWL, CnM, fric, WrF or FLC values from a .mid or similarly formatted (comma delimited) file. The first three columns in the file must be "n, m, <value>", where n and m are the 2D grid row, column and <value> is the value of the material ID (MAT), initial water level (IWL), bed resistance value (CnM), ripple height (FRIC), weir factor (WrF) or form loss coefficient (FLC). CnM is a Chezy C, Manning’s n or Manning’s M value as set by [Bed Resistance Values](#). Any columns after the third are ignored.

Note: An IWL .mid file can also be read from the .tcf file (see [Read RowCol IWL](#)). This is preferable if the initial water levels vary from simulation to simulation as it removes the necessity to create a new

.tgc file each time the initial water levels change. Any IWL values set in the .tcf file override those specified in the .tgc file for the same cells.

WrF values can vary throughout the model. A value of zero (0) turns the weir function OFF at BOTH the u and v points of a cell (ie. right and upper sides). The WrF values are multiplied by the general weir factor specified in the .tcf file.

Note, as of Build 2007-07-AA, that any [Read RowCol Fric](#) commands in old models used to set the material values must now be renamed [Read RowCol Mat](#). Also, [Read RowCol Mat](#) is incompatible with the new default as of Build 2007-07-AA, of [Bed Resistance Cell Sides](#) == INTERROGATE (for backward compatibility use [Bed Resistance Cell Sides](#) == AVERAGE M).

Read RowCol Grid == <mid_file>

(Optional)

Reads in a text file, which must be of the same format as the first four data columns of the .mid file produced by the [Write MI Grid](#) or [Write Check Files](#) command. The file could also be created by a text editor or in Excel .csv format.

Only the first four data items on each row are read in free-field comma-delimited format. These four fields must be n, m, Code, Material as defined in Section 4.4.1. Note: the Grid_Ref and ZC attributes created by the [Write MI Grid](#) command, and any other additional columns are ignored.

Note: This command is now redundant with the use of the [Read GIS](#) commands.

Set [Code | Mat | IWL | CnM | Fric | WrF] == <value>

Set Code [{} | ZERO ABOVE ZC] == <ZC>

(Optional)

Sets the cell code, material, initial water level, bed resistance value, ripple height or weir factor value over the entire grid. Used for initialising grid values.

The ZERO ABOVE ZC option for Set Code applies a Code value of zero (0) to all cells that have a ZC value greater than the <ZC> value specified for this command. For example, “Set Code Zero Above ZC == 50.” applies a Code 0 (inactive) to all 2D cells in the domain that have a ZC value above 50 (this includes any unassigned ZC values as the default value is 99999. Also, if any ZU, ZV or ZH points on that cell have not yet been assigned an elevation by any previous Zpt commands, they are set to the ZC value of a cell they are attached to, thereby removing any 99999 elevations around the edge of the model. As this option is dependent on the Zpt values, the location of this command relative to the Zpt commands within the .tgc file is important. It should occur after the Zpts have been assigned elevations. You may also not want to use the [Set Zpt](#) command so that the automatic trimming of ZU, ZV and ZH points occurs. This command is very useful for setting the active area for direct rainfall models where the DEM has been trimmed to the catchment boundary.

Material value is a Bed Material ID used for defining a Manning’s n (see Section A.5 and [Read Materials File](#)).

Note there is an equivalent IWL command in the .tcf file that is often preferred as the IWL may vary from simulation to simulation. Any IWL values set in the .tcf file override those specified in the .tgc file for the same cells.

CnM is a Chezy C, Manning's n or Manning's M value as set by [Bed Resistance Values](#).

WrF is the Weir calibration factor (this value is multiplied by the general weir factor specified in the .tcf file). A value of zero disables the weir application unless overridden by subsequent commands that assign WrF values.

Write MI Domain == <gis_layer>

(Optional)

Creates 2d_dom.mif and .mid files containing a rectangular region representing the extent of the 2D domain. Useful for cross-checking the 2D domain's extent in the GIS rather than generating a large 2d_grd file using [Write MI Grid](#). Incorporated in Build 2005-05-AN.

A 2d_dom layer is also created using [Write Check Files](#), however, it will contain a rectangular region for all 2D domains.

Write MI Grid == <gis_layer>

(Optional)

Creates .mif and .mid files representing the 2D domain's grid based on the dimensions, origin and orientation. The grid is a mesh of square polygons.

All information relating to grid cells as defined by any previous commands up until that point within the .tgc file is included.

Tip: Use this command to check that the grid's data (code, material, etc.) is setup correctly by writing to temporary mif/mid files, and importing and viewing in the GIS at different stages in the .tgc file (this command can be used any number of times within a .tgc file – remember to specify a different .mif filename each time though!).

A 2d_grd layer is also created using [Write Check Files](#), however, it will contain the active cells of all 2D domains.

C.4 Model Bathymetry / Elevation Commands (.tgc)

Allow Dangling Z Lines == [ON {OFF}]	C-10
Create TIN Zpts [{} WRITE TIN] [XF ON XF OFF] == <mif/mid_file>	C-10
Default Land Z == <elevation_in_meters>	C-10
Interpolate ZC [{} ALL] [{} LOWER]	C-11
Interpolate ZHC [{} ALL]	C-11
Interpolate ZUV [{} ALL] [{} MAX]	C-11
Interpolate ZUVC [{} ALL] [{} AFTER FC]	C-12
Interpolate ZUVH [{} ALL] [{} MAX]	C-12
Pause When Polyline Does Not Find Zpt == [ON {OFF}]	C-12
TIN Angles == <point_angle>, <edge_angle> {55, 30}	C-13
TIN Coincident Point Distance == [<dist_in_m> {0.001}]	C-13
Read GIS FC Shape == <mif/mid_file>	C-13
Read GIS Layered FC Shape == <mif/mid_file>	C-13
Read GIS Z HX Line [{} RIDGE or MAX or RAISE] == <mif/mid_file>	C-13
Read GIS Variable Z Shape [XF ON XF OFF] == <mif/mid_file>	C-14
Read GIS Z Line [{} RIDGE or MAX GULLY or MIN HX] [{} CC] [{} THICK] [{} ADD] == <mif/mid_file>	C-14
Read GIS Z Shape [{} RIDGE or MAX or RAISE GULLY or MIN or LOWER] [XF ON XF OFF] == <mif/mid_file>	C-16
Read GIS Z Shape Route [{} RIDGE or MAX or RAISE GULLY or MIN or LOWER] == <mif/mid_file>	C-17
Read GIS Zpts [{} ADD MAX MIN] == <mif/mid_file>	C-17
Read RowCol Zpts [{} ADD MAX MIN] [XF ON XF OFF] == <zpt_file>	C-18
Read TIN Zpts [{} ADD MAX MIN] == <tin_file>	C-18

Set Zpt == <elevation_in_meters>	C-18
Write MI Zpts == <mif/mid_file>	C-19
ZC == MIN(ZU,ZV)	C-19
Zero Z Point == [{ERROR} WARNING]	C-19

Allow Dangling Z Lines == [ON | {OFF}]

(Optional)

If a breakline using the [Read GIS Z Line](#) command does not find a snapped point at the end (ie. the end is dangling), but the line has at least one snapped point elsewhere along the line, this command if set to ON assigns the elevation of the nearest snapped point to the dangling end. This command may be used several times through a .tgc file to change the setting before different [Read GIS Z Line](#) commands. Elevations applied to dangling ends are displayed to the screen and log file. The default (OFF option) is to not allow dangling breaklines, in which case, a paused warning is displayed to the screen and the elevation adopted is that given to the line (ie. all snapped points are ignored). This command was introduced at Build 2002-08-AC.

**Create TIN Zpts [{} | WRITE TIN] [XF ON | XF OFF]
== <mif/mid_file>**

(Optional)

Creates a TIN for each polygon in the GIS layer, and assigns elevations to any Zpts falling within the TIN. Any points and lines falling within the polygon are used for creating the TIN surface. Lines can have points snapped to them to create a 3D breakline effect through the TIN. For more information and examples, see Section 4.4.8.

If WRITE TIN is specified, a SMS .tin file is written to the same location as the GIS layer. The TIN can be viewed/checked in SMS, edited and modified is needed. [Read TIN Zpts](#) can be used to assign Zpt values resulting from the modified TIN. If [Write Check Files](#) is active and WRITE TIN is specified, the triangles are written to the 2d_sh_obj_check.mif layer.

XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <zpt_file>. The global default for using XF files is ON and can be changed using [XF Files](#).

Default Land Z == <elevation_in_meters>

(Optional)

Sets any previously unspecified ZH, ZU, ZV or ZC Zpts to the value for land cells only. Is useful where all the land cells and their Zpts have been removed from the GIS layers to keep file sizes to a minimum.

Note that unspecified cells are automatically set to land and that Zpts in these cells should be assigned a "flood-free" Z-value.

Interpolate ZC [{} | ALL] [{} | LOWER]

(Optional)

Interpolates ZC elevations where they have not been specified. If ALL occurs at the end of the command, then all ZC elevations are interpolated.

Note: If a value already exists (for example, from previous [Read GIS Zpts](#) commands) it will not be affected unless the ALL option is specified.

The LOWER option sets the ZC value to the average of the two lowest of the four ZU and ZV points. This is useful in models with highly variable or bumpy topography (eg. of urban areas with buildings incorporated), as it will open up and smooth some flowpaths that were blocked by a high ZC value. The default is to set the ZC value to the average of the four ZU and ZV values.

Also see [Interpolate ZHC](#), [Interpolate ZUV](#), [Interpolate ZUVC](#), [Interpolate ZUVH](#)

This command was incorporated into Build 2004-03-AA.

Interpolate ZHC [{} | ALL]

(Optional)

Interpolates ZH and ZC elevations where they have not been specified. If ALL occurs at the end of the command, then all ZH and ZC elevations are interpolated. The values are a linear interpolation of the ZU and ZV values.

This command can provide some “smoothing” of the cell centre and corner elevations that may be desirable in a model, particularly if the DTM data is “bumpy”, such as occurs from airborne laser surveys.

Also see [Interpolate ZC](#), [Interpolate ZUV](#), [Interpolate ZUVC](#), [Interpolate ZUVH](#)

Interpolate ZUV [{} | ALL] [{} | MAX]

(Optional)

Interpolates ZU and ZV elevations where they have not been specified. If ALL occurs at the end of the command, then all ZU and ZV elevations are interpolated. The ZU and ZV values are a linear interpolation of the neighbouring ZC values.

This command can provide some “smoothing” of the cell side elevations that may be desirable in a model, particularly if the DTM data is “bumpy”, such as occurs from airborne laser surveys.

The MAX option sets the ZU and ZV values to the higher of the neighbouring ZC values (rather than a linear interpolation). This option is experimental and is not recommended for practical use.

This command was incorporated into Build 2005-11-AB.

Also see [Interpolate ZC](#), [Interpolate ZHC](#), [Interpolate ZUVC](#), [Interpolate ZUVH](#)

Interpolate ZUVC [{} | ALL] [{} | AFTER FC]

(Optional)

Interpolates ZC, ZU and ZV elevations where they have not been specified. If ALL occurs at the end of the command, then all ZC, ZU and ZV elevations are interpolated. The ZU and ZV values are a linear interpolation of the neighbouring ZH values, while the ZC value is the average of the four surrounding ZH values (this was the standard approach of earlier versions of TUFLOW where only the Zpts at the H location were specified).

The AFTER FC option interpolates ZC, ZU and ZV points after flow constrictions (FC) have been set. This allows old TUFLOW models that used the .tgf file to be read via a .tgc file and retain backward compatibility. The AFTER FC option should be used in conjunction with the ALL option to have full backward compatibility.

Also see [Interpolate ZC](#), [Interpolate ZHC](#), [Interpolate ZUV](#), [Interpolate ZUVH](#)

Interpolate ZUVH [{} | ALL] [{} | MAX]

(Optional)

Interpolates ZH, ZU and ZV elevations where they have not been specified. If ALL occurs at the end of the command, then all ZH, ZU and ZV elevations are interpolated. The ZU and ZV values are a linear interpolation of the neighbouring ZC values, while the ZH value is the average of the four surrounding ZC values. This option is particularly useful for converting MIKE 21 models where the elevations are only specified at the cell centres.

The MAX option sets the ZU and ZV values to the higher of the neighbouring ZC values (rather than a linear interpolation). This option is experimental and is not recommended for practical use.

This command was incorporated into Build 2005-05-AA.

Also see [Interpolate ZC](#), [Interpolate ZHC](#), [Interpolate ZUV](#), [Interpolate ZUVC](#)

Pause When Polyline Does Not Find Zpt == [ON | {OFF}]

(Optional)

If a breakline using the [Read GIS Z Line](#) command does not find a Zpt, TUFLOW by default (the ON option), pauses with a warning message and waits for a return key to be entered. To switch the pause off, use the OFF option. This command may be used several times through a file to change the setting before different [Read GIS Z Line](#) commands. Warnings are displayed to the screen and to the log file irrespective of the setting above.

This command is useful where there are very short breaklines (for example, survey lines imported from another software which has lost the connectivity between line segments), which do not affect any Zpts.

Prior to Build 2004-03-AB the default was ON.

TIN Angles == <point_angle>, <edge_angle> | {55, 30}

(Optional)

Introduced for the 2010-10 release, provides the user with the ability to vary the formation of triangles in TINs created from Read GIS Shape polygons.

<point_angle> controls the minimum angle from an internal point to two vertices on the triangulation perimeter to be used. The smaller the angle the greater the priority given to triangulating to an internal point. For builds prior to the 2010-10 release, <point_angle> was set to 95°.

<edge_angle> controls the formation of triangles from vertices along internal boundary of the TIN as it is created. The greater the angle the greater the priority given to triangulating using boundary vertices only. For builds prior to the 2010-10 release, <edge_angle> was set to 0°, which in rare occasions would cause problems with a triangulation failing, especially where there are automatically inserted vertices along a straight line segment. For this reason, [Defaults](#) == PRE 2010-10 does not set <edge_angle> to 0°. There have also been some other minor enhancements to prevent triangulations failing, making full backward compatibility not recommended.

The command maybe repeated within the .tgc file to change the angles for different commands.

TIN Coincident Point Distance == [<dist_in_m> | {0.001}]

(Optional)

New command for Build 2008-08-AA that changes the distance used for removing coincident points prior to creating a TIN. Can be used several times within the .tgc file to apply different distances for different layers/commands. Applies to any polygon shape or command that generates a TIN.

Read GIS FC Shape == <mif/mid_file>

(Optional)

New command for Build 2008-08-AA that offers much improved application of flow constriction (FC) cells. See Section 4.7.2.1.

Read GIS Layered FC Shape == <mif/mid_file>

(Optional)

New command and feature for Build 2008-08-AA that offers the ability to vary flow constriction (FC) parameters with height so as to model obstructions such as bridges and pipelines in 2D at all flow heights. See Section 4.7.2.3 for details.

**Read GIS Z HX Line [{} | RIDGE or MAX or RAISE]
== <mif/mid_file>**

(Optional)

New command for Build 2008-08-AA that uses HX lines and ZP points from a 2d_bc (or 2d_hx) layer to set the cell elevations along HX lines. There must be at least one “ZP” (Type) point snapped to

each HX line in the 2d_bc layer (alternatively a separate points layer can be used - see Section 4.4.11). The 2d_bc “f” attribute is used to set the elevation and the “d” attribute is used to adjust the elevation (ie. if d = 0, the elevation remains unchanged – useful if you wish to raise the line by, say, 0.2m) - see Table 4.33.

If no “ZP” points are snapped to the HX line then no Zpts are adjusted along that line and a CHECK is issued stating this.

When adjusting the cell heights, the THICK approach described for [Read GIS Z Line](#) is used (ie. the whole cell is modified).

If RIDGE or MAX or RAISE (they all perform the same function!) are specified this becomes the default setting for the treatment of all HX lines. Alternatively, the “R” Flag can be used to force the ridge option for that line. If RIDGE or MAX or RAISE is specified the “R” flag on a HX line is redundant - see Table 4.33. As of Build 2008-08-AC, an “A” flag adjusts all elevations irrespective of whether RIDGE, MAX or RAISE are used.

Read GIS Variable Z Shape [XF ON | XF OFF] == <mif/mid_file>

(Optional)

New command for Build 2008-08-AA used to define the final 3D shape of the Zpts at the completion of a breach, or other change in topographic shape, during a simulation. Similar to [Read GIS Z Shape](#), but with additional attributes to control the trigger mechanism and time period of the failure. For examples and detailed description of the GIS layer attributes see Section 4.4.10.

Build 2008-08-AC includes a bug fix that the initial water level is now adjusted with the adjustment of the cell elevations whilst the cell is dry (otherwise water is generated if any dry cells start being lowered).

XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <zpt_file>. The global default for using XF files is ON and can be changed using [XF Files](#).

Read GIS Z Line

[{} | RIDGE or MAX | GULLY or MIN | HX]

[{} | CC]

[{} | THICK]

[{} | ADD]

== <mif/mid_file>

(Optional)

See also [Allow Dangling Z Lines](#) and [Pause When Polyline Does Not Find Zpt](#) commands.)

Reads .mif/.mid formatted files containing polylines that are treated as breaklines in the model’s bathymetry. The breakline can vary in height along its length (ie. a 3D breakline).

This is a powerful feature for quickly and easily entering a breakline feature such as a road, railway, levee, creek, drain, etc. It is particularly useful where TUFLOW’s fixed grid discretisation does not

guarantee that the crest along, for example, a road, is picked up from the DTM, or the lowest point along a drain. It saves having to incorporate roads, levees, etc into the DTM.

The modified Zpts, except for the GULLY option, are output to the 2d_zln_zpt_check.mif layer (see Table 7.1) if [Write Check Files](#) has been set.

The approach uses the polylines in the layer to set the nearest Zpt values in the TUFLOW grid to the polyline's height.

A variable height polyline is created in the GIS by snapping the polyline to points in the same layer. The first attribute column must be a number (real or integer) representing the elevation of the points. Other attributes are ignored. If the polyline is not snapped with a point at its beginning and end, the polyline is assumed to be horizontal (the height is taken from the polyline's attribute). Otherwise, the polyline's grade is determined by the height of the points snapped to the polyline nodes. It is not necessary to snap a point at every polyline node – the minimum requirement is a point snapped to each polyline end. Height values for nearby TUFLOW Z-points are interpolated.

The default is to modify a “thin” line following the ZH, ZU and ZV Zpts. If the THICK option occurs, interpolated Z values are applied to whole cells (ie. at the cell centres, all cell sides and cell corners).

If the RIDGE option is specified, the Z values are only modified where the polyline height is higher than the current Z values. This is useful where, for example, a weir occurs in a river and it is easier to just digitise the weir from bank to bank without having to determine where it should exactly end. The keyword MAX can be substituted for RIDGE.

Conversely, the GULLY option adjusts the ZU, ZV and ZC values where the polyline is lower than the current Z value. This option is useful for ensuring low flow paths such as small creeks or drains are modelled without “dams” across their path. **The GULLY option should not be seen as a method to accurately define the shape of a waterway.** The keyword MIN can be substituted for GULLY. **Note: The THICK option is not available with the GULLY option.**

As of Build 2006-06-AA, if [Line Cell Selection](#) is set to Method C (the default), a more advanced approach for the RIDGE option is used to interpolate Zpt values. The approach interpolates from the Zpt to the nearest intersection of the Z line (ie. the perpendicular), or if there is no perpendicular intersection, the nearest vertex on the Z Line. The previous approaches used a more simplistic approach of intersecting the polyline with a line extending perpendicular to the cell side, and ZC and ZH values were an average of the modified ZU and ZV values. The new approach produces “smoother” Zpts, and is not prone to unpredictable final elevations where multiple Z lines cross through a cell. For RIDGE, the highest value of the Z lines that intersect with the “cross-hairs” is chosen, even if there are closer Z lines. If RIDGE (or MAX) is not specified, the value from the closest eligible Z line is used. ADD works for both scenarios.

The GULLY option takes the intersection of the polyline with the cell side to determine elevations and has not changed with Build 2006-06-AA.

If neither the RIDGE or GULLY option is specified, the Z values are adjusted along the entire polyline length, irrespective of whether the height of the line is higher or lower than the current Zpt values.

The RIDGE (not the GULLY) methodology is used in determining which Zpts are selected for modification.

As of Build 2006-06-AA, the HX and CC options are considered redundant other than for backward compatibility. If Method C (the default) is set for both [Boundary Cell Selection](#) and [Line Cell Selection](#), there is no need to specify HX or CC. If CC is specified, TUFLOW stops with an ERROR message. If HX is specified, it has the same effect as specifying the THICK option. Method C uses the cell “cross-hair” approach where a cell, or a cell side (ZU and ZV points), is only selected if the line intersects imaginary “cross-hairs” that extend from cell mid-sides to cell mid-sides.

The HX option was introduced as of Build 2005-05-AN. It uses the same process as used for selecting cells along a HX line, and always uses the THICK approach. It is designed for setting the elevations of cells along HX lines. Note that further improvements to this approach may occur as the interpolation of elevation values when several elevation points fall within one cell is not ideal. The CC, RIDGE or GULLY options are ignored if HX is specified.

The CC option shifts polyline vertices to their nearest cell centres. This is useful when specifying a ridge line along a 2d_bc HX line, as 2d_bc lines are always shifted to the nearest cell centres. The option forces the Z-line and the 2d_bc line to follow the same set of cells. Note, both the Z-line and the HX line must be digitised in the same direction to ensure compatibility. **Note: This paragraph and the CC option are redundant, except for backward compatibility, as of Build 2004-01-AA – see [Oblique Boundary Alignment](#).**

The ADD option adds (use negative values to subtract) the height value along the polyline to the current Zpt values.

This feature is also incorporated into [Read GIS Z Shape](#), [Read GIS Variable Z Shape](#) and other shape commands incorporated into Build 2008-08-AA. These commands offer more flexible application of Z lines in that a layer can contain a mixture of thin and thick lines, a mixture of RIDGE, GULLY and ADD options, and a width in metres can be applied to lines if more than one cell width is needed to be raised/lowered/added. See Sections 4.4.5 and 4.4.9 for further information.

Read GIS Z Shape

```
[ { } | RIDGE or MAX or RAISE | GULLY or MIN or LOWER ]  
[ XF ON | XF OFF ]  
== <mif/mid_file>
```

(Optional)

New command for Build 2008-08-AA that offers powerful options for modifying Zpt elevations using points, lines and polygons to define 3D shapes. For examples and detailed description of the options available see Section 4.4.9. The full functionality (and more) of [Read GIS Z Line](#) is incorporated into this command.

If RIDGE or MAX or RAISE (they all perform the same function!) are specified this becomes the default setting for the treatment of lines unless the Shape_Option attribute for a line overrides this setting. Similarly if GULLY or MIN or LOWER is specified.

The 2009-07 release incorporates an improved buffering of wide Z Lines. 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.

XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <zpt_file>. The global default for using XF files is ON and can be changed using [XF Files](#).

Read GIS Z Shape Route

**[{} | RIDGE or MAX or RAISE | GULLY or MIN or LOWER]
== <mif/mid_file>**

(Optional)

New command for the 2009-07 release that provides output on: the degree of inundation along routes, and helps define evacuation routes, warning times, risks and durations that routes are cut off. It also performs the same adjustment to Zpts as per [Read GIS Z Shape](#).

See Section 4.8.2 for a description of this feature. Also see [Set Route Cut Off Values](#) and [Set Route Cut Off Type](#).

Read GIS Zpts [{} | ADD | MAX | MIN] == <mif/mid_file>

(Optional)

Reads the Zpt values from a GIS layer exported as .mif/.mid files. The first attribute (column) must be the Zpt value attached to the GIS objects. Any other attribute columns are ignored.

Any Zpt (ZC, ZU, ZV and ZH) falling within/on an object is assigned the object's first attribute value. The object may be a region (polygon), line or point.

The ADD option adds the first attribute value of the object to the Zpts. Use a negative value to subtract.

The MAX option will only raise a Zpt from its existing value, while the MIN option will only lower the Zpt value from its existing value.

This command is similar to the [Read RowCol Zpts](#) command, and is preferred where an area of Zpts needs to be modified to the same height (eg. setting a proposed development to a flood free height) or adjusted (using the ADD option) by the same amount (eg. deepening a channel by half a meter). The [Read RowCol Zpts](#) should be used to assign individual Zpt values based on a point inspection of a DTM.

```
Read RowCol Zpts [ {} | ADD | MAX | MIN ] [ XF ON | XF OFF ]  
== <zpt_file>  
(Optional)
```

Reads in Zpt elevation data. The .mid file must be the same format as that produced by the [Write MI Zpts](#) command.

The ADD option adds the Zpt value to the current Zpt value.

The MAX and MIN options only modify the current Zpt value if the value is higher (MAX option) or lower (MIN option) than the existing value.

The GIS layer can be trimmed to contain either only H values or only U and V values to minimise the size of the file. In this case use an [Interpolate](#) command to interpolate other Z values.

XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <zpt_file>. The global default for using XF files is ON and can be changed using [XF Files](#).

```
Read TIN Zpts [ {} | ADD | MAX | MIN ] == <tin_file>  
(Optional)
```

New command for Build 2008-08-AA. Use to read a triangulation file (TIN) to assign elevation values to Zpts. Accepted TINs at present are SMS .tin files and 12D TINs as a .12da file. The type of TIN is determined by the file extension.

ADD adds the TIN value to the Zpt elevations.

MAX only changes the value of a Zpt if the TIN value is greater than the current Zpt value.

MIN only changes the value of a Zpt if the TIN value is less than the current Zpt value.

Also see [Create TIN Zpts](#) to have TUFLOW create and write out a TIN.

```
Set Zpt == <elevation_in_meters>  
(Optional)
```

Sets all ZC, ZU, ZV and ZH Zpts to the value specified.

Write MI Zpts == <mif/mid_file>

(Optional)

Writes .mif and .mid files containing the points where Zpts (model elevation) values are defined.

Use this command in the first instance to generate a GIS Zpts table from which to carry out a point inspection on each of the points using a 3D surface modelling package such as Vertical Mapper.

Export the table back to MIF/MID format and use the "READ GIS ZPTS" command to read in the elevations.

For less topographic detail, remove either H and C Z-points, or U, V and C Z-points and use the [Interpolate](#) commands below to calculate the missing values.

Tip: If you wish to modify a section of the model's original bathymetry then:

- select and save the relevant Zpts as another GIS layer;
- modify the height values;
- export the new layer to MIF/MID format; and
- use the [Read RowCol Zpts](#) command to override the original bathymetry.

Tip: Use this command to check that the model's elevation data is correct. After building the topography use this command to write temporary .mif and .mid files. Import into the GIS and check the elevations are as expected. The layer could also be used to generate a DTM representing exactly how TUFLOW "sees" the data.

ZC == MIN(ZU,ZV)

(Optional)

Sets the ZC Zpt equal to the minimum of the two ZU and two ZV Zpts either side and above and below it.

This essentially allows a grid cell to wet and dry according to when water first enters and last leaves the cell. It may provide enhanced stability in models with severe wetting and drying.

Zero Z Point == [{ERROR} | WARNING]

(Optional)

New command for Build 2008-08-AA. If set to ERROR, causes an ERROR 2049 message if a snapped point to a Z Line, or inside or on the perimeter of a Shape region has a zero value. If set to WARNING, a warning message is issued and the simulation does not stop.

If the ADD option is used no ERRORS or WARNINGS are issued except in the case of points snapped to TIN lines.

Note: If [Defaults](#) == PRE 2008-08 is specified, the default setting is WARNING.

C.5 Other Commands (.tgc)

Read File == <file>	C-20

Stop	C-20

Read File == <file>

(Optional)

Directs input to another file. When finished reading <file>, TUFLOW returns to reading the .tgc file.

This command is particularly useful for projects with a large number of .tgc files. Repetitive commands are grouped and placed in another text file. If one of these commands changes, the command only has to be edited once, rather than in every .tgc file.

For example, as the grid size, location and orientation commands are likely to be the same for all runs, placing these commands in their own text file could be advantageous if ever the grid's size, location and/or orientation changes (ie. only one file would have to be edited).

NOTE: As of Build 2002-03-AA, this command can now be used in redirected file(s) up to a maximum of ten levels.

Stop

(Optional)

Stops TUFLOW (useful while just developing the model grid and Zpts).

Appendix D .tbc File Commands

[BC Database](#)

[BC Event Name](#)

[BC Event Text](#)

[Global Rainfall Area Factor](#)

[Global Rainfall BC](#)

[Global Rainfall Continuing Loss](#)

[Global Rainfall Initial Loss](#)

[Read GIS BC](#)

[Read GIS RF](#)

[Read GIS SA](#)

[Read RowCol RF](#)

[Unused HX and SX Connections](#)

D.1 Boundary Condition Commands (.tbc)

BC Database == <.csv_file>	D-2
BC Event Name == <bc_event_name>	D-3
BC Event Text == <bc_event_text>	D-3
Global Rainfall Area Factor == [{1.0} <area_factor>]	D-3
Global Rainfall BC == <BC_name>	D-3
Global Rainfall Continuing Loss == [{0} <CL_in_mm/h>]	D-4
Global Rainfall Initial Loss == [{0} <IL_in_mm>]	D-4
Read GIS BC == <gis_layer>	D-4
Read GIS RF == <gis_layer>	D-4
Read RowCol RF == <gis_layer>	D-4
Read GIS SA [{} ALL PITS] [{} RF] == <gis_layer> Read GIS SA TRIGGER == <gis_layer>	D-5
Unused HX and SX Connections == [{ERROR} WARNING]	D-5

BC Database == <.csv_file>

(Mandatory)

Sets the active BC Database file as described in Sections 4.10.1 and 4.10.2. The file is usually created using spreadsheet software such as Microsoft Excel.

If the BC Database is specified in the TUFLOW .tcf file, it is set as the active database for both 2D and 1D models. However, the active database can be changed at any stage in the .tbc and .ecf files by repeating the command with the new database set as the <.csv_file>.

A BC Database must be specified before any of the other BC commands are used.

BC Event Name == <bc_event_name>

(Optional)

Sets the active BC name to be substituted where <bc_event_text> (see [BC Event Text](#)) occurs in the BC Database. See Section 4.10.3 for a description of how the BC event commands operate.

This command is normally specified in the .tcf file, and only used in the .tbc file if the event boundaries vary by event within the model. For example, it may be set to “Q100” to read in the 100 year catchment inflows, then set as “H010” to read in the 10 year ocean levels for the downstream boundary. Note that, in this case, the locations of the catchment inflows and downstream boundaries would have to be placed in two separate GIS layers, with each layer read using [Read GIS BC](#) after the relevant [BC Event Name](#) command as shown below:

```
BC Event Name == H010
Read GIS BC == mi\2d_bc_head_boundaries.mif
BC Event Name == Q100
Read GIS BC == mi\2d_bc_flow_boundaries.mif
```

BC Event Text == <bc_event_text>

(Optional)

Sets the text in the BC Database that is to be substituted by the [BC Event Name](#) command value. See Section 4.10.3 for a description of how the BC event commands operate.

This command is normally specified in the .tcf file, and only used in the .tbc file if for some reason the <bc_event_text> value needs to change (this should be very unlikely unless wanting to split the different boundaries into groups). Also see [BC Event Text](#) for the .tcf file.

Global Rainfall Area Factor == [{1.0} | <area_factor>]

(Optional)

Sets the factor applied to the global rainfall after the initial loss and continuing losses have been applied. This is useful if you wish to include catchment area outside the area covered by the active (Code 1) cells.

Global Rainfall BC == <BC_name>

(Optional)

Sets the BC name in the BC database that defines the global rainfall. The rainfall is specified as mm versus time in hours. This is converted to m/s and applied as a source versus time (ST) boundary to all active (wet) cells. Note that if there are no wet cells, there is no rainfall applied. The converted time-series after extraction of any losses (see [Global Rainfall Initial Loss](#) and [Global Rainfall Continuing Loss](#)) and any area factor (see [Global Rainfall Area Factor](#)) is output to the .tlf file (Build 2002-01-AC or later) for checking.

Whilst global rainfall is applied only to wet cells, it is factored up to include any water (Code 1) cells that are dry. The catchment area for global rainfall is therefore all cells with a Code of one (1) (see

Section 4.4.1). Use the [Global Rainfall Area Factor](#) command to increase the catchment area to include areas not covered by the water cells.

Global Rainfall Continuing Loss == [{0} | <CL_in_mm/h>]

(Optional)

Sets the continuing loss rate in mm/h for any global rainfall.

Global Rainfall Initial Loss == [{0} | <IL_in_mm>]

(Optional)

Sets the initial loss in mm for any global rainfall.

Read GIS BC == <gis_layer>

(Mandatory if not using fixed field text entry)

Reads the location and attributes of 2D model boundary conditions as described in Section 4.10.

Read GIS RF == <gis_layer>

(Optional)

Reads the polygons for applying rainfall directly to 2D cells as described in Table 4.32 and uses 2d_rf layers as described in Table 4.35.

Build 2008-08-AA or later (without any backward compatibility switches) should be used if using this command.

Read RowCol RF == <gis_layer>

(Optional)

Reads the rainfall cell by cell using just the .mid file (in a similar manner to other Read RowCol commands). The first two attributes of the .mid file must be the row and column of the 2D cell and the next three attributes must be as described in Table 4.35. To create this layer, select all ZC points from a 2d_zpt layer, save the selection as a another layer, restructure the attributes so that row (n) and column (m) remain as the first two, remove the other attributes, and add the attributes as described in Table 4.35. Update the Name attribute to one or more rainfall boundaries. To vary the rainfall spatially, create 3D surfaces (eg. using Vertical Mapper) of the total rainfall variation and carry out a point inspection of this surface for the f1 or f2 attribute.

Different proportions of different rainfall hyetographs can be applied by duplicating the layer and having one layer for each rainfall boundary. Create 3D surfaces of the proportion of each rainfall hyetograph to be applied and assign these values to the f1 or f2 attribute.

Build 2008-08-AA or later (without any backward compatibility switches) should be used if using this command.

```
Read GIS SA [ {} | ALL | PITS ] [ {} | RF ] == <gis_layer>
```

```
Read GIS SA TRIGGER == <gis_layer>
```

(Optional)

Reads the polygons for distributing source flows over the 2D domain(s) as described in Table 4.32. Usually used for specifying rainfall runoff directly onto the 2D domain(s).

As of Build 2005-05-AN, the RF (rainfall) option is available to specify rainfall hyetographs (mm versus hours) instead of flow hydrographs. See Table 4.32 for more information.

The ALL and PITS options were incorporated in Build 2008-08-AA. The ALL option is available to apply the flow/rainfall to all Code 1 cells (wet or dry active cells) within the polygon. Not applied to any inactive or water level boundary condition or HX 1D/2D linkage cells. If using the ALL option the double precision version may be needed as this is a similar approach to direct rainfall modelling.

The PITS option directs the inflow only to 2D cells that are connected to a 1D pit or node connected to the 2D domain using “SX” for the Conn_2D (previously Topo_ID) 1d_nwk attribute. The inflow is spread equally over the applicable 2D cells. If no 2D cells are found within the region an ERROR occurs.

The TRIGGER option allows 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 reaches the reservoir rather than at a fixed time. See Section 4.10.7.1 for more details.

```
Unused HX and SX Connections == [ {ERROR} | WARNING ]
```

(Optional)

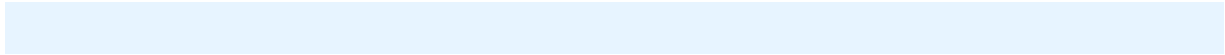
See [Unused HX and SX Connections](#) under .tcf file commands. The command can be used several times in a .tbc file to change from ERROR to WARNING and vice versa if a different level of checking is required for different 2d_bc layers. When reading and checking a 2d_bc layer, the latest occurrence of this command applies

Appendix E .tef File Commands

[1D](#)

[Define Event](#)

[End Define](#)



E.1 Event File Commands (.tef)

1D <ecf_command>	E-2
Define Event == <event_name>	E-2
End Define	E-2

1D <ecf_command>

(Optional)

To distinguish between 1D and 2D commands in a .tef file, prefix .ecf commands by “1D” followed by a space.

Define Event == <event_name>

(Optional)

Starts a block of .tcf and .ecf commands for an event named <event_name>.

Only one block can be specified for each unique <event_name>.

Use [End Define](#) to terminate the block. An ERROR occurs if [End Define](#) is not specified.

End Define

(Optional)

Ends a [Define Event](#) block of .tcf and .ecf commands for an event.

Appendix F New Features and Changes for Past Releases

F.1	2008-08 Release	F-2
F.2	2007-07 Release	F-13
F.3	2006-06 Release	F-24
F.4	2005-05 Release	F-33
F.5	Builds 2004-06-AC to 2001-03-AA	F-37

F.1 2008-08 Release

New software features and changes incorporated for the 2008-08 release are presented in the table below.

New Features and Changes for Build 2008-08-XX

Links	Description
Bug Fixes and Changes in 2008-08-AC	
Shallow Depth Stability Factor	The default value for Shallow Depth Stability Factor has been set to 0 (which disables this feature) for non-direct rainfall models. For most models this causes slightly different results for models compared with Builds 2008-08-AA and AB (see Shallow Depth Stability Factor for more information).
Read GIS Variable Z Shape	For Variable Z Shape cells, the initial water level is now adjusted with the adjustment of the ZC value for dry cells (otherwise water is generated if any dry cells start being lowered).
	Pathnames to and within the Pit Inlet Database are now correctly recognised.
Table 4.14	The 1d_nwk Height attribute was used as a flow multiplier instead of the Width attribute for Q pits.
	Symbology display problems in the 1d_bc_check.mif layer have been fixed.
Section 4.7.4.3	A number of major bug fixes relating to the new irregular (I) channel feature.
Interpolate Cross-Sections Section 4.6.5	A number of bug fixes and enhancements when using 1d_nwk connectors (X Type) to control which channels to traverse upstream and downstream when interpolating cross-sections as follows: • Pit channels are now ignored. • Parallel structures (including automatic W channels) now accepted when traversing upstream/downstream. • New ERROR message that checks that there is only one channel snapped to the start of a connector (if more than one, the channels should be separated and individual connectors digitised for each channel).
	Fixed bug that ignored one of the connectors if a channel had connectors starting at both ends of the channel.
Table 4.17	1d_tab HW (or CS) tables that used the N flag to vary Manning's n with height now work correctly (this fix is not backward compatible).

Links	Description
Section 7.4.4.1	The variation in structure loss coefficients has been moved out of the 1d_TSF.mif layer to its own .mif layer (extension _TSL.mif) due to a previously unknown limit in MapInfo of a total of 4,000 characters for all attributes of an object.
	The display of the “Stop TUFLOW Simulation” message box when an ERROR occurred during startup has been fixed.
	Better ERROR trapping for unconnected HX lines.
	TUFLOW no longer crashes when running more than one simulation on the same computer using the new Read GIS FC Shape and Read GIS Layered FC Shape features.
	TUFLOW can now set the timesteps (if specified in the .tcf file) when running via ISIS (this may require using an updated version of ISIS).
	-c (copy model) TUFLOW run option is now available to external 1D schemes such as ISIS and XP-SWMM.
Read GIS Z HX Line	Bug fixes and enhancements relating to the new Read GIS Z HX Line feature as follows: • HX lines with an “R” flag now apply the RIDGE approach. (The “R” flag cannot be applied to ZP points as documented in the 2008-08-AA manual). • New “A” flag for HX lines that adjusts all cell elevations (ie. does not apply the RIDGE approach if “Read GIS Z HX Line RIDGE ==” is specified). • If a mixture of “R” and “A” flags occur within the same layer, the “A” HX lines are applied first followed by the “R” lines.
Section 4.4.11	If a separate points layer is specified for a Read GIS Z Line, Shape or other command after the main layer, the path to this layer is now correctly assumed to be relative to the .tgc file. Also, if the WRITE TIN option is specified (where available) and a separate points layer is specified, this now correctly writes the .tin file.
	The 2d_sh_obj_check.mif layer now no longer contains empty polyline objects that caused MapInfo to not be able to import this layer.
	If -99999 elevation points were snapped to a polygon using Create TIN Zpts TUFLOW no longer goes into an infinite loop.
	If running the Tutorial Model without a dongle, the “_ TUFLOW Tutorial Simulations.log” file is now written to the same folder as the .tcf file (rather than to C:\) in case the user does not have write permission to C:\.
	If using Grid Size (N,M) , and a region is used for Read GIS Location , this command was not applied correctly.

Links	Description
	The 2d_dom_check.mif layer is now written immediately after setting the 2D domain dimensions (rather than just before the hydraulic calculations start), so is available for viewing if an ERROR occurs.
	Reporting of the viscosity formula and coefficient settings to the .tlf file has been corrected and improved.
	If using Viscosity Formulation == CONSTANT, the (constant) viscosity coefficient is now output to the _t.dat (viscosity coefficient) file.
	Added improved culvert input data tables and new Q pit input data tables to the .ecf file.
New Features and Changes for Build 2008-08-AA	
Defaults (.tcf file) Defaults (.ecf file – 1D only models) Table 9.1	Build 2008-08-AA uses new defaults as listed further below. If upgrading a model from the previous release (Builds 2007-07-XX), use Defaults == PRE 2008-08, if backward compatibility is required. The backward compatibility switches are documented under Defaults . The effects of the defaults that can influence the hydraulic calculations are described in Table 9.1.
Section 5.2 Model Precision Double Precision	Three (!) versions of TUFLOW are provided with this release. Intel Fortran single precision (iSP), Intel Fortran double precision (iDP) and Compaq Fortran single precision (cSP). For more information on which version to use see Section 5.2.
Write X1D Check Files	Outputs the water levels and flows between 2D domains and an external 1D scheme such as ISIS or XP-SWMM.
	Extended maximum line length of boundary database source files (ie. files accessed via BC Database) from 10,000 to 100,000 characters.
Read GIS Code	New INVERT option to reverse the Code value. Very useful for 2D-2D linking of different 2D domains. Also incorporated into Build 2007-07-BD.
Table 7.6	Separated out in and out flows across HX and SX connections for 1D/2D links into those between TUFLOW's 1D scheme (ESTRY) and those to an external 1D scheme. Overall mass balance reported does not include flows in any external 1D scheme, but does include flows across links to an external 1D scheme. Also incorporated into Build 2007-07-BD.
Read GIS SA	New ALL option for Read GIS SA and Read GIS SA RF that applies the flow/rainfall to all active (wet or dry) cells within the polygon. Ignores inactive or water level boundary condition cells.
Read GIS SA	New Read GIS SA PITS option that distributes SA inflows only to 2D cells connected to 1D pits or nodes using SX for the 1d_nwk Conn_2D attribute. The selected 2D cells are shown in the new 2d_sac_check.mif layer.

Links	Description
Table 4.14	New option for 1D pit and node SX connections only. If the 1d_nwk Conn_2D (previously Topo_ID) attribute is set to “SXL”, the 2D cell is lowered by the 1d_nwk US_Invert value of the pit, and the pit invert is set to the lowered 2D ZC level. This allows pits to be lowered (should only be slightly so don’t specify a large value for US_Invert) so that they “attract” the water. Primarily intended for use with the new Read GIS SA PITS command.
ESTRY Control File	For 1D/2D models, all 1D output filenames are now based on the .tcf filename, not the .ecf filename. This means that if the .ecf file does not change when setting up a new simulation based on a previous simulation, there is no need to make a copy of the .ecf file (ie. the .ecf file can be treated in a similar manner to the .tgc and .tbc files).
Zero Z Point	New .tgc command Zero Z Point that controls whether an ERROR or a WARNING is issued when points snapped to lines or polygons, or inside polygons, have a zero value (which usually indicates that the user has accidentally not assigned an elevation to that point).
Create TIN Zpts Read TIN Zpts Read GIS Z Shape Read GIS Variable Z Shape Section 4.4	New .tgc commands Create TIN Zpts , Read TIN Zpts , Read GIS Z Shape and Read GIS Variable Z Shape that offer greatly expanded options for modifying Zpts using TINs created externally, TINs created by TUFLOW and for specifying wide lines for embankments. Read GIS Variable Z Shape has the added benefit of being able to vary Zpt values over time to simulate, for example, the breaching of an embankment. See the new text and illustrated examples in Section 4.4.
Read GIS Z HX Line Table 4.33	New .tgc command to use HX lines to set the cell elevations of HX cells (eg. along the top of a levee).
Read GIS FC Shape Read GIS Layered FC Shape Section 4.7.2	New .tgc commands to make it much easier to create flow constriction (FC) cells, and a new feature, layered FCs, that allows FC attributes to vary with height. The latter means that is feasible to represent flow under and over a bridge structure in 2D. The bridge deck and rails can vary in 3D within the one shape, and different layers can have different percentage blockages. Another example is modelling a pipeline crossing a waterway.
	Now recognises XY coordinates of Point objects in a .mif file when written to the line after the keyword “Point”, as formatted by some GIS software.
Section 7.2.5	New or enhanced check files/layers: • 2d_zln_zpt_check.mif now contains a dZ attribute and new symbology that indicates whether an elevation point has been raised or lowered. • 2d_zsh_zpt_check.mif that contains Zpts modified by a 2d_zsh layer.

Links	Description
	• 2d_sh_obj_check.mif containing TINs and other objects to illustrate how shape objects modified cell values. • 2d_vzsh_zpt_check.mif which contains information on the Zpts that are varied over time from Read GIS Variable Z Shape commands. • 2d_fcsh_uvpt_check.mif for information on ZU and ZV points that are effected by Read GIS FC Shape commands. • 2d_lfcsh_uvpt_check.mif for information on ZU and ZV points that are effected by Read GIS Layered FC Shape commands. • 2d_sac_check.mif for checking Read GIS SA inputs.
Input Drive Output Drive	New .tcf commands Input Drive and Output Drive to change the drive letter of input files and output files that are specified with a full pathname.
	If a folder does not exist when starting up a TUFLOW simulation, TUFLOW now prompts the user if they wish to manually create the folder, or alternatively have TUFLOW create the folder.
	Writing of SMS .dat files is now 100 times faster!
Map Output Format	Map Output Format == SMS HIGH RES is now available for use. This option outputs not just at the cell corners, but also at the cell centres and mid-sides. See Map Output Format for more information. SMS recognises this high resolution format, but the utilities such as TUFLOW_to_GIS do not as yet. There is an option to use the high resolution approach but only output at corners.
Section 3.7 Viscosity Formulation Viscosity Coefficient	Several improvements to the viscosity term treatment: • Improvements to the dry boundary treatment of the viscosity term. • The calculation of the Smagorinsky viscosity coefficient is now cell-centred (was previously centred around the top right corner of the cell). • The viscosity coefficient now has a Smagorinsky component and a constant component. See Section 3.7 for more information.
	Inertia and viscosity terms are now not transferred across dry cell sides when constructing the coefficients for the solution arrays. This was having the effect of generating a circulation on the other side of the wall (albeit a weak one), which of course shouldn't happen!
Bed Resistance Depth Interpolation	New .tcf command Bed Resistance Depth Interpolation that controls how to interpolate the n values when using varying Manning's n with depth.
Verbose	New .tcf command Verbose that controls the amount of information displayed to the console window and .tlf file.
Maximum Points	New .tcf commands to allow the user to increase the maximum number of points in a GIS layer, and the maximum number of vertices in a single

Links	Description
Maximum Vertices	polyline or polygon, for commands such as Read GIS Z Shape .
Minimum NA	New “(%)” option for Minimum NA .
Map Output Data Types Section 7.3.1	Three new 2D map outputs: FLC (form loss coefficients); CWF (cell width factor); and Cr (Courant number). FLC and CWF are only available if Map Output Format == SMS HIGH RES and allow the user to view how the cell sides are being effected over time if using Read GIS Layered FC Shape . None are available for 1D WLL output as yet.
TIN Coincident Point Distance	New tgc command TIN Coincident Point Distance for setting the distance for removing coincident points prior to creating a TIN. Can be used several times in the .tgc file as needed.
Table 4.30	Can now have up to 10 QT region inflows assigned to a single 1D node (previous limit was only 1 QT region). This limit can be increased further upon request.
Table 4.31	New flag options C, O and P for 1D QT region boundaries to distribute flow to closed nodes, open nodes or pits. See Flags attribute in Table 4.31 for more information. Very useful for distributing catchment runoff from a hydrology model in urban areas.
Pit Inlet Database Section 4.5.1.1 Section 4.5.1.2	Pit channels can now be specified as a 1d_nwk Type of “Q” and the flow through the pit inlet is based on a depth-discharge curve extracted from the Pit Inlet Database . For more information see Section 4.5.1.1 and Section 4.5.1.2.
Section 4.5.1.1 Section 4.5.1.2 Section 4.5.1.3 Table 4.14	The 1d_nwk Branch, Topo_ID and XSect_ID or Chainage attributes have been renamed Inlet_Type, Conn_2D and Conn_No. These attributes provide the link to the Pit Inlet Database for the new Q pit channels, and useful functionality for automatically connecting pits and nodes to 2D domains. See Sections 4.5.1.1, 4.5.1.2 and 0, and Table 4.14 for further information. Note that these three attributes can still be named Branch, Topo_ID and XSect_ID or Chainage (they can actually be named anything) and can still be used for accessing cross-section data from MIKE11 and ISIS.
Table 4.14	Energy loss coefficients can now be assigned directly to pits and nodes using the 1d_nwk Form_Loss attribute. The loss is applied to all outgoing culverts, so the value should be based on the outgoing pipe velocity (eg. the K values). See Table 4.14 for more information.
Zero Culvert Coefficients	New .ecf command for setting default values for any zero culvert coefficient values. If this command is not specified, a zero value is interpreted as per previous builds.
Table 4.14	The 1d_nwk Use_Chan_Storage_at_Node attribute has been renamed UCS and changed from a Logical to a Char(1) attribute. There is no need to change existing 1d_nwk layers, although you can if you wish.

Links	Description
	The treatment for UCS is now as follows: • If left blank (the MapInfo default), it is treated as T (true) and the channel's storage is applied to the nodes at each end (unless a NA table overrides this). Previously, the Logical F (false) was the default, ie. don't apply the channel's storage at its nodes. • Other acceptable entries are: "T" or "Y" for true/yes, and "F" or "N" for false/no. The attribute is not case sensitive. This attribute is now correctly reproduced in the 1d_nwk_check.mif layer (prior releases always showed this attribute as F (false) irrespective of that specified in the 1d_nwk input layers).
Section 7.4.3	The 1d_nwk_check.mif layer has been split into two layers. 1d_nwk_check now only contains the channels (including pit channels), while the new 1d_nwk_N_check layer only contains the nodes. This makes it easier when interrogating and displaying these layers.
Weir Flow Defaults	New .ecf command Weir Flow. Method B provides improved stability of weir flows when the head drop across a weir is very small. Method A is the original approach and that used if Defaults == PRE 2008-08.
Set Code	New option ZERO ABOVE ZC for Set Code. For example, "Set Code Zero Above ZC == 50." applies a Code 0 (inactive) to all 2D cells in the domain that have a ZC value above 50. This command is very useful for setting the active area for direct rainfall models where the DEM has been trimmed to the catchment boundary. For more details see Set Code.
Culvert Flow Defaults	New Method E option for Culvert Flow. This incorporates minor improvements for transitioning between Regimes A and B, and between inlet and outlet controlled regimes, for circular culverts. Method E is the default as of Build 2008-08-AA. If Defaults == PRE 2008-08 is specified, Method D is the default setting.
Section 4.7.4.3 Table 4.14	New 1d_nwk culvert Type "I" for irregular shaped culverts. The cross-section shape of the culvert is specified in the same manner as for open channels (eg. as a XZ or CS/HW table). Contraction coefficients and flow regimes are as for R culverts.
Culvert Critical H/D	Culvert Critical H/D can optionally now have three values that apply depending on the type of junction at the upstream end of the culvert. See Culvert Critical H/D for more details.
Section 7.4.4.2	New 1D output layer _ccA.mif that contains information on the performance of culverts and bridges in terms of how effective they performed. The layer is coloured and uses different line thicknesses to illustrate how full different culverts/bridges flowed. There are several attributes that allow thematic mapping of other performance indicators such as the percentage of time the culvert/bridge flowed full.

Links	Description
	The full shape of 1D channel lines is now stored and used for layers such as 1d_nwk_check.mif and _ccA.mif. There has also been some adjustment of where points are located and of styles in the _TS.mif and _TSF.mif layers. Previously only the coordinates of the channel ends and mid-point were retained, therefore the channel alignment in the 1d_nwk_check.mif layer would appear different to that in the 1d_nwk input layer(s) for channels digitised with more than two vertices.
Table 4.17	An additional (optional) attribute has been added to the 1d_tab (1d_xs) layer to set the skew angle for XZ and HW/CS cross-section data.
Interpolate Cross-Sections Section 4.6.5 Defaults	Channel cross-sections are now interpolated from the nearest cross-sections. This means that a series of channels may now be digitised between two cross-sections, and the cross-section properties at each channel are linearly interpolated between the two cross-sections. The protocols applied when interpolating cross-sections and setting Manning's n values are described in Section 4.6.5. This feature can be switched off using the new .ecf command Interpolate Cross-Sections . If Defaults == PRE 2008-08 is specified, the cross-section interpolation is switched off for backward compatibility.
Interpolate Cross-Sections	New 1D check layer, 1d_xsl_check.mif, that outputs XS and CS lines used for interpolating cross-sections. Contains XS ID and other useful information on the cross-section properties, etc. Only output if Interpolate Cross-Sections == ON.
Interpolate Culvert Inverts Defaults	Unassigned culvert inverts (ie. 1d_nwk inverts assigned a value of -99999) are now interpolated by searching upstream and downstream for the nearest specified inverts and linearly interpolating the unassigned inverts. This feature can be switched off using the new .ecf command Interpolate Culvert Inverts . If Defaults == PRE 2008-08 is specified, the invert interpolation is switched off for backward compatibility..
Section 7.4.4.1	The entry, exit and any fixed culvert loss coefficients, and the bridge loss coefficient, are output to the _TSL.mif layer. If Structure Losses == ADJUST, or the A flag was specified in the 1d_nwk Type attribute, how the coefficients vary during the simulation can be viewed using the attributes of the _TSF.mif layer. For further information see Section 7.4.4.1.
Section 4.6.6.2	Negative material values in the M column of a XZM cross-section table can now be specified to disable (ie. block out or remove) sections of a cross-section. Previously a negative value would result in an ERROR message stating that that material could not be found. The negative value must be the negative of a valid Material ID.
	The area and other flow parameters are now output for CS/HW tables in

Links	Description
	the 1d_tables_check.csv file for bridges and weirs.
	Now adjusts the height of the _message.mif text for 1D messages according to the Y-direction extent of all the 1D domains. That is, for a model whose 1D domain extends over a large area a larger font size is used as opposed to a 1D domain of a small urban area.
	WLLs are now processed after all channels have been processed so as to not duplicate WLLs for overridden channels. This feature is switched off if Defaults == PRE 2008-08. This means: • If there are any overridden channels, the 2D WLL mesh will be different between the 2007 release and the 2008 release unless Defaults == PRE 2008-08 is specified. If the meshes are different the results cannot be compared in SMS. • For automatic culvert WLLs, overridden channels may have a different triangulation pattern where the width/diameter of culvert has changed due to the new culvert dimensions. This may cause issues when comparing results in SMS between the two simulations. This feature is switched off if Defaults == PRE 2008-08.
WLL Vertical Offset	New .ecf command WLL Vertical Offset to set the vertical adjustment of WLL elevations in the SMS .2dm file. The default value of 10 generally means that the 1D WLL output sits above the 2D cell output and is more visible in SMS (which presents data in order of increasing height). However, in 3D the 1D appears perched on top of the 2D. To show the WLL mesh at its correct height specify a WLL Vertical Offset value of 0.0.
	Removed the need for the _cnch.tmp file for reading structure coefficients. If Defaults == PRE 2008-08, the file is still used.
Cell Side Wet/Dry Depth	Cell Side Wet/Dry Depth has been disabled and is now always set to half the Cell Wet/Dry Depth value. The latter can also now not be below 0.0002m. Does not apply if Defaults == PRE 2008-08. Default values of 0.002 and 0.001m remain unchanged.
	New layers for have been incorporated into Write Empty MI Files .
	New WARNING messages for where Manning's n values (1D or 2D) fall below 0.01 or above 0.5.
	The computer name is now written to the .tlf file to provide improved traceability on which machine the simulation was carried out on.
Bug Fixes Post 2007-07-AA and Pre 2008-08-AA	
	Fixed bug that caused negative rainfall (eg. to model infiltration) to be set to zero and to be incorrectly applied in the mass balance calculations. Incorporated at Build 2007-07-BC.

Links	Description
	Fixed bug that caused problems if one 2D domain didn't have a .tbc file. Incorporated at Build 2007-07-BD.
	Fixed bug that didn't correctly apply Orientation Angle in a 2nd, 3rd, etc 2D domain if one of the previous 2D domains used Read GIS Location or Orientation . Incorporated at Build 2007-07-BD.
	Fixed bug relating to ISIS WLLs (occurred when WLL was located at the end of the last unit). Incorporated at Build 2007-07-BD.
	Fixed bug that sometimes didn't divide all of the SA RF values by the cell area. Incorporated into Builds 2006-06-CB and 2007-07-BE.
	Fixed bug that occurred if more than one automatic HQ boundary was specified. Would previously lump all the HQ boundaries together as shown in the 2d_bc_check.csv file. Incorporated at Build 2007-07-BF.
Bug Fixes Post 2007-07-BF	
	Fixed bug that would cause a crash if the 1D output interval was less than the 2D timestep. This occurred where Output Times Same as 2D was set to ON (the default), and Time Series Output Interval was not set.
	Fixed bug that would indeterminately try to read the 1D restart file. This would only occur on 2D only models using 2d_bc QT boundaries or 2D-2D links and with no 1D domain.
	Fixed intermittent bug that can allocate excessive amount of memory if an automatic HQ boundary is used.
	WLLs are now processed after all channels have been processed so as to not duplicate WLLs for overridden channels. This feature is switched off if Defaults == PRE 2008-08. This means: • If there are any overridden channels, the 2D WLL mesh will be different between the 2007 release and the 2008 release unless Defaults == PRE 2008-08 is specified. If the meshes are different the results cannot be compared in SMS. • For automatic culvert WLLs, overridden channels may have a different triangulation pattern where the width/diameter of culvert has changed due to the new culvert dimensions. This may cause issues when comparing results in SMS between the two simulations.
	Bug fix that didn't close files properly (does not affect results or runs, but explains how sometimes access to a file by another application that has been supposedly closed by TUFLOW).
	Fixed bug that did not correctly test or adjust 1D timestep when it exceeded minimum 2D timestep. Not backward compatible due to potential for mass loss if 1D timestep exceeds smallest 2D timestep (which now cannot occur).

F.2 2007-07 Release

New software features and changes incorporated between Build 2006-06-AA and 2007-07 builds are presented in the table below, and a summary of other changes and new features are discussed below.

The documentation on the old fixed field formats was removed from this manual for the 2007 release as they are essentially redundant (believe it or not it's a sad moment for some of us!). These formats are still supported by the software, but given they are rarely used, other than for a small number of old models, and by a few "hard-core" modellers, it was decided to remove them from the documentation. Documentation on the fixed field formats can be obtained by downloading one of the manuals prior to 2007 from www.tuflow.com and reading Appendix E.

New Features and Changes for Build 2007-07-XX

Links	Description
New Features and Changes	
Defaults (.tcf file) Defaults (.ecf file – 1D only models) Table 9.1	Build 2007-07-AA uses new defaults (not as many as for 2006-06-AA!) as listed further below. If upgrading a model from the previous release (Builds 2006-06-Bx), use Defaults == PRE 2007-07-AA, if backward compatibility is required. The new defaults and the backward compatibility switch are listed below and briefly described further down in the table. The affects of the defaults that can influence the hydraulic calculations is described in Table 9.1. 2D Domains (.tcf file) - The default material value is now zero (was previously one). For backward compatibility use Change Zero Material Values to One == ON in the .tcf file. - The routines that process polygons is now much faster. To use the old routines set Inside Region == Method A in the .tcf file. - The calculation of 1D water level weighting factors for 2D HX cells has been reworked. To use the previous approach set Line Cell Selection == Method C in the .tcf file. - New default approach for adjusting ZU and ZV values on VG cells that enhances stability. Use VG Z Adjustment == ZC in the tcf file for the previous approach. - Materials are now, by default, directly sampled at the cell mid-sides, in addition to the cell centres, thereby giving a higher resolution and better definition of Manning's n values. For backward compatibility use Bed Resistance Cell Sides == AVERAGE M. - The GIS projection is now checked on all input GIS layers. If the projection is not the same as specified by MI Projection, an ERROR

Links	Description
	occurs. For backward compatibility, use MI Projection Check == WARNING so as to not stop the simulation. 1D Domains (.ecf file) Further improvements/changes to some culvert flow regimes and the transitioning between regimes. To use the previous approach specify Culvert Flow == Method C and Culvert Critical H/D == 1.5 in the .ecf file.
Section 4.10.7	1D H (water level) boundaries can now be connected directly to a 2D HX boundary. This is particularly useful for setting a 2D water level boundary that varies spatially and temporally along its length (eg. a tidal boundary that varies in amplitude and phase along the length of the boundary).
Table 5.2 Section 7.1.2	Windows message boxes are now used to advise when the simulation has finished, has an error in the input data, etc. (Note, this feature was incorporated into the 2006-06-Bx builds but was not documented.) The message boxes can be switched off using the –nmb run-time option (See Table 5.2).
Section 9.3	Re-compiled to allow for models that use more than around 1.4Gb RAM.

Links	Description
Section 7.2.2	Continued improvements to ERROR/WARNING/CHECK message reporting. New messages added, and numbers have been assigned to messages within the 2D code that links to a database of messages and further explanation of likely causes for the message to occur. Note that some of the new ERROR messaging may cause old models to stop. It is strongly advised to check the message(s) and resolve the issue(s). Some of the new messages are: • WARNING 2074 if SA object does not select any 2D cells (active or inactive). • More warnings and checks for when pits lie within active 2D cells of more than one 2D domain. • Further checking that compatible boundary and linking conditions are occurring at 2D cells allocated more than one boundary and/or link. • ERROR 2084 message if a Read RowCol Zpts command is not reading a .mid file with the correct number of attributes (this usually occurs if a .mif file is accidentally specified). • ERROR 2071 message for checking that the cell side wet/dry depth is not greater than the cell wet/dry depth. If running with Defaults == Pre 2007-07-AA, then this message is issued as a WARNING 2071. • ERROR 2206 message that occurs if one or more Zpt elevations on active cells have not been initialised (if this message occurs then the Zpt elevation exceeds 99,998m, usually because the default Zpt value is 99,999m and it hasn't been assigned a value). Use Set Zpt in the .tgc file before other Zpt commands to set a global (flood-free) level. If running with Defaults == Pre 2007-07-AA, then this message is issued as a WARNING 2206.
Section 7.1.1 Section 7.5 Section 8.1	Mass balance reporting is now much more comprehensive and has been extended from just the 2D domains to now include: • Overall model (ie. combination of all 1D and 2D domains). • 1D domains • 1D/2D links Familiarisation and monitoring of the mass balance reporting is strongly recommended so that “unhealthy” models are detected and corrected.
Section 5.10	Added -b, -nmb, -mb, -t, -x run-time options to ESTRY.exe.
Section 7.4.4	_TS.mif files now automatically vary the number of output times so that they do not exceed the 250 attribute limit in MapInfo (ie. the whole hydrograph will now appear when graphing time-series output in MapInfo – this was previously not the case if more than 245 time-series output times occurred).

Links	Description
Section 7.1.1.1	Ctrl-C on the Console (DOS) window now pops up a message box asking whether the simulation is to be stopped. If Yes is selected the simulation is finished off neatly, and any network licence is released. Note that for some, yet unknown, reason Ctrl-C can only be pressed once (a second occurrence will terminate the process without finishing off the simulation neatly).
Table 4.32	If a negative rainfall (ie. infiltration) occurs at a 2D cell, it is only applied to wet cells (previously it was applied irrespective of whether the cell was wet or dry). This is useful for modelling infiltration from the surfacewater into the ground.
Read GIS Cyclone	Added background wind attributes to the Read GIS Cyclone GIS layer (these attributes will need to be added to existing layers).
Section 4.7.4.1 Structure Losses Table 4.13	New A and F flags for 1d_nwk B, C and R channels to individually override the global Structure Losses setting. Use A to adjust structure losses and F to fix. For example to fix the inlet and outlet losses on a rectangular culvert specify a Channel_Type of “RF”.
Section 4.11.2.2	Z values along WLLs for bridge channels are now always based on the processed data (ie. will overwrite any WLLp Z values). This ensures the bridge deck underside is correctly represented.
WLLP Interpolate Bed	New command WLLP Interpolate Bed that sets the centre WLL point to the channel bed based on the processed data (rather than use any value from a WLLp layer).
VG Z Adjustment	New option for VG Z Adjustment . Can now specify MAX before or after ZC option to force the adjusted ZU/ZV and ZH points to be set to the max ZC value rather than an interpolated value based on the surrounding ZC values. This option provides significant enhancements in some situations to the stability of the flow over the breach.
Section 4.11.2.1	If a WLL is snapped to a node, the processed data used for setting any bed elevations is now that from the higher channel unless it is a bridge in which case it uses the bridge processed data.
Line Cell Selection	Enhancement to selection of cells along 1D/2D HX Interface lines that uses a perpendicular interpolation from the 2D cell centres onto the line providing a “smoother” water surface profile along HX lines, and better stability along 1D/2D HX interfaces.
Table 5.2	New TUFLOW.exe run-time option to make a copy of a model. This is particularly useful for transferring a model from one site to another, or for making an archive or backup of the model. (It is invaluable for large models with many inputs!)

Links	Description
Section 4.10.5.2 Read GIS ISIS Network Read GIS ISIS Nodes	For ISIS linked models, the Node and Channel IDs in the 1d_isis.mif layers are now case sensitive to conform with ISIS. Therefore, the ISIS IDs must be identical in terms of case and characters to the IDs in the layers connecting ISIS and TUFLOW.
Culvert Flow Section 4.7.4.3	Further enhancements to culvert flow and stability. These include: - Retention of previous timestep entrance/exit velocities for both upstream and downstream controlled flow conditions (previously the prevailing regime was only stored), which provides smoother transitioning between upstream and downstream controlled flow regimes. - Much improved convergence for Regime C and tests for non-convergence (if does not converge, inlet control is assumed). - Allows unsubmerged downstream controlled flow Regime D to be included for zero length channels (usually pits), providing improved results in pit channels as surcharging starts. Previously only critical and fully submerged conditions applied. - Fix to Regime E (see Bug Fixes section of this table). These enhancements are set using Culvert Flow == Method D (the new default). For backward compatibility use Method C.
Culvert Critical H/D	New .ecf command Culvert Critical H/D that sets when downstream controlled flow regimes (Regime B to E or F) take preference over upstream controlled regimes based on the ratio of the upstream head above the culvert sill over the culvert height. The default is OFF (ie. infinitely large H/D). In previous builds a value of 1.5 was used. For backward compatibility use 1.5.
Table 4.14	New option for the Length_or_ANA attribute in a 1d_nwk layer: If the ANA value for a 1d_nwk point is < 0 then it is used as a multiplier on the NA for that node. For example, setting a value of -1.5 will increase the storage of that node by 50% (ie. multiply the NA table by 1.5). This may be useful in stabilising problematic 1D nodes, noting that excessively increasing the storage of a model may unacceptably distort the results.
Minimum NA Pit	New .ecf Minimum NA Pit command. This only applies to the upstream node of a pit channel, not the pit itself. If the Length_or_ANA 1d_nwk attribute value of a pit is less than or equal to zero, the upstream pit channel node will be assigned the Minimum NA Pit value. The default is a value of 1m².

Links	Description
Minimum NA Defaults	The Minimum NA value is now NOT applied to automatically created NA tables (this occurs for the upstream pit channel nodes and any node with an ANA value that is greater than 0.001 and there is no NA table that has been manually specified or created from channel storages). Backward compatibility only through using the Defaults == PRE 2007-07-AA command.
Minimum Channel Storage Length	New .ecf Minimum Channel Storage Length command. If a channel's length is less than the value specified, then the Minimum Channel Storage Length is used for calculating any storage contributions from the channel. This may be useful in stabilising short channels, noting that artificially increasing the storage of a model may unacceptably distort the results and should be sensitivity tested. The default is a value of zero metres.
	Objects in .mif file are now not case sensitive to be compatible with other software that export .mif files. Previously they were case sensitive based on that used by MapInfo.
Section 4.3	If a polygon or region's "Center" is not specified in the .mif file (this occurs for some software when exporting .mif files), it is now automatically calculated (previously TUFLOW would give a WARNING and ignore the region). The centre is estimated by averaging the x and y values (note that for Multiple Region objects, the result maybe unpredictable). TUFLOW uses the region's centre for some objects such as 2d_fc and 2d_po polygon objects.
Inside Region	New and much faster routine for assigning values to the TUFLOW 2D grid within a polygon from a .mif file. The new .tcf command Inside Region provides backward compatibility by using the old, slower approach (Method A). Method B is the new approach and is the default. Testing thus far has shown the two methods yield identical results although it is possible that if a point lies exactly on a polygon boundary different results may occur. To appreciate the increase in startup time this feature offers, testing on two large models reduced the startup time from 20 minutes to 3 minutes for one model, and from 40 minutes to 5 minutes for the other. The faster startup time occurs for any polygon layers being accessed from the .tgc and .tbc files, particularly those containing large number of vertices.

Links	Description
Bed Resistance Cell Sides Read GIS Mat	New .tcf command Bed Resistance Cell Sides that is used to define how the bed resistance value at a 2D cell's mid-side (ie. that used in the momentum equation) is calculated. The approach prior to this release was AVERAGE M which takes the average Manning's M (1/Manning's n) value of the two adjoining cell centre values. The AVERAGE n option takes the average Manning's n values of the cell centres, and INTERROGATE (the default) applies the exact value from the material polygons using Read GIS Mat. The INTERROGATE option provides a higher resolution sampling of material values compared with just sampling at the cell centres. This higher resolution sampling is particularly useful in modelling urban areas where frequent and large changes in Manning's n occurs.
Change Zero Material Values to One	New .tcf command Change Zero Material Values to One. The default material value is now zero (previously one), which means that every cell must be assigned a material value (ie. use Set Mat). A zero material value now produces an ERROR. For backward compatibility use Change Zero Material Values to One == ON.
	The maximum Material ID is now 32,767 (was previously 99,999). If material values greater than 32,767 were used, these will need to be reworked.
	Added new attribute, dH, to 1d_mmH.mif output layer that contains the largest head drop across any channels that end at that node (ie. only channels that are digitised so that their downstream end is at that node are used to determine dH). Provided channels are digitised from upstream to downstream this is very useful for identifying any increases in water level (thematically map the dH attribute) caused by any instabilities.
Section 5.3 Section 7.1.3 (for -slp switch)	New “_ TUFLOW Dongle <serial_no> Simulations.log” file for logging all simulations using that dongle to one location. For network dongles, the URL for where this is to be located needs to be coded into the dongle using the -slp run-time option. For standalone dongles this will also work. The default (if no URL has been specified) is to write the file to C:\.
Section 7.1.3	The _ TUFLOW Simulations.log files now contain the computer name and active licence site.

Links	Description
	Network dongle monitoring changed so that if the network is disconnected or crashes, or the TUFLOW Server goes down, during a simulation, TUFLOW then checks every 3 minutes until a new network licence is available. Also, if there is no network licence available when TUFLOW starts up because they are all used, TUFLOW keeps searching for a licence every 3 minutes until a licence becomes available. Of note is that “ghost licences” may occur on some networks when a simulation is running and the network is disconnected (either by pulling out a cable or the network being down). A notebook computer that started a simulation and is then removed from the network may also create a ghost licence. At present the only way to remove these ghost licences is to restart the TUFLOW Security Server (which can be done even if there are simulations running as TUFLOW will now keep checking for a network licence until it finds one). Ghost licences don’t seem to occur if a simulation is terminated by closing the window or killing the TUFLOW process. Different networks may yield different outcomes in terms of the generation of ghost licences.
BC Zero Flow	New .tcf command BC Zero Flow == [{OFF} START END START and END]” that zeros the start and/or end of 1D and 2D flow hydrographs (QT, ST, SA). This is particularly useful if the simulation extends outside the duration of the hydrograph and the start/end flow values need to be set to zero.
MI Projection Check	New .tcf command MI Projection Check == [{ERROR} WARNING] that checks that the GIS Projection for all input layers is the same as that specified by MI Projection . If Defaults == Pre 2007-07-AA is set, the default setting for this new command is WARNING.
	Increased limit on number of vertices of a single region (polygon) object to 100,000. Was previously 50,000. An ERROR is issued if this is exceeded.
	If TUFLOW is run from SMS, and there is no SMS TUFLOW executable licence, TUFLOW now searches for a TUFLOW dongle.
	TUFLOW Console Window now automatically resizes to 30 rows by 122 characters during model startup, and then to 40 rows by the number of characters required to display output during run time.
Section 7.1.1	New output to the Console Window during run time that displays the flow into the model (Qi), flow out (Qo) and the change in volume since the previous display time (dV).

Links	Description
	ZU and ZV elevations at 2D FC cell sides blocked using the 2d_fc Blocked_sides attribute were previously assigned an elevation of 100,001m. They are now assigned an elevation equal to the instability water level so as to not have extremely high elevations in the 2d_zpt_check.mif file. If Defaults == Pre 2007-07-AA is set, this new feature is not applied and the ZU/ZV elevations remain at the 100,001m elevation. This provides backward compatibility and will also cause the new WARNING 2206 message that the Zpts exceed 99998m at these ZU/ZV points (these warnings can be ignored!).
	Further checks have been incorporated into the computational code when testing for dry cell sides. This provides in some situations a slightly more robust solution. The change in results are typically negligible. If Defaults == Pre 2007-07-AA is set, this new feature is not applied so as to provide backward compatibility.
	2D HQ boundaries using the automatic stage-discharge curve option now take into account the variation in Manning's n with depth feature when calculating the stage-discharge curve. Previously, the Manning's n at the lower depth was used at all depths. This enhancement only applies if Defaults == PRE 2007-07-AA or PRE 2006-06-AA is not specified. Incorporated into Build 2007-07-AE.
Map Output Data Types	Map Output Data Types type ME (M1 in earlier releases) changed to MB1 for time varying 2D mass error % map output. ME caused _E.dat output to also be created. Incorporated into Build 2007-07-AF.
Bug Fixes	
	The 2d_uvpt_check.mif layer did not correctly write out for multiple 2D domain models when one or more 2D domains did not contain flow constriction cells, while others did.
	Bug fixes relating to pits: previous builds did not always correctly allocate pit SX connections for 2nd, 3rd, etc, 2D domains; occasionally a pit near the perimeter of a 2D domain was not included (TUFLOW would stop with an obscure message).
	Bug fix that did not correctly write out the number of vertical walls attribute in the 2f_fc_check.mif file.
	Bug fix that tried to assign automatic WLLs to culverts with a duplicate point along the 1d_nwk channel line.
	Bug fix that did not correctly write the upstream_invert level to the 1d_nwk_check.mif layer for the upstream pit channel node. (It was correctly written to the .eof file.) Does not affect the hydraulic calculations.

Links	Description
Check MI Save Date	Bug fix that if Check MI Save Date == WARNING is set and a warning is issued, TUFLOW goes into an infinite loop and crashes. Workaround was to set Check MI Save Date to OFF.
Structure Losses Section 4.7.4.3	Corrected problem with culvert Regime E flow when Structure Losses == ADJUST, that occurred due to the exit loss coefficient being adjusted when the full exit loss should apply. This would result in an abnormal jump in culvert flow while in Regime E. This fix is incorporated into Culvert Flow == Method D (the new default). For backward compatibility use Method C, however, occurrence and influence of Regime E flow should be reviewed.
Table 4.15	Changed the setting of the default width (if $eN1 < 0.001$) of automatic weirs over R and C channels (ie. RW and CW) to be the diameter/width multiplied by the number of culverts (previously, the width was not multiplied by the number of culverts). For backward compatibility, original weir width can be set by manually setting the eN1 attribute to the Diameter_or_Width attribute value of the culvert.
	Bug fix that when using a restart file TUFLOW occasionally set the 2D FC bridge deck additional loss value incorrectly. Not backward compatible.
	Bug fix that caused a MATH Error when writing the uvpt_check.mif file.
	Bug fix that accidentally connected a CN line to a hidden node created by a 2d_bc 2D or QT line. TUFLOW was previously stopping with a message that the CN line was unused or not snapped.
	Fixed up 1d_bc_check.mif file so that QT region inputs are now correctly shown (previously inflows at nodes with H boundaries, for example at a SX connection, were being shown when they weren't really there). 1d_bc_tables_check.csv file correctly showed the inputs.
	Bug fix that sometimes caused TUFLOW to not startup due to memory allocation problem if reading more than one 2D SA RF polygon. This fix also incorporated into 2006-06-BG.
	Bug fix that incorrectly set the water levels on dried VG cells (only applies to simulations with source inflows, eg. SA, RF). May cause slight changes in results, but and is likely to provide greater stability if there are dried VG cells. Backward compatibility provided if Defaults == Pre 2007-07-AA is set (noting that setting this command reinstates the bug). This bug also causes the mass error calculations to falsely give a mass error that is not occurring.

Links	Description
Section 4.7.2	Fixed bug that did not correctly apply the reduction in conveyance for a FC BD (bridge deck) of FD (floating deck) cell using the 2d_fc Mannings_n attribute. Backward compatibility applied if Defaults == PRE 2007-07-AA is set, however, note that this reinstates the bug and the resistance to flow at FC BD and FD cells may need to be reviewed. Indications are that only minor changes in results occur. The flow area under 2D FC BD and FD cells is correctly calculated.
	Fixed bug that didn't correctly recognise restart files created for 2D only models that utilised hidden 1D nodes (eg. 2D QT boundary and 2D-2D links).
	Fixed bug that if Write Check Files is set to OFF, a 2D2D link may generate mass and produce different results than if Write Check Files is set to ON.
Map Output Data Types	Map Output Data Types type MB1 (M1 and ME in earlier releases) bug fix that did not correctly calculate the mass error % along H boundaries and links. Incorporated into Build 2007-07-AF.
Utilities	
Section 11	The various utilities supplied “free” to TUFLOW licensees have all been updated and are available from www.tufLOW.com. They have finally also been documented (click on the relevant Section number below). The utilities available are: • TUFLOW_to_GIS.exe (Section 11.2) • dat_to_dat.exe (Section 11.3) • tin_to_tin.exe (Section 11.4) • 12da_to_from_mif.exe (Section 11.5) • asc_to_asc.exe (Section 11.6) • convert_to_tsl.exe (Section 11.7) • xsGenerator.exe (Section 11.8) • TUFLOW_Tools.xls (Section 11.9) • MapInfo Tools (Section 11.10)

F.3 2006-06 Release

New features and changes incorporated between Build 2005-05-AN and 2006-06-AA are presented in the table below.

New Features and Changes for Build 2006-06-AA

Links	Description
New Features and Changes	
Defaults (.tcf file) Defaults (.ecf file – 1D only models)	Build 2006-06-AA uses new defaults as listed further below. If upgrading a model from the previous release (Builds 2005-05-AN/Bx), use Defaults == PRE 2006-06-AA, if backward compatibility is required. The new defaults are: 2D Domains (.tcf file) Cell Wet/Dry Depth == 0.002 (previously 0.05) Cell Side Wet/Dry Depth == 0.001 (previously 0.03) Adjust Head at ESTRY Interface == OFF (previously ON) Boundary Cell Selection == Method C (previously Method A) Line Cell Selection == Method C (previously Method A) Viscosity Formulation == Smagorinsky (previously Constant) Viscosity Coefficient == 0.2 (previously 1.0) 1D Domains (.ecf file) Structure Losses == ADJUST (previously FIX) Storage Above Structure Obvert (%) == 5 (previously Storage Above Structure Obvert == CHANNEL WIDTH) Depth Limit Factor == 10 (previously 1) Culvert Flow == Method C (previously Method B) Culvert Add Dynamic Head == ON (previously OFF) Bridge Flow == Method B (previously Method A) WLL Approach == Method B (previously Method A) Apply All Inverts == ON (previously OFF – see note at end of Apply All Inverts) Conveyance Calculation == ALL PARALLEL (previously CHANGE IN RESISTANCE) Flow Calculation == Method B (previously Method A)

Section 5.2	TUFLOW now consists of three files: TUFLOW.exe, TUFLOW_LINK.dll and TUFLOW_USER_DEFINED.dll. These files must remain together within the same folder at all times. When archiving, place in a folder named using the Build ID, eg. 2006-06-AA. If running TUFLOW via ISIS or XP-SWMM2D, TUFLOW.exe is not required, but the two .dll files are.
Section 5.9	The new TUFLOW_USER_DEFINED.dll allows users to customise TUFLOW to suit their specific needs. In this build, it only contains the various flood hazard formulations. Users can modify the code, send it to support@tufLOW.com , and a new TUFLOW_USER_DEFINED.dll will be emailed back. In the future, much greater access is envisaged so that users can customise TUFLOW in many ways. Some knowledge of programming in Fortran or similar is required.
Map Output Data Types	New Z4 hazard output option as per the Australian Guidelines for Map Output Data Types . Also a new Z0 output that simply outputs the VxD product value.
Map Output Data Types UK Hazard Debris Factor UK Hazard Formula UK Hazard Land Use	The UK Hazard formulae and categories (based on DEFRA R&D Outputs: Flood Risks to People Phase Two Draft FD2321/TR1 and TR2) are available as map output. There are three new commands, UK Hazard Debris Factor , UK Hazard Formula and UK Hazard Land Use , and two new options, ZUK0 and ZUK1 for Map Output Data Types .
Map Output Data Types	Only one hazard map output can now be specified per simulation.
Map Output Format	SMS map output has two new options, HIGH RES and HIGH RES CORNERS ONLY (see Map Output Format). HIGH RES outputs results at all Zpt locations, ie. at the cell centres, mid-sides and corners, and uses a more sophisticated approach that includes taking into account weir and supercritical flow effects across cell sides. HIGH RES CORNERS ONLY, only outputs at the cell corners (as per usual), but uses the more sophisticated approach. These options are still being trialled and are known to have some minor problems at the time of publishing. It is recommended that they NOT be used until further notice is given on the www.tufLOW.com News page. The HIGH RES option will only work correctly with SMS Version 9.2 or higher. Also note that the utility programs available on www.tufLOW.com (eg. dat_to_dat.exe and TUFLOW_to_GIS.exe) have not yet been configured to handle the HIGH RES format. There will be a notification on the web page when this is done.

Read GIS Cyclone Read GIS Hurricane Density of Air Density of Water Wind/Wave Shallow Depths Map Output Data Types (AP and WI10 options)	The atmospheric pressure differential has been added to the 2D hydrodynamic equations, allowing the modelling of cyclones and other intense low-pressure systems. Cyclone wind and pressure fields are generated from a GIS layer of the cyclone track. The feature is presently being trialled and calibrated to Category 5 Cyclone Larry that struck the North Queensland coast in 2006.
Map Output Data Types	Two .sup files for loading map data into SMS are now created. They have the extensions .hV.sup (only loads the _h.dat and _V.dat files), and .ALL.sup (loads all .dat files specified by Map Output Data Types .
Table 4.32 Table 4.33	New 2D2D link line to interface two 2D domains. The line, digitised in a 2d_bc layer, connects two 2D domains. This feature is still under development and testing, and is likely to be subject to further improvements and changes.
	Reference time information is now included in the SMS .dat files, in accordance with the time referencing features in SMS Version 9.2.
Table 4.32 Table 4.33	New 2D QT boundary that automatically creates a 1D node, 1D NA table, CN lines and HX line, and applies the QT hydrograph to the 1D node (saves a lot of messing around!). The original 2D QT boundary (which no one uses) is still available by specifying an “A” (for angle) in the Flags attribute.
Table 4.32 Table 4.33	New 2D HQ boundary sets stage-discharge relationships for the downstream boundaries of 2D domains. Either specify a water level versus flow series in the BC database, or specify a slope in the 2d_bc “b” attribute from which TUFLOW automatically creates a HQ relationship based on the 2D topography and materials along the HQ line.
Table 7.1	New 2d_zln_zpts_check.mif GIS layer that shows which Zpts have been modified by Read GIS Z Line commands, the type of Z Line and the Z Line filename. This feature is very useful for tracing which Z Lines have modified which Zpts. Note, GULLY lines are not yet included.
Table 7.1	New 2d_bcc_check.mif GIS layer that replaces the 2d_bc_check.mif layer. The layer provides trace back information and uses cells, rather than point/line objects to show 2D BCs. The BC ID (eg. BC000001) allows easy trace back to the 2d_bc_tables_check.csv file.
Table 7.1	New 2d_uvpt_check.mif GIS layer that contains the initial velocities, roughness value, FLC, WrF, FC lid depth and FC BD factor at the U and V points.

Boundary Cell Selection Line Cell Selection Read GIS Z Line	Finally, HX lines and THICK Z Lines select the same cells. The new approach is referred to as Method C in Boundary Cell Selection and Line Cell Selection, and is the default in this build. Backward compatibility is provided via the Method A and Method B options. With Method C, hopefully now there is no need to patch up “holes” in levees and flood defence walls. The HX line and Z Line also do not need to be identical in terms of vertices, although, they do need to be identical in alignment to ensure they are compatible. This means that additional vertices can be inserted in, say, the Z Line as long as the line’s shape does not change. Note, the Read GIS Z Line CC option is incompatible with Method C and an ERROR message is given if it is specified when using Method C. Also, the HX option is redundant with Method C and, if specified, is treated the same as specifying the THICK option.
Boundary Cell Selection	The new “Method C” (see above) for selecting HX cells also applies to 2D boundaries that are defined using a line. As a consequence, a cell can only be assigned one boundary from a single GIS layer (except for: pit SXs; sink/source points or lines with two vertices only; and polygon boundaries such as SA and RF). If other boundaries are subsequently assigned (eg. to apply a storm surge on top of an ocean tide), these must be in separate GIS layers. Note that some boundary types cannot be assigned more than once to the same cell (eg. HX 1D/2D interface boundaries and the new 2D QT boundaries).
Oblique Boundary Method	The oblique boundary method has been improved and offers better stability for water level boundaries (HT, HS, HQ) and HX lines. To use the previous approach for backward compatibility use the tcf command Oblique Boundary Method.
Read GIS Z Line	New approach for interpolating Zpt values along a Read GIS Z Line for Line Cell Selection == METHOD C (the new default). The approach now takes the level of the Z line based on where the Z line is closest to the Zpt (ie. the perpendicular), or where no perpendicular intersection, the nearest vertex on the Z Line. For RIDGE (or MAX) flags, the highest value is chosen, even if there are closer Z lines. If RIDGE or MAX is not specified, the value from the closest eligible Z line is used. ADD works for both scenarios. This new approach should ensure that the modified Zpts remain unchanged even if the order of lines changes in the 2d_zln layer.

Section 4.10.7	Added additional error checks have been incorporated during the input of 2D BCs. TUFLOW will now stop with an error if a cell is: • assigned a HT or HS and is already a Q, S or HX cell • assigned a 2D or HX and is already any other boundary • assigned a Q or V and is already a H or S • assigned a ST, RF or SX and is already a H or Q Note these checks are only made if Boundary Cell Selection == METHOD C (the new default).
VG Z Adjustment	Variable Geometry (VG) 2D boundaries have been adapted to the ZC, ZU, ZV and ZH Zpt resolution (this feature was originally coded when only ZH Zpts were input). The original approach can be instigated using VG Z Adjustment.
	The sink/source in/out columns in the _MB.csv file has been further split into sink/sources from boundaries and sink/sources from SX connections to 1D nodes.
Instability Water Level	The default 2D instability water level now ignores any 99999 Zpt values, and is set to 10m higher than the highest ZC value.
Log Folder	Log Folder == command now also available in .ecf file for 1D only models.
	The 2D domain name has been added to a number of check GIS layers and messages to more easily identify which domain is being referred to for multiple 2D domain models.
Taper Closed NA Table Storage Above Structure Obvert	To improve stability in the 1D solution of pipes and culverts, new command Taper Closed NA Table, and the (%) option in Storage Above Structure Obvert are provided to transition the flow from partially full to full.
Table 4.33	New Flag “L” for HX objects in 2d_bc layers that sets the ZU and ZV elevations to be the same as the ZC value only if they are lower in height. This can improve stability where the ZU and ZV values are significantly lower in elevation, and can cause a sudden increase in transfer of water to/from the cell when the cell wets.
Table 4.33	New Flag “2” for HX lines in 2d_bc layers that can offer improved performance when using HX lines to connect 2D domains. This feature tries to more smartly allocate water levels along a HX line when there is a dry section between the 1D nodes.
BC Wet/Dry Method	The water level at a HX cell is now set to be not less than the ZC plus Cell Wet/Dry Depth value for when the 1D water level falls below the HX cell. This enhances stability in some situations. For backward compatibility use BC Wet/Dry Method == PRE 2005-11-AF.
	Increased maximum length of 2D domain names from 8 to 12 characters.

Interpolate ZC Interpolate ZHC Interpolate ZUV Interpolate ZUVC Interpolate ZUVH	New interpolate commands Interpolate ZUV and Interpolate ZUVH option to interpolate ZH, ZU and ZV values from the ZC values. These commands are useful when converting MIKE 21 models, which only specify elevations at the cell centres (ZC).
	Increased maximum line length to 10,000 characters for cross-section csv files.
Map Cutoff Depth	New tcf command Map Cutoff Depth that only outputs results for cells with depths above the cut-off depth. This feature is useful for direct rainfall modelling where there is a need to differentiate between sheet flow and flooding.
Read GIS RF Section 4.10.7	New tbc command Read GIS RF that applies rainfall directly to 2D cells. Section 4.10.7.
Read GIS SA Table 4.34	New RF option for Read GIS SA applies a SA boundary as a rainfall hyetograph. A SA RF layer must have five attributes: Name, Catchment_Area, Rain_Gauge_Factor, IL (initial loss) and CL (continuing loss). Catchment area is in m², IL in mm and CL in mm/h (see Table 4.34).
Global Rainfall Area Factor	Global Rainfall Area Factor is now applied to before applying initial and continuing losses. No backward compatible switch is available.
Table 4.14	New 1D channel flag “D” for C and R channels. If D is specified in the Channel_Type attribute, the culvert can only have downstream controlled flow regimes, unless it is a zero length channel (ie. channel length less than 0.01m).
WLL Automatic WLL No Weirs	New ecf command WLL Automatic for automatically generating 1D WLLs along culverts. The WLL will have the same width as the culvert width. This feature can save a lot of digitising! Another command, WLL No Weirs, stops the allocation of WLLs to weirs (this is useful for where a weir and a bridge/culvert are in parallel and you wish the WLL geometry to be based on the bridge/culvert rather than the weir over the top).
Read Materials File	Manning’s n values can now vary with depth on a material-by-material basis by adding data to the .tmf file. Four more values, in Columns 5 to 8 (Columns 3 and 4 are reserved for direct rainfall initial and continuing loss values), are optionally added for each material, namely y1, n1, y2, n2, where y is depth and n is Manning’s n. Below depth y1, n1 is applied, above y2, n2 is applied and between y1 and y2 the Manning’s value is interpolated. If these four values are specified, the primary Manning’s n value in Column 2 is not used.

Read GIS ISIS Network Read GIS ISIS WLL Read GIS ISIS WLL Points Read GIS XP Network Read GIS XP WLL Read GIS XP WLL Points Section 4.11.2	New WLL commands for the .tcf file that allows external 1D ISIS and XP-SWMM schemes to utilise the 1D WLL (Water Level Lines) feature that integrates 1D and 2D results in the map output.
Table 4.28	Added recognition of XP-SWMM interface file format (.int and .ext file extensions) for reading data files through the BC Database . The utility program convert_to_ts1.exe also recognises these files.
Table 4.28	Added recognition of XP-RAFTS 5g20 format in .tot and .loc files.
Distribute HX Flows	Built in alternative option for distributing the flow across HX lines to/from the 1D nodes. The distribution is based on a linear interpolation based on the distance of the HX cell from the 1D node. This option may improve model performance if the 1D/2D interface is being problematic. The feature is still under trial and should be benchmarked before adopting. It is not available for the ISIS 1D link as incorrect results presently occur. It has not been tested with the XP-SWMM 1D link.
Table 4.14	New connector feature in 1d_nwk layers that connects the end of a channel to the end of another channel. The line must be given a Channel_Type “X” (no other attribute data is needed). This is particularly useful for connecting a side tributary or pipe into the main flowpath. It also allows a different end cross-section to be specified for the side channel, rather than using the end cross-section on the main channel.
	Additional messaging has continued to be built in.
	1D map output via WLLs now includes energy (E) and Froude No (F) – these were previously set to zero.
Bug Fixes	
	Fixed bug that would search for a cross-section “SX” for a weir pit channel if using a M11 or ISIS x-sect database.
	If the 2D domain is not on a north-south orientation, the wrong cell was sometimes selected for SX connections to pits. This bug fix was also made to the previous TUFLOW release and placed as Build 2005-05-BA and BC on www.tuflow.com .
	Fixed bug that did not correctly apply the Storage Above Structure Obvert correctly. Presently no backward compatible switch, and should only slightly effect results. Does not affect results if CHANNEL WIDTH option specified (the default prior to Build 2006-03-AB).

Flow Calculation	The 1D flow values were incorrectly output where the channel flow regimes are oscillating every half timestep (for example, between super and sub-critical flow regimes). Where the channel is switching flow regimes between timesteps (nearly always the case), the correct flow is calculated. This fix also affects the flow in/out of 2D SX connections if the connected 1D channel is affected. The bug fix does not change 1D water level and velocity results, unless they are influenced by changes due to any effects on SX flows. Use the Method A option in Flow Calculation for backward compatibility.
	Fixed bug that did not correctly display some messages in the _messages.mif layer when viewed in MapInfo.
	FLC coefficients (using Read GIS FLC) for point objects were not being divided by two when applied to each cell mid-side, and were not being correctly accumulated at cell sides. This does not affect line and polygon objects. This bug fix is not backward compatible.
	If a pit channel, after extension northwards by the Pit Channel Offset , accidentally snaps to another 1D channel, they are no longer connected.
	Fixed bug relating to 2D PO not being written to _TS.mif file for the 2nd, 3rd, etc 2D domains.
	Fixed bug relating to selecting cells inside polygons using the various Read GIS commands. The problem occurs when several consecutive vertices lie along a horizontal line that intersects the cell centre. Also fixed a similar bug that occurred when duplicate vertices in a polygon are the last two vertices. Both of these bugs are very rare! There is no backward compatible switch.
	Fixed bug where if Write PO Online == ON and Start Time Series Output is after Start Time , the simulation would crash.
	Fixed bug that occurred if the second line of the .ecf file is blank, the 1D start time would be reset to zero and not be correctly set to the tuflow .tcf start time (if non-zero). The problem related to providing backward compatibility for old fixed field formats.
	Fixed bug that prevented SX U links from working correctly. This bug was introduced in January 2005 due to reworking of the code to allow different timesteps in different domains. SX U cells now appear in the 1d_to_2d_check.mif layer.
	Fixed bug that included in SMS .2dm mesh file, WLLs that did not snap or connect to a channel or node – this caused zero nodes to occur in the 2dm file and caused SMS not to be able to open the file. This problem was introduced at 2004-11-AA when WLLs not connected to a channel/node was changed from an ERROR to a WARNING.

	If a ST flow is negative and the cell is dry, no flow is now extracted from the cell (previously TUFLOW would extract flow from the cell even if it was dry). This change is not backward compatible.
Utilities	
	There have been continued enhancements to utilities for preparing data and post-processing results. These utilities are available as freeware, and can be downloaded from www.tuflow.com. Documentation generally remains limited, but is planned to be included in this manual in the future. Also, monitor www.tuflow.com for new or updated utilities.

F.4 2005-05 Release

New features and changes incorporated between Build 2004-06-AC and 2005-05-AN are presented in the table below.

New Features and Changes for Build 2005-05-AN

Links	Description
New Features and Changes	
Section 5.2	TUFLOW.exe split into two files: TUFLOW.exe and TUFLOW_LINK.dll. These files must remain together within the same folder at all times. When archiving, place in a folder named using the Build ID, eg. 2005-05-AN.
Section 4.5.1 Section 4.5.1.1 Table 4.14 Pit Channel Offset	Pit objects in the 1d_nwk layer and automatic creation of pit channels.
Section 4.5.1 Table 4.14	Use of node attributes to set channel upstream and downstream inverts.
Section 4.5.1 Table 4.14	Use of 1d_nwk Length attribute to add additional storage to a node or to automatically generate a NA (node storage) table. Length attribute on any node must be zero for backward compatibility.
Apply All Inverts Table 4.14	New command Apply All Inverts to use the upstream and downstream invert attributes to all channels (previously only applied to C, G, S, R and rectangular W channels).
Conveyance Calculation Section 4.6.7	New command Conveyance Calculation offers an alternative approach to calculating channel conveyance without the risk of conveyance reducing with height.
Section 4.6.4	Cross-sections specified using a 1d_tab (often named 1d_xs) layer can now be specified at the channel ends for G and S channels. The upstream and downstream inverts for these channels can optionally be based on the beds of the cross-sections.
Storage Above Structure Obvert Section 4.6.2.3	Storage contributions from B, C and R channels above their obvert can now be more accurately calculated using Storage Above Structure Obvert .
Bridge Flow Section 4.7.4.2	Improved stability for bridges when they wet and dry and changes to interpretation of the BG loss coefficients table available using Bridge Flow = Method B. The Method A option uses the original bridge routines and remains the default.

Table 4.13 Table 4.15	Automatic generation of weir channels to represent the overtopping of B, C and R channels. Also includes allowances for representing rails and blockage.
Table 4.14 Section 4.7.4.1 Structure Losses	1d_nwk Form_or_Bend_Loss attribute now used to model additional dynamic head losses in culverts (previously not used).
Table 4.14 Section 4.7.4.2 Section 4.7.4.1 Structure Losses	1d_nwk Form_or_Bend_Loss attribute for B channels can be used to automatically create a BG loss coefficient table or specify an additional loss to a user-defined BG table.
Structure Losses Section 4.7.4.1	New .ecf command Structure Losses to either fix structure entrance and exit losses or to adjust them according to the approach and departure velocities in the upstream and downstream channels.
Read GIS Z Line	HX option in Read GIS Z Line that uses the same approach for selecting cells along a HX line.
Snap Tolerance	New Snap Tolerance command for the .tcf and .ecf files that allows changing the search radius used to determine whether two objects are snapped.
Check Inside Grid	New .tcf Check Inside Grid command to allow objects in 2d_po, 2d_fc and some other layers to extend outside the 2D domain.
	“none” objects now recognised in .mif layers. “none” objects occur when attribute data has been entered but no object has been digitised. TUFLOW does not use “none” objects, but would previously cause problems reading a .mif file if “none” objects existed.
Table 4.28	New Group option for specifying inflow hydrographs. Can drastically reduce the size of the BC_database.csv file.
Section 4.10.6 Table 4.30	Regions now recognised for 1D QT boundaries in 1d_bc layers. The flow hydrograph is distributed equally to all nodes that fall within the region (except H boundary nodes).
Table 4.32 Read GIS SA Read Materials File	New Rainfall option in the Read GIS SA command that allows the application of rainfall depths to all active cells, whether wet or dry, within each SA region. Initial and continuing loss rates can be applied on a material-by-material basis. Note, this feature is undergoing testing and maybe subject to change. In particular, initially high mass errors can occur at the start when the whole model suddenly becomes wet (these are normally small in the overall simulation, but need to be monitored). Very small Cell Wet/Dry Depth and Cell Side Wet/Dry Depth depths have been found to assist, as well as using a smaller timestep.
	The startup time for reading and processing 1D domain data has been markedly reduced by up to a factor of five.

Head Rate Limit Section	The Head Rate Limit command is now available for stabilising problematic 1D nodes. Use with caution as inappropriate use may cause unacceptable mass errors.
Section 3.6 Timestep (.tcf) Timestep (.ecf)	A different timestep can now be specified for the 1D domain and for different 2D domains.
Section 7.5	Mass balance output is available for 2D domains (see Mass Balance Output) and automatically output for 1D domains. Note, this feature is not yet complete and may be subject to change.
Map Output Data Types Table 7.2	Two new flood hazard categories (Z2 and Z3) have been added to the Map Output Data Types options.
Table 7.1	Further enhancements to the symbology of the 1d_nwk_check.mif file to show whether nodes are connected to two or more, one or no channels – useful for detecting unsnapped nodes and channels.
Table 7.1	Upstream and downstream nodes of a channel are now included as attributes to the 1d_hydroprop_check.mif file.
Table 7.1 Write MI Domain	New 2d_dom_check file that contains a rectangle showing the extent of each 2D domain. Can also generate individual domains using Write MI Domain in the tgc file.
Section 7.2.2	Further changes to the _messages.mif file to de-clutter it! More changes are also planned in this regard.
Bug Fixes	
	The .wor file can now handle up to 1,000 unique GIS layers, and won't stop the simulation once the 1,000 has been reached. Previously there was a limit of 100 that would also stop the simulation. Yes, someone has more than 100 separate GIS layers in a model! A bug that inserted two back slashes (ie. \\) in the .wor file in some situations has been fixed.
	Fixed a bug that assigned negative 1D velocities a positive flow direction in the SMS_V.dat and _q.dat files.
	Flood hazard Z1 output now correctly works when the 1D velocity is negative.
	Fixed a bug that correctly allocates a 2D boundary cell when the first segment in a boundary line falls entirely within the first 2D cell.
	If one or more column headings in a BC .csv file could not be found, the first (time) column was used for values. ERROR message now issued if this occurs.
	Lowest cell in a SA region was not being correctly found when using a restart file and if every cell was dry at the time of the restart file

	Global Rainfall Initial Loss was not correctly being applied when the amount was greater than that in the first rainfall depth
New Modules	
Section 4.4.13	The multiple 2D domains module has completed testing and is now available for purchase (contact support@tufLOW.com or visit www.tufLOW.com).
Utilities	
	There have been a number of changes and new utilities for preparing data and post-processing results. Sms_to_mif.exe has been replaced by TUFLOW_to_GIS.exe, which offers extended capabilities including generation of longitudinal profiles that can optionally show nearby calibration levels. It also can now handle over a million cells (yes, someone has cracked the 1,000,000 mark!). Several MapBasic utilities are being made available that include a bulk .mif file exporter and an improved time-series graphing tool. These utilities are available as freeware, and can be downloaded from www.tufLOW.com. Documentation generally remains limited, but is planned to be included in this manual in the future. Also, monitor www.tufLOW.com for new or updated utilities.
Web Page	
www.tufLOW.com	And last, but not least, there is finally a TUFLOW web page! Visit www.tufLOW.com for information, downloading the latest builds, manuals, utilities, etc.

F.5 Builds 2004-06-AC to 2001-03-AA

The new features and changes to TUFLOW and ESTRY from March 2001 to Build 2004-06-AC have been removed for this manual edition. Please see previous manuals available from www.tuflow.com should you require this information or email support@tuflow.com if the manual you require is not available.

Appendix G Command Hyperlinks

.tcf File Commands

Adjust Head at Estry Interface	Event File	Model Events
Apply Wave Radiation Stresses	Excel Start Date	Model Platform
Apply Wind Stresses	Extrapolate Heads at Flow Boundaries	Model Precision
BC Control File		Model Scenarios
BC Database	First Sweep Direction	Model TUFLOW Build
BC Event Name	Free Overfall	Model TUFLOW Release
BC Event Source	Free Overfall Factor	
BC Event Text	Froude Check	Negative Depth Approach
BC Wet/Dry Method	Froude Depth Adjustment	Null Cell Checks
BC Zero Flow		Number Iterations
Bed Resistance Cell Sides	Geometry Control File	Number 2D2D Link Iterations
Bed Resistance Depth Interpolation	Global FC Ch Factor	
Bed Resistance Values	Global Weir Factor	Oblique Boundary Alignment
Boundary Cell Selection	HQ Boundary Approach	Oblique Boundary Method
	HX ZC Check	Output Drive
Calibration Points MI File		Output Folder
Cell Wet/Dry Depth	If Scenario	
Cell Side Wet/Dry Depth	Input Drive	Rainfall Boundaries
Cell Size	Inside Region	Rainfall Gauges
Change Zero Material Values to One	Instability Water Level	Recalculate Chezy Interval
Check Inside Grid	Latitude	Read File
Check MI Save Date	Line Cell Selection	Read Materials File
Check MI Save Ext	Link 2D2D Approach	Read GIS Cyclone
CSV Time	Log Folder	Read GIS Hurricane
		Read GIS ISIS Network
Defaults	Map Cutoff Depth	Read GIS ISIS Nodes
Density of Air	Map Output Data Types	Read GIS ISIS WLL
Density of Water	Map Output Format	Read GIS ISIS WLL Points
Depth/Ripple Height Factor Limit	Map Output Interval	Read GIS FC
Display Water Level	Mass Balance Output	Read GIS GLO
Distribute HX Flows	Mass Balance Output Interval	Read GIS IWL
Double Precision	Maximum Points	Read RowCol IWL
	Maximum Velocity Cutoff Depth	Read GIS LP
End 1D Domain	Maximum Vertices	Read GIS PO
End 2D Domain	Meshparts	Read GIS XP Network
End Time	MI Projection	Read GIS XP Nodes
ESTRY Control File	MI Projection Check	Read GIS XP WLL
		Read GIS XP WLL Points
		Read Restart File
		Screen/Log Display Interval

Set Route Cut Off Type	Supercritical	
Set Route Cut Off Values	SX Head Adjustment	Warmup Time
Set IWL	SX ZC Check	Water Level Checks
Snap Tolerance		Wave Period
Shallow Depth Stability Factor	Time Series Output Interval	Wetting and Drying
Shallow Depth Weir Factor Cut Off Depth	Time Output Cutoff Depths	Wind Output Interval
Shallow Depth Weir Factor Multiplier	Timestep	Wind/Wave Shallow Depths
Simulations Log Folder	Timestep During Warmup	Write Check Files
Start 1D Domain	UK Hazard Debris Factor	Write Empty MI Files
Start 2D Domain	UK Hazard Formula	Write PO Online
Start Map Output	UK Hazard Land Use	Write Restart File at Time
Start Time	Unused HX and SX Connections	Write Restart File Interval
Start Time Series Output	Verbose	Write X1D Check Files
Start Wind Output at Time	Viscosity Coefficient	
Store Maximums and Minimums	Viscosity Formulation	XF Files
	VG Z Adjustment	Zero Negative Depths in SMS

.tgc File Commands

Allow Dangling Z Lines	Pause When Polyline Does Not Find Zpt	Read TGF Read TIN Zpts
Cell Size Create TIN Zpts	Read File Read GIS Read GIS Code Read GIS FC Shape Read GIS Layered FC Shape Read GIS Location Read GIS [Mat IWL CnM Fric WrF FLC] Read GIS Variable Z Shape Read GIS Z HX Line Read GIS Z Line Read GIS Z Shape Read GIS Z Shape Route Read GIS Zpts Read RowCol [Code Mat IWL CnM Fric WrF] Read RowCol Grid Read RowCol Zpts	Set [Code Mat IWL CnM Fric WrF] Set Route Cut Off Type Set Route Cut Off Values Set Zpt Stop TIN Angles TIN Coincident Point Distance Write MI Domain Write MI Grid Write MI Zpts ZC == MIN(ZU,ZV) Zero Z Point
Default Land Z		
External Bndy		
Grid Size (N,M) Grid Size (X,Y)		
Interpolate ZC Interpolate ZHC Interpolate ZUV Interpolate ZUVC Interpolate ZUVH		
Orientation Orientation Angle Origin		

.tbc File Commands

BC Database BC Event Name BC Event Text	Global Rainfall Continuing Loss Global Rainfall Initial Loss	Read GIS SA Read RowCol RF Unused HX and SX Connections
Global Rainfall Area Factor Global Rainfall BC	Read GIS BC Read GIS RF	

.tef File Commands

1D	Define Event	End Define

.ecf File Commands

[Apply All Inverts](#)

[BC Database](#)

[BC Event Name](#)

[BC Event Text](#)

[BG](#)

[BG Data](#)

[Bridge Flow](#)

[Check MI Save Date](#)

[Check MI Save Ext](#)

[Create Nodes](#)

[CS](#)

[CS Data](#)

[CSV Format](#)

[CSV Time](#)

[Conveyance Calculation](#)

[Culvert Add Dynamic Head](#)

[Culvert Critical H/D](#)

[Culvert Flow](#)

[Defaults](#)

[Depth Limit Factor](#)

[EB Data](#)

[End Time](#)

[Flow Area](#)

[Flow Calculation](#)

[Froude Check](#)

[Froude Depth Adjustment](#)

[Head Rate Creep Factor](#)

[Head Rate Limit](#)

[Head Rate Limit Minimum](#)

[Interpolate Cross-Sections](#)

[Interpolate Culvert Inverts](#)

[Log Folder](#)

[M11 Network](#)

[MI Projection](#)

[Manhole Default Loss](#)

[Approach](#)

[Manhole Default C Exit](#)

[Coefficient](#)

[Manhole Default R Exit](#)

[Coefficient](#)

[Manhole Default Side](#)

[Clearance](#)

[Manhole Default Type](#)

[Manhole K Maximum](#)

[Bend/Drop](#)

[Manhole Minimum](#)

[Dimension](#)

[Manholes at All Culvert](#)

[Junctions](#)

[Minimum Channel](#)

[Conveyance Length](#)

[Minimum Channel Storage](#)

[Length](#)

[Minimum NA](#)

[Minimum NA Pit](#)

[Momentum Equation](#)

[NA](#)

[NA Data](#)

[Order Output](#)

[Output Folder](#)

[Output Interval](#)

[Output Times Same as 2D](#)

[Pit Channel Offset](#)

[Pit Default 2D Connection](#)

[Pit Inlet Database](#)

[Pit Search Distance](#)

[Read File](#)

[Read Materials File](#)

[Read GIS BC](#)

[Read GIS IWL](#)

[Read GIS Manhole](#)

[Read GIS Network](#)

[Read GIS Table Links](#)

[Read GIS WLL](#)

[Read GIS WLL Points](#)

[Relative Resistance](#)

[S Channel Approach](#)

[Set IWL](#)

[Snap Tolerance](#)

[Start Output](#)

[Start Time](#)

[Storage Above Structure](#)

[Obvert](#)

[Structure Losses](#)

[Structure Losses Approach](#)

[Taper Closed NA Table](#)

[Timestep](#)

[Trim XZ Profiles](#)

[Vel Rate Creep Factor](#)

[Vel Rate Limit](#)

[Vel Rate Limit Minimum](#)

[VG](#)

[VG Data](#)

[Weir Flow](#)

[WLL Additional Points](#)

[WLL Adjust XS Width](#)

[WLL Approach](#)

[WLL Automatic](#)

[WLL No Weirs](#)

[WLL Vertical Offset](#)

[WLLp Interpolate Bed](#)

[Write CSV Online](#)

[Write Check Files](#)

[Write Empty MI Files](#)

[XS Database](#)

[Zero Culvert Coefficients](#)