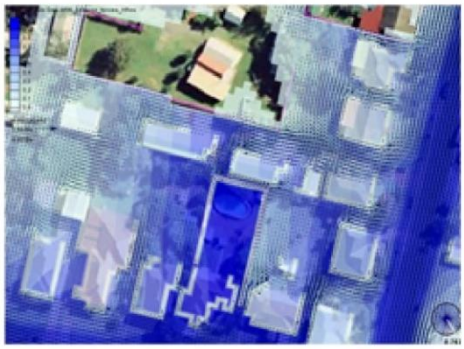
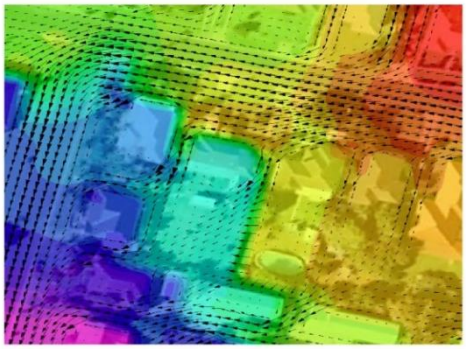
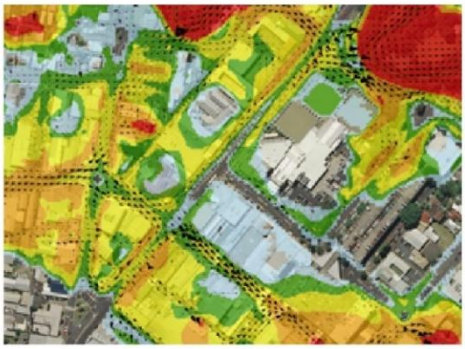






TUFLOW User Manual

Build 2016-03-AA

www.tuflow.com
TUFLOW Forum
TUFLOW Wiki
TUFLOW Tutorial Model
support@tuflow.com

How to Use This Manual
Chapters
Table of Contents
List of Figures
List of Tables
Appendices

.tcf Commands
.tgc Commands
.tbc Commands
1D (.ecf) Commands
.toc Commands
.trfc Commands

How to Use This Manual

This manual is designed for both hardcopy and digital usage. It is provided in both its native Microsoft Word 2013 version and as a pdf. The pdf version is faster to navigate around and, for the first time, seems to have all links preserved (previously the manual was only issued as a Word 2010 document as not all links were preserved in the pdf conversion).

Section, Table and Figure references are styled like this and linked. In Word to go to the link hold down the control (Ctrl) key and click on the Section, Table or Figure number in the text to move to the relevant page. In pdf, the links should be directly accessible.

Similarly, and most importantly, script or control file commands are hyperlinked and are easily accessed through the lists at the end of the manual. To quickly go to the end of the manual press Ctrl End. There are also command hyperlinks within the text (normally blue and underlined). Command text can be copied and pasted into the text files to ensure correct spelling.

Web and email links are styled like this. An increasing amount of content now resides on the TUFLOW Wiki with web links provided in the document to these pages. Other useful keys are Alt Left / Right arrow to link backwards / forwards to the last locations. Ctrl Home returns to the front page, which contains useful links.

A secondary window can be opened in Word by selecting View, New Window or in Adobe Acrobat by selecting Window, New Window, allowing you to view different sections of the document in different windows. For example, the TUFLOW command lists could be viewed in one window and the section describing the functionality in another window.

Constructive suggestions are always welcome (please email support@tuflow.com).

About This Manual

This document is the User Manual for the TUFLOW and TUFLOW GPU hydrodynamic computational engines for the 2016-03 release. Of particular note for users of earlier releases is that superseded content relating specifically to prior TUFLOW releases, especially prior to 2011-09, has largely been removed.

For changes to different releases refer to the links in Chapter 16, and for those particular to 2016-03 see the 2016-03 release notes.

If simulating models using prior TUFLOW builds, reference may need to be made to the release notes and documentation relevant for that particular build. These can be downloaded from the All TUFLOW Downloads page or requested from support@tuflow.com.

Chapters

1	Introduction	1-1
2	Getting Started	2-1
3	Model Design	3-1
4	Control Files and GIS Layers	4-1
5	1D Domains	5-1
6	2D Model Domains	6-1
7	Boundaries and Initial Conditions	7-1
8	Linked 1D and 2D Models	8-1
9	Customising Output	9-1
10	TUFLOW GPU Solver	10-1
11	Managing and Starting Simulations	11-1
12	Check and Log Files	12-1
13	Viewing and Post-Processing Output	13-1
14	Quality Control	14-1
15	Utilities	15-1
16	New (and Past) Features and Changes	16-1
17	References	A-1

Appendix A .tcf File Commands A-3

Appendix B 1D (.ecf) Commands B-1

Appendix C .tgc File Commands C-1

Appendix D .tbc File Commands D-1

Appendix E .toc File Commands E-1

Appendix F .trfc File Commands F-1

Appendix G Hyperlinks to All Commands G-1

Table of Contents

1 Introduction 1-1

1.1 Introduction 1-2

1.2 TUFLOW (Classic and GPU) 1-3

1.2.1 TUFLOW 2D Implicit Solver 1-3

1.2.2 TUFLOW 1D Solver (ESTRY) 1-3

1.2.3 TUFLOW 2D Explicit GPU Solver 1-4

1.2.4 TUFLOW Advection Dispersion and Heat Balance (AD) Module 1-5

1.2.5 Sediment Transport and Morphology (ST) Module 1-6

1.3 TUFLOW FV 1-7

1.4 Limitations and Recommendations 1-8

1.4.1 UK Benchmarking Study 1-9

1.5 Modelling Environment 1-11

2 Getting Started 2-1

2.1 The TUFLOW Modelling Concept 2-2

2.2 Graphical User Interface (GUI) Options 2-5

2.3 Installing and Running TUFLOW 2-6

2.3.1 TUFLOW Downloads and Installation 2-6

2.3.2 USB Locks (Dongles) and Licencing 2-6

2.3.3 Performing Simulations 2-6

2.4 Licence Free Simulations 2-7

2.4.1 Tutorial Models 2-7

2.4.2 Demo Models 2-7

2.4.3 Free Mode 2-8

2.5 Tips and Tricks 2-9

2.6 Folders and File Types 2-1

2.6.1 Suggested Folder Structure 2-1

2.6.2 File Types 2-2

2.6.3 Naming Conventions 2-7

3 Model Design 3-1

3.1 Introduction 3-2

3.2 Model Schematisation 3-3

3.3 Model Resolution 3-4

3.3.1 2D Cell Size 3-4

3.3.2 1D Network Definition 3-5

3.4 Computational Timestep 3-6
3.4.1 2D Domains (Courant Number) 3-6
3.4.2 1D Domains (Courant Number) 3-6
3.4.3 1D/2D Models 3-7
3.4.4 Adaptive Timestep 3-7
3.5 Simulation Times 3-9
3.6 Eddy Viscosity 3-10

4 Control Files and GIS Layers 4-1
4.1 Introduction 4-2
4.2 Control File Rules and Notation 4-3
4.3 Absolute and Relative File Paths 4-5
4.4 Units – Metric or US Customary/English) 4-6
4.5 TUFLOW Control File (.tcf file) 4-7
4.5.1 _TUFLOW_OVERRIDE Files 4-8
4.6 1D Commands 4-10
4.7 Geometry Control File (.tgc file) 4-11
4.8 Boundary Control File (.tbc file) 4-13
4.9 GIS Formats 4-14
4.9.1 “GIS” or “MI” Commands 4-14
4.9.2 “RowCol” or “MID” Commands 4-15
4.9.3 GIS Object Interpretation 4-15
4.10 XF Files 4-17
4.11 Fixed Field Formats 4-18
4.12 Run Time and Output Controls 4-19

5 1D Domains 5-1
5.1 Introduction 5-4
5.2 Schematisation 5-5
5.3 Solution Scheme 5-6
5.4 1d_nwk Attributes (All Nodes and Channels) 5-7
5.5 Channels Overview 5-8
5.6 Open Channels 5-13
5.6.1 Inertial Channels 5-13
5.6.2 Non-Inertial Channels 5-13
5.7 Structures 5-17
5.7.1 Culverts and Pipes 5-17

5.7.2 Bridges 5-24
5.7.2.1 Bridges Overview 5-24
5.7.2.2 Bridge Cross-Section and Loss Tables 5-24
5.7.2.3 B Bridge Losses Approach 5-25
5.7.2.4 BB Bridge Losses Approach 5-26
5.7.3 Weirs 5-33
5.7.3.1 Weirs Overview 5-33
5.7.3.2 Original Weirs (W) 5-34
5.7.3.3 Advanced Weirs (WB, WC, WD, WO, WR, WT, WV, WW) 5-36
5.7.3.4 Advanced Weir Submergence Curves 5-39
5.7.3.5 Automatically Created Weirs 5-46
5.7.3.6 VW Channels (Variable Geometry Weir) 5-2
5.7.4 Spillways (SP) 5-4
5.7.5 Sluice Gates (SG) 5-4
5.7.6 Adjustment of Contraction and Expansion Losses 5-5
5.8 Special Channels 5-8
5.8.1 M Channels (User Defined Flow Matrix) 5-8
5.8.2 Q Channels (Upstream Depth-Discharge Relationship) 5-9
5.8.3 X Connectors 5-10
5.8.4 Legacy Channels 5-10
5.8.5 1d_nwk Attributes (M, P, Q, SG, SP Channels) 5-11
5.9 Operational Channels 5-16
5.9.1 .toc File Commands and Logic 5-16
5.9.1.1 Define Control Command 5-16
5.9.1.2 User Defined Variables 5-17
5.9.1.3 Logic Rules 5-19
5.9.2 Types of Operational Structures 5-22
5.9.2.1 Pumps (P and PO) 5-22
5.9.2.2 QO Channels 5-23
5.9.2.3 Gated Drowned Rectangular Culverts (RO) 5-23
5.9.2.4 Sluice Gates (SG and SGO) 5-24
5.9.2.5 Spillways with Gates (SPO) 5-27
5.9.2.6 Weirs (WBO, WCO, WDO, WOO, WRO, WTO) 5-29
5.10 Cross-Sections 5-30
5.10.1 Type “XZ” Optional Flags 5-34
5.10.1.1 Relative Resistances 5-34
5.10.2 Type “HW” Optional Flags 5-37
5.10.2.1 Flow Area (A) 5-37
5.10.2.2 Wetted Perimeter (P) 5-37

5.10.3 Parallel Channel Analysis 5-37
5.10.4 Effective Area versus Total Area 5-39
5.10.5 Mid Cross-Sections 5-39
5.10.6 End Cross-Sections 5-40
5.10.7 Interpolated Cross-Section Protocols 5-40
5.11 Nodes 5-42
5.11.1 Storage Calculated from Channel Widths 5-47
5.11.2 Storage above Structure Obverts 5-48
5.11.3 Storage Nodes (User Defined NA Tables) 5-48
5.11.4 Using Nodes to Define Channel Inverts 5-50
5.11.5 Automatically Connecting Nodes to 2D domains 5-50
5.12 Pipe Networks 5-51
5.12.1 Pipes 5-51
5.12.2 Pits and Pit Channels 5-51
5.12.3 Pit Inlet and Depth/Stage vs Discharge Databases 5-56
5.12.3.1 Road Crossfall Options 5-59
5.12.4 Connecting Pits and Nodes to 2D Domains 5-59
5.12.5 Manholes 5-61
5.12.5.1 Automatically Assigned Manholes 5-61
5.12.5.2 Manually Assigned Manholes (1d_mh Layer) 5-62
5.12.5.3 Digitising Culverts Connected to Manholes 5-64
5.12.5.4 Engelund Manhole Loss Approach 5-64
5.12.5.5 Fixed Manhole Loss Approach 5-67
5.12.5.6 Discussion on Approaches to Modelling Pipe Junction Losses 5-67
5.13 Boundaries and 1D / 2D Links 5-69
5.14 Presenting 1D Domains in 2D Output 5-70

6 2D Model Domains 6-1
6.1 Introduction 6-3
6.2 Schematisation 6-4
6.3 Solution Scheme 6-6
6.3.1 2D Upstream Controlled Flow (Weirs and Supercritical Flow) 6-7
6.4 Boundaries and 1D and 2D/2D Links 6-9
6.5 2D Domain Extent and Resolution 6-10
6.6 Layering Datasets 6-11
6.7 Active / Inactive Areas 6-14
6.8 Elevations 6-16
6.8.1 Direct Reading of DEM Grids 6-16

6.8.2 Zpt Layers (2d_zpt) 6-17
6.8.3 3D Breakline Layers (2d_zln) 6-19
6.8.4 3D TIN Layers (2d_ztin) 6-19
6.8.5 Z Shape Layers (2d_zsh) 6-25
6.8.6 Variable Z Shape Layer (2d_vzsh) 6-34
6.8.7 Using Multiple Layers and Points Layers 6-39
6.8.7.1 Point Only Layers 6-39
6.9 Land Use (Materials) 6-41
6.9.1 Bed Resistance 6-41
6.9.2 Log Law Depth Varying Bed Resistance 6-43
6.9.3 Materials File 6-45
6.9.3.1 .tmf Format 6-45
6.9.3.2 .csv Format (Manning's n vs Depth Curves) 6-48
6.9.4 Rainfall Losses 6-53
6.10 Soil Infiltration 6-54
6.10.1 Green-Ampt 6-54
6.10.2 Horton 6-57
6.10.3 Initial Loss/Continuing Loss (ILCL) 6-57
6.10.4 Soils File (.tsoilf) 6-58
6.10.5 Groundwater 6-61
6.11 Cell Modification 6-62
6.11.1 Storage Reduction (2d_srf) 6-62
6.11.2 Cell Width Factor (CWF) 6-62
6.11.3 Form Loss Coefficient (FLC) 6-62
6.12 2D Hydraulic Structures 6-64
6.12.1 Introduction 6-64
6.12.2 2D Flow Constrictions (2d_fcsh and 2d_fc Layers) 6-68
6.12.2.1 Applying FC Attributes 6-74
6.12.2.2 Layered Flow Constrictions (2d_lfcsh Layers) 6-77
6.13 Modelling Urban Areas 6-83
6.13.1 Buildings 6-83
6.13.2 Roads 6-84
6.13.3 Fences and Walls 6-85

7 Boundaries and Initial Conditions 7-1
7.1 Introduction 7-2
7.2 Recommended BC Arrangements 7-3
7.3 1D Boundaries (1d_bc Layers) 7-5
7.4 2D Boundaries (2d_bc, 2d_sa and 2d_rf Layers) 7-9

7.4.1 2d_bc Layers 7-16
7.4.2 2d_sa Layers 7-20
7.4.2.1 Approaches to distributing flow 7-20
7.4.2.2 Streamlines 7-20
7.4.2.3 RF Option 7-22
7.4.2.4 Trigger Option 7-22
7.4.2.5 Flow Feature 7-23
7.4.3 Rainfall 7-24
7.4.3.1 Rainfall Overview 7-24
7.4.3.2 2d_rf Layers 7-24
7.4.3.3 Gridded Rainfall 7-25
7.4.3.4 Rainfall Control File (.trfc file) 7-27
7.5 Boundary Condition (BC) Database 7-28
7.5.1 BC Database Example 7-31
7.5.2 Using the BC Event Name Command 7-33
7.6 Initial Conditions 7-35
7.6.1 Initial Water Levels (IWL) 7-35
7.6.1.1 1D Domains 7-35
7.6.1.2 2D Domains 7-36
7.6.1.3 Automatic Initial Water Level (Set IWL == AUTO) 7-37
7.6.2 Restart Files 7-38

8 Linked 1D and 2D Models 8-1
8.1 Introduction 8-2
8.2 Linking 1D and 2D Domains (1D/2D) 8-3
8.2.1 Linking Mechanisms 8-5
8.2.1.1 HX 2D Head Boundary 8-5
8.2.1.2 SX 2D Flow Boundary 8-8
8.2.2 TUFLOW 1D (ESTRY) Domains 8-10
8.2.3 External 1D Solutions (Flood Modeller, XP-Solutions, 12D Solutions) 8-12
8.3 Linking TUFLOW 1D to External 1D Domains 8-15
8.3.1 Flood Modeller 1D/1D Link 8-15
8.4 Linking 2D Domains (2D / 2D) 8-16

9 Customising Output 9-1
9.1 Introduction 9-3
9.2 Output Control Commands 9-4
9.3 Configuring Plot Output Options 9-9
9.3.1 Reporting Locations (1D and 2D Combined) 9-9

9.3.2 Grouped Structure (1D and 2D) Output 9-11
9.3.3 2D Time-Series (Plot) Output (2d_po, 2d_lp) 9-12
9.4 Customising Map Output 9-18
9.4.1 Overview 9-18
9.4.2 Changing the Output for Different File Formats 9-18
9.4.3 Output Zones – Enhancing Map Output 9-19
9.4.4 Gauge Level Map Output (2d_glo) 9-22
9.5 Including 1D Results in Map Output 9-24
9.5.1 Overview 9-24
9.5.2 Water Level Lines (WLL, 1d_wll) 9-24
9.5.3 Water Level Line Points (WLLp) 9-26
9.5.4 Manually Adding Triangles into the 1d_WLL Layer 9-28
9.6 Map Output Formats 9-29
9.6.1 Overview 9-29
9.6.2 Which GIS/GUI Supports Which Formats? 9-29
9.6.3 Mesh Based Map Output Formats 9-30
9.6.3.1 DAT and XMDF 9-30
9.6.3.2 TMO 9-31
9.6.3.3 WRB 9-31
9.6.3.4 T3 9-31
9.6.4 Grid Based Map Output Formats 9-32
9.6.4.1 ASC, FLT 9-32
9.6.4.2 NC – NetCDF 9-33
9.6.4.3 TGO 9-33
9.6.4.4 WRR 9-34
9.6.5 Mesh and Grid Combined Map Output 9-34
9.6.5.1 WRC 9-34
9.6.6 GIS Based Map Output Formats 9-34
9.6.7 Mesh Configurations (.2dm File) 9-35
9.6.7.1 Quadrilateral and Triangle Mesh Option 9-35
9.6.7.2 Triangular Mesh Option 9-35
9.6.7.3 2D Cell Corner Interpolation/Extrapolation 9-35
9.7 Map Output Data Types 9-41
9.8 Specialised Outputs 9-56
9.8.1 Recording Gauge Data at Receptors (2d_obj, 2d_rec) 9-56
9.8.2 Evacuation Routes (2d_zshr) 9-58
9.8.3 Calibration Points GIS Layer 9-60

10 TUFLOW GPU Solver 10-1

10.1 Introduction 10-2
10.1.1 Will the Results be the same as TUFLOW Classic? 10-2
10.2 Solution Scheme 10-4
10.2.1 Conservativeness 10-4
10.2.2 Stability and Adaptive Timestepping 10-4
10.3 Getting Started 10-6
10.3.1 Compatible Graphics Cards 10-6
10.3.1.1 Compute Capability 10-9
10.3.2 Updating NVIDIA Drivers 10-9
10.4 Running TUFLOW GPU 10-10
10.4.1 Single and Double Precision 10-10
10.5 TUFLOW GPU .tcf Commands 10-11
10.6 Data Input 10-12
10.6.1 Cell Codes 10-12
10.6.2 Topography 10-12
10.6.3 Bed Resistance 10-12
10.6.4 Initial Water Levels 10-12
10.6.5 Boundary Conditions 10-12
10.6.6 Infiltration 10-13
10.6.7 Virtual Pipe Feature 10-13
10.6.8 Excluded TUFLOW Functionality 10-16
10.7 Data Output 10-17
10.7.1 Time-Series Output (2d_po) 10-17
10.8 Troubleshooting 10-18

11 Managing and Starting Simulations 11-1
11.1 Introduction 11-2
11.2 File Naming 11-3
11.3 Simulation Management 11-5
11.3.1 Events 11-5
11.3.2 Scenarios 11-9
11.3.3 Variables 11-12
11.4 TUFLOW Executable Download 11-14
11.4.1 Overview and Where to Install 11-14
11.4.2 Single and Double Precision 11-16
11.4.3 Using TUFLOW with Flood Modeller / XP-SWMM, or from SMS 11-17
11.4.4 Customising TUFLOW using TUFLOW_USER_DEFINED.dll 11-18
11.5 Running Simulations 11-19

11.5.1 Dongle Types and Setup 11-19
11.5.1.1 Protocols for Accessing Dongles 11-20
11.5.1.2 "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File 11-20
11.5.1.3 Dongle Failure during a Simulation 11-21
11.5.2 Starting a Simulation 11-21
11.5.2.1 Batch File Example and Run Options (Switches) 11-21
11.5.2.2 Advanced Batch Files 11-26
11.5.3 Windows Priority Levels 11-27

12 Check and Log Files 12-1
12.1 Introduction 12-2
12.2 Console (DOS) Window Display 12-3
12.2.1 The Console (DOS) Window Does Not Appear! 12-5
12.2.2 Console Window Shortcut Keys 12-6
12.2.3 Customisation of Console Window 12-7
12.3 Message Boxes 12-8
12.4 _TUFLOW Simulations.log Files 12-9
12.4.1 Local .log File 12-9
12.4.2 Global .log File 12-11
12.5 ERROR, WARNING and CHECK Messages 12-12
12.6 Simulation Log Files (.tlf and .tsf files) 12-13
12.7 1D Output File (.eof file) 12-14
12.8 GIS Workspaces (.wor and .qgs files) 12-16
12.9 Check Files and GIS Layers 12-17
12.10 Visualising and Querying Check Layers 12-22
12.11 Mass Balance Output 12-23

13 Viewing and Post-Processing Output 13-1
13.1 Introduction 13-2
13.2 Plot Output Files and Formats 13-3
13.2.1 Overview 13-3
13.2.2 .csv File Outputs 13-3
13.2.3 _TS GIS Layers 13-5
13.3 Plot Output Viewers and Tools 13-7
13.3.1 GIS Plot Viewers (TuPlot and miTools) 13-7
13.3.2 Other Plotting Viewers (eg. Excel) and Scripts 13-8
13.4 Map Output 13-9
13.4.1 Overview 13-9

13.4.2 GIS Layers (.MIF and .SHP) 13-9
13.4.2.1 Maximum and Minimum Output 13-9
13.4.2.2 _ccA GIS Layer 13-10
13.5 Map Output Viewers and Animators 13-11
13.6 Conversion to GIS Formats 13-12
13.7 Impact Analysis Mapping 13-13

14 Quality Control 14-1
14.1 Introduction 14-2
14.2 Model Health 14-3
14.2.1 Healthy Model Indicators 14-3
14.2.2 Timestep 14-6
14.3 Troubleshooting 14-7
14.3.1 General Comments 14-7
14.3.2 Identifying the Start of an Instability 14-7
14.3.2.1 Tips for 2D Domains 14-8
14.3.2.2 Tips for 1D Domains 14-9
14.3.2.3 Tips for 1D/2D Links 14-10
14.3.3 Suggestions and Recommendations 14-11
14.4 Large Models (exceeding RAM) 14-13
14.4.1 Influence of TUFLOW Version 14-13
14.4.2 Influence of Model Design 14-14
14.4.3 Memory Usage Reporting 14-16
14.4.4 Temporary Memory Usage 14-17
14.4.5 GPU models and RAM requirements 14-17
14.5 Past Release Version Backward Compatibility 14-18
14.6 Check List 14-19

15 Utilities 15-1
15.1 Introduction 15-2
15.2 Console Utilities 15-3
15.2.1 TUFLOW_to_GIS 15-3
15.2.2 ASC_to_ASC 15-4
15.2.3 RES_to_RES 15-5
15.2.4 Convert_to_TS1 15-5
15.2.5 TIN_to_TIN 15-6
15.2.6 xsGenerator 15-7
15.2.7 12da_to_from_gis 15-7
15.3 Excel Utilities 15-9

15.4 GIS Based Utilities 15-10

15.4.1 MiTools 15-10

15.4.2 TuPlot QGIS Plugin 15-10

15.4.3 ArcGIS Toolbox 15-11

16 New (and Past) Features and Changes 16-1

17 References A-1

Appendices

Appendix A .tcf File Commands A-3

Appendix B 1D (.ecf) Commands B-1

Appendix C .tgc File Commands C-1

Appendix D .tbc File Commands D-1

Appendix E .toc File Commands E-1

Appendix F .trfc File Commands F-1

Appendix G Hyperlinks to All Commands G-1

List of Figures

Figure 1-1	Schematisation of 1D, 2D Grid and 2D Flexible Mesh Models	1-2
Figure 2-1	TUFLOW Data Input and Output Structure	2-4
Figure 3-1	Impact of Cell Size on Model Results	3-4
Figure 3-2	Example of a Poor Representation of a Narrow Channel in a 2D Model	3-5
Figure 3-3	Effect of Enhanced Dry Boundary Viscosity Term Treatment	3-11
Figure 5-1	1D Inlet Control Culvert Flow Regimes	5-22
Figure 5-2	1D Outlet Control Culvert Flow Regimes	5-23
Figure 5-3	Bradley Weir Submergence Curve, Bradley 1978	5-35
Figure 5-4	Weir Submergence Curves from the Literature	5-41
Figure 5-5	Weir Submergence Curves using Villemonte Equation	5-41
Figure 5-6	‘All Parallel’ Conveyance Calculation Method	5-38
Figure 5-7	‘Change in Resistance’ Conveyance Calculation Method	5-38
Figure 5-8	Example of Pit Inlet Database	5-57
Figure 5-9	Road Crossfall Option	5-59
Figure 6-1	Location of Zpts and Computation Points	6-5
Figure 6-2	Example of Using a 2d_ztin or 2d_zsh Layer to Remove a Highway Embankment	6-24
Figure 6-3	Example of Log Law Variation of Manning’s n with Depth	6-44
Figure 6-4	Example of Log Law versus Constant Manning’s n with Depth	6-44
Figure 6-5	Example of Materials .csv File Format	6-52
Figure 6-6	Green-Ampt Model Concept	6-55
Figure 6-7	Example of Horton Infiltration Rate over Time	6-57
Figure 6-8	Example Soils .tsoilf File Format	6-59
Figure 6-9	Different Flow Patterns from 2D FCs and 1D/2D Links when Modelling a Submerged Culvert	6-67
Figure 6-10	Setting FC Parameters for a Bridge Structure	6-76
Figure 6-11	Setting FCSH Parameters for a Bridge Structure	6-77
Figure 7-1	Cell Cross-hair Selection Approach	7-9
Figure 7-2	Example Rainfall Database	7-26
Figure 7-3	Simple BC Database Example	7-32
Figure 7-4	Example BC Database Source Files	7-32
Figure 7-5	Example BC Database Using Event Text	7-34
Figure 7-6	Example BC Database Source Files Using Event Text	7-34
Figure 8-1	1D/2D Linking Mechanisms	8-4
Figure 8-2	Modelling a Channel in 1D and the Floodplain in 2D	8-5
Figure 8-3	Modelling a Pipe System in 1D underneath a 2D Domain	8-5

Figure 8-4 HX Schematic – Plan View 8-7
Figure 8-5 HX Schematic – Section View 8-7
Figure 8-6 SX Schematic – Cross Drainage Structure 8-9
Figure 8-7 SX Schematic – Pipe Network 8-9
Figure 8-8 Examples of 2D HX Links to 1D Nodes 8-11
Figure 8-9 Schematic of a Multiple Domain Model linked via a 1D Domain 8-16
Figure 8-10 Schematic of a Multiple 2D Domain Model using the 2d_bc “2D” Link 8-17
Figure 8-11 Multiple 2D Domain Model “2D” Link Check Files 8-19
Figure 9-1 Example of the QGIS TuPlot Plugin for a Reporting Location 9-10
Figure 9-2 Interpretation of PO Objects and SMS Output 9-17
Figure 9-3 Adding Triangles into 1d_WLL Layer to Infill Areas 9-28
Figure 9-4 Example Use of Gauge Data Output Layer 9-57
Figure 11-1 Simulation Priority via Windows Task Manager 11-28
Figure 12-1 Example Console (DOS) Display Window 12-3
Figure 12-2 Example TUFLOW Message Boxes 12-8
Figure 13-1 Example of the QGIS TuPlot Plugin Profile Plot 13-7
Figure 13-2 Example of the _ccA GIS Layer Showing Culvert Performance 13-10
Figure 14-1 Influence of 2D Domain Size on RAM Allocation 14-15

List of Tables

Table 1-1	Table Summary of United Kingdom Environment Agency Benchmark Tests	1-10
Table 2-1	Suggested Supporting Software	2-3
Table 2-2	Recommended Sub-Folder Structure	2-1
Table 2-3	List of Most Commonly Used File Types	2-3
Table 2-4	GIS Input Data Layers and Recommended Prefixes	2-8
Table 4-1	Reserved Characters – Text Files	4-4
Table 4-2	Notation Used in Command Documentation – Text Files	4-4
Table 4-3	Model Units	4-6
Table 4-4	TUFLOW Interpretation of GIS Objects	4-15
Table 5-1	1D Channel (Line) Types	5-9
Table 5-2	Open Channels: 1D Model Network (1d_nwk) Attribute Descriptions	5-14
Table 5-3	Culverts and Pipes: 1D Model Network (1d_nwk) Attribute Descriptions	5-18
Table 5-4	1D Culvert Flow Regimes	5-21
Table 5-5	Bridges: 1D Model Network (1d_nwk) Attribute Descriptions	5-28
Table 5-6	1D Bridge Geometry Table Link (1d_bg) Attribute Descriptions	5-31
Table 5-7	Weir Types	5-33
Table 5-8	Default Attribute Values for the Weir Equation for Different Weir Flows	5-37
Table 5-9	Weirs: 1D Model Network (1d_nwk) Attribute Descriptions	5-42
Table 5-10	1D Model Network (1d_nwke) OPTIONAL Attribute Descriptions	5-46
Table 5-11	1D Model Network (1d_nwk) Attribute Descriptions (Special Channels)	5-11
Table 5-12	Variable Value Types	5-17
Table 5-13	Channel Cross-Section Hydraulic Properties from External Sources	5-31
Table 5-14	1D Cross-Section Table Link (1d_xs) Attribute Descriptions	5-31
Table 5-15	1D Node and Pit Types (1d_nwk Point Objects)	5-43
Table 5-16	Nodes: 1D Model Network (1d_nwk) Attribute Descriptions	5-44
Table 5-17	1D Model Network (1d_nd) Attribute Descriptions for Nodes	5-46
Table 5-18	1D Nodal Area Table Link (1d_na) Attribute Descriptions	5-49
Table 5-19	Pits: 1D Model Network (1d_nwk) Attribute Descriptions	5-53
Table 5-20	Pit Inlet Database Format	5-56
Table 5-21	Q Pit Flow Regimes	5-58
Table 5-22	1D Manhole (1d_mh) Attribute Descriptions	5-62
Table 6-1	Location (2d_loc) Attribute Descriptions	6-10
Table 6-2	2D Zpt Commands	6-12
Table 6-3	Cell Codes	6-15
Table 6-4	2D Code (2d_code) Attribute Descriptions	6-15

Table 6-5 2D Z-point (2d_zpt) Attribute Descriptions (Read RowCol Zpts Command) 6-18
Table 6-6 2D Z-point (2d_zpt) Attribute Descriptions (Read GIS Zpts Command) 6-18
Table 6-7 2D Tin (2d_ztin) Attribute Descriptions 6-20
Table 6-8 2D Z-Shape (2d_zsh) Attribute Descriptions 6-31
Table 6-9 2D Variable Z-Shape (2d_vzsh) Attribute Descriptions 6-34
Table 6-10 2D Z (2d_z_) Attribute Descriptions 6-40
Table 6-11 2D Material (2d_mat) Attribute Description 6-42
Table 6-12 Materials .tmf File Format 6-45
Table 6-13 Materials .csv File Format 6-49
Table 6-14 USDA Soil types for the Green-Ampt Infiltration Method 6-56
Table 6-15 .tsoilf Parameters 6-58
Table 6-16 2D Soil (2d_soil) GIS Attribute Descriptions 6-59
Table 6-17 2D Groundwater (2d_gw) GIS Attribute Descriptions 6-61
Table 6-18 Hydraulic Structure Modelling Approaches 6-65
Table 6-19 Flow Constriction Shape (2d_fcsh) Attribute Descriptions 6-69
Table 6-20 Flow Constriction (2d_fc) Attribute Descriptions 6-72
Table 6-21 Layered Flow Constriction Shape (2d_lfcsh) Attribute Descriptions 6-79
Table 6-22 Layered Flow Constriction Shape Point (2d_lfcsh…_pts) Attribute Descriptions 6-82
Table 7-1 Recommended BC Arrangements 7-4
Table 7-2 1D Boundary Condition and Link Types 7-6
Table 7-3 1D Boundary Conditions (1d_bc) Attribute Descriptions 7-8
Table 7-4 2D Boundary Condition and Link Types 7-10
Table 7-5 2D Boundary Conditions (2d_bc) Attribute Descriptions 7-16
Table 7-6 2D Source over Area (2d_sa) Attribute Descriptions 7-21
Table 7-7 2D Direct Rainfall[1] over Area (2d_rf) Attribute Descriptions 7-25
Table 7-8 BC Database Keyword Descriptions 7-29
Table 7-10 1D Initial Water Level (1d_iwl) Attribute Descriptions 7-35
Table 7-9 2D Initial Water Level (2d_iwl) Attribute Descriptions 7-37
Table 9-1 Commands used to Control TUFLOW Output 9-4
Table 9-2 0d_RL Reporting Location Attributes 9-9
Table 9-3 Time-Series (PO) Data Types 9-13
Table 9-4 2D Plot Output (2d_po) Attribute Descriptions 9-16
Table 9-5 2D Longitudinal Profiles (2d_lp) Attribute Descriptions 9-16
Table 9-6 2D Gauge Level Output (2d_glo) Attribute Descriptions 9-23
Table 9-7 1D Water Level Line (1d_wll) Attribute Descriptions 9-25
Table 9-8 1D Water Level Line Point (1d_wllp) Attribute Descriptions 9-27
Table 9-9 Map Output Format Options 9-37

Table 9-10	Map Output Types (Excluding Hazard (Z) Types)	9-42
Table 9-11	Map Output Hazard (Z) Types	9-46
Table 9-12	2d_GDO_ Gauge Data Output Attributes	9-57
Table 9-13	2D Z-Shape Route (2d_zshr) Attribute Descriptions	9-58
Table 9-14	_RCP Output (2d_zshr) Attribute Descriptions	9-60
Table 10-1	TUFLOW GPU Boundary Types	10-12
Table 10-2	1d_pit (Virtual Pipe) Attributes	10-14
Table 11-1	TUFLOW Versions (iSP, iDP, w32, w64)	11-15
Table 11-2	TUFLOW.exe Options (Switches)	11-23
Table 12-1	Channel and Node Regime Flags (.eof File)	12-14
Table 12-2	Types of Check Files	12-17
Table 12-3	_MB.csv File Columns	12-25
Table 12-4	_MB1D.csv File Columns	12-26
Table 12-5	_MB2D.csv File Columns	12-28
Table 12-6	_TSMB GIS Layer Attributes	12-30
Table 12-7	_TSMB1d2d GIS Layer Attributes	12-30
Table 13-1	plot Folder File Descriptions (2016-03 Release)	13-4
Table 13-2	_TS GIS Layer Descriptions	13-6
Table 13-3	TUFLOW Map Output GIS and GUI Software Links	13-11
Table 14-1	Simulation Summary Healthy Model Indicators	14-4
Table 14-2	Quality Control Check List	14-19

GLOSSARY & NOTATION

ArcMap/ArcGIS ArcMap, also referred to as ArcGIS, is distributed by ESRI (http://www.esri.com). ArcMap is a GIS software package that can be used to develop the georeferenced input data for TUFLOW. TUFLOW writes result and check files which are compatible with ArcMap also.

attribute Data associated with 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 .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, included in mesh, but excluded from the calculation.
- 0 for a permanently dry cell
- 1 for a possibly wet cell
- 2 for an external boundary cell (assigned automatically)

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 A discrete feature in a 1D domain model, for example, a node or channel. The term "element" and "cell" are interchangeable for the 2D domain portions of the model.

fixed field Lines of text in a text file that are formatted to strict rules regarding which columns values are entered in to. This format was used for previous versions of ESTRY and TUFLOW and is no longer supported. If using an older version of TUFLOW which requires this format, please refer to previous manuals for the full documentation.

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

GIS Geographic Information System, for use in TUFLOW modelling it will need to be able to import/export files in .mif or .shp formats.

Read GIS The TUFLOW command "`Read GIS <data type> ==`" is used to input spatial data into TUFLOW. For example `Read GIS IWL == 2d_IWL.mif` Would be used to read into Initial Water Level (IWL) data.

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).

MapInfo MapInfo is distributed by Pitney Bowes (http://www.mapinfo.com/). MapInfo is a GIS software package that can be used to develop the georeferenced input data for TUFLOW. TUFLOW writes result and check files which are compatible with MapInfo also.

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.

QGIS QGIS is freeware software (www.qgis.org/). QGIS is a GIS software package that can be used to develop the georeferenced input data for TUFLOW. TUFLOW writes result and check files which are compatible with QGIS also.

Read 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". For example:

```
Read MI IWL == 2d_IWL.mif
```

Since the 2010-10 release, TUFLOW also recognises the .shp format and the notation "GIS" (`Read GIS IWL == 2d_IWL.mif`) is the preferred option, however "MI" (`Read MI IWL == 2d_IWL.mif`) may still be used for .mif format file references.

Read MID

See Read RowCol. 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" is the preferred option, however, the Read MID notation can still be used. The functionality is identical between Read RowCol and Read 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. The .mif and .mid files are a pair, and can't be opened in GIS unless both files are present.

.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 model 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. See Section 5.12.2 for more information on pits and pit channels.

pit channel

A small channel inserted at a pit typically used to convey water from overland 2D domains to 1D pipe networks. See Section 5.12.2 for more information on pits and pit channels.

pit inlet

The entrance to a pit channel (eg. a gully trap). See Section 0.

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 line segments that make up a polyline.

region A GIS object representing an enclosed area (ie. a polygon). A region has a centroid, perimeter and area. Polygons can have internal holes.

Read RowCol Used to read input where the input 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. For example:

```
Read RowCol Mat == <file.mid or file.dbf>
```

As the cell row and column are referenced in the data, the spatial data does not need to be read. This makes the file faster to read, however, if the cell size, orientation, origin or extent are changed the file needs to be re-created. For this reason the Read RowCol format is not frequently used.

.shp ESRI GIS layer file containing the geographic coordinates of objects. This is referred to as a Shapefile, however, a GIS dataset in Shapefile format will also contain a number of additional file (.dbf, .shx, .prj).

SMS Surface Water Modelling Software distributed by Aquaveo (formerly EMS-I) (www.aquaveo.com). SMS can be used as an interface for TUFLOW, allowing the user to view results and also to create a TUFLOW model within the SMS interface. This is commercially available software and a licence is required.

snap When geographic objects are connected exactly at a point or along a side. ArcMap, QGIS and MapInfo all have a "snap" feature, which ensures the features have the same coordinates. The snap tolerance can be changed in TUFLOW using the Snap Tolerance command.

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.

vertices Plural of vertex.

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

Chapter Contents

1 Introduction 1-1

1.1 Introduction 1-2

1.2 TUFLOW (Classic and GPU) 1-3

1.2.1 TUFLOW 2D Implicit Solver 1-3

1.2.2 TUFLOW 1D Solver (ESTRY) 1-3

1.2.3 TUFLOW 2D Explicit GPU Solver 1-4

1.2.4 TUFLOW Advection Dispersion and Heat Balance (AD) Module 1-5

1.2.5 Sediment Transport and Morphology (ST) Module 1-6

1.3 TUFLOW FV 1-7

1.4 Limitations and Recommendations 1-8

1.4.1 UK Benchmarking Study 1-9

1.5 Modelling Environment 1-11

1.1 Introduction

TUFLOW, or as sometimes referred to as TUFLOW Classic, is one of two products, TUFLOW and TUFLOW FV, for carrying out free-surface computational hydraulics of long waves such as floods and tides. TUFLOW is a grid based 2D hydrodynamic free-surface flow solver, while TUFLOW FV is a flexible mesh 2D and 3D solver. The engines are suited for simulating flood, tide, storm tide, tsunamis and coastal hydraulics. TUFLOW Classic incorporates world leading 1D/2D dynamic links to an advanced 1D scheme, with this functionality presently being incorporated into TUFLOW FV.

TUFLOW offers two 2D numerical schemes: a CPU based 2nd order implicit scheme, and the TUFLOW GPU module that value adds as a heavily parallelised 1st order rapid solver specifically designed for high end graphic cards (GPUs) that can reduce run-times by one to two orders of magnitude. The 1D and 1D/2D functionality of TUFLOW Classic is presently being built into TUFLOW GPU.

TUFLOW FV uses a flexible mesh of triangles and quadrilaterals, solving 2D and 3D forms of the equations with options for 1st and 2nd order solutions, and dynamic links to TUFLOW's 1D scheme. TUFLOW FV is briefly discussed in Section 1.3 and its documentation can be downloaded.

This manual covers TUFLOW and TUFLOW's GPU module.

For any enquiries on TUFLOW or TUFLOW FV, please email info@tuflow.com or sales@tuflow.com.

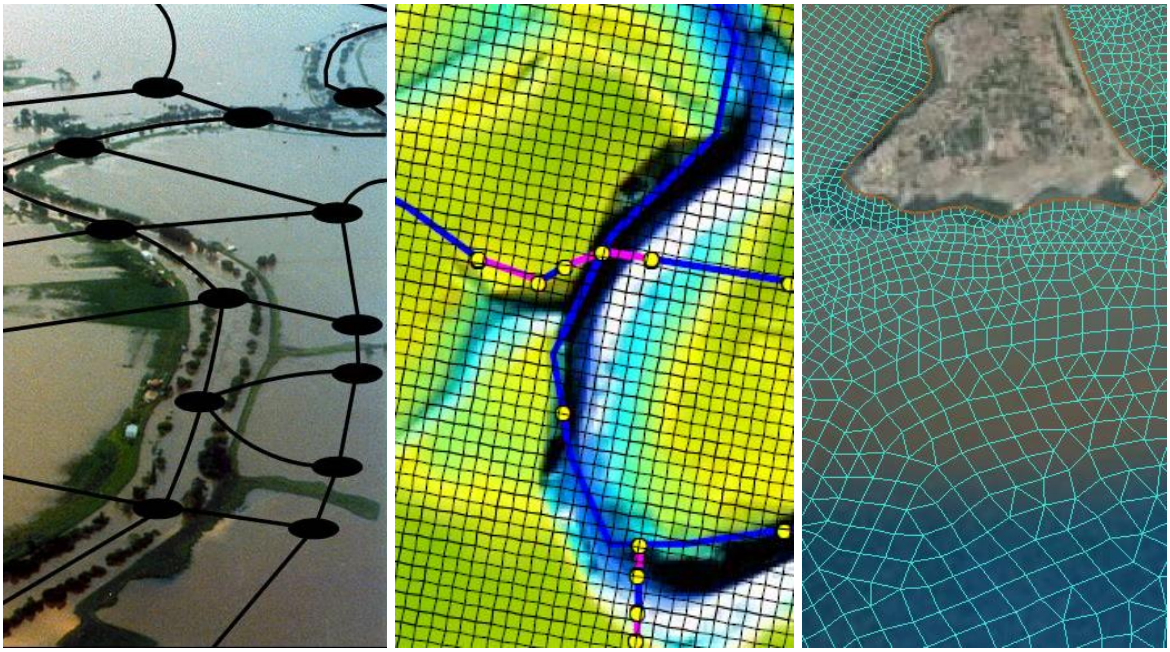


Figure 1-1 Schematisation of 1D, 2D Grid and 2D Flexible Mesh Models

1.2 TUFLOW (Classic and GPU)

TUFLOW is a computer program for simulating depth-averaged, one and two-dimensional free-surface flows such as occurs from floods and tides, with the 2D solution occurring over a regular grid of square elements. TUFLOW was originally developed for modelling two-dimensional (2D) flows, and stands for Two-dimensional Unsteady FLOW.

TUFLOW incorporates two grid based solvers; a CPU based second order semi-implicit solution often referred to as TUFLOW Classic, and a heavily parallelised first order explicit solver built for speed referred to as TUFLOW GPU.

TUFLOW also includes 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. In addition to the ESTRY 1D engine, TUFLOW has also been dynamically linked (fully integrated) with the following external 1D solvers: Flood Modeller (formerly known as ISIS); XP-SWMM 1D solution engines; and 12D Civil Engineering Solutions 1D drainage module.

1.2.1 TUFLOW 2D Implicit Solver

TUFLOW's implicit 2D solver 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 using a 2nd order semi-implicit matrix solver. 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 (now BMT WBM) 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 and 2D/2D 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; Néelz and Pender 2013).

TUFLOW is specifically orientated towards establishing flow and inundation patterns in floodplains, coastal waters, estuaries, rivers 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, Flood Modeller, XP-SWMM and 12D. The user sets up a model as a combination of 1D network domains linked to 2D domains. As such, the 2D and 1D domains are linked to form one overall model.

1.2.2 TUFLOW 1D Solver (ESTRY)

ESTRY is the primary 1D engine used by TUFLOW. ESTRY solves the full one-dimensional (1D) free-surface St Venant flow equations using a Runge-Kutta explicit solver, and is in its own right a powerful 1D network dynamic flow software suitable for modelling of floods and tides (and/or surges) in a virtually unlimited number of combinations.

The program was developed by BMT WBM Pty Ltd (previously WBM Oceanics Australia) over a period of forty years and has been successfully applied to a wide range of investigations from simple single channel applications to complex quasi-2D (branched or looped 1D) flood models and large underground 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.

In addition to the normal open channel flow situations, a wide range of additional channel types are available including:

- Circular, rectangular (box) and irregular culverts;
- Pit or manhole inlets;
- Bridges;
- Weir channels (including V-notch, ogee, crump broad crested and user defined);
- Spillway, radial and sluice gates;
- Pumps; and
- User defined structures.

All channel types can be specified as uni-directional, which allows flow in only one direction (upstream to downstream). The engine can handle both subcritical and supercritical flow regimes.

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.

The ESTRY 1D model is described in detail in Chapter 5.

1.2.3 TUFLOW 2D Explicit GPU Solver

TUFLOW's 2D explicit solver was first developed in 2011 to harness the power of heavily parallelised processing units found in Graphics Processing Units (GPU, also called a video or graphics card). TUFLOW's 2D implicit solver is not as suited to benefiting from parallelisation as explicit solvers are due to dependencies within numerical loops. The 2D Explicit GPU Solver is sold as a module of TUFLOW referred to as TUFLOW GPU.

TUFLOW GPU is a 2D fixed grid hydrodynamic solver that uses an explicit finite volume solution that is 1st order in space and 4th order in time (1st and 2nd order in time solutions are also available but are not recommended). TUFLOW GPU uses adaptive timestepping with the ability to revert back in time should a numerical inconsistency occur, thereby providing extreme numerical stability. The solution solves the full 2D free-surface equations including the inertia and sub-grid turbulence (eddy viscosity) terms.

The solver has been specifically developed to harness the heavily parallelised processing capabilities of GPU cards. Modern GPU cards have large numbers of processing cores (at the time of writing a single card may have in excess of 3,000 cores). By utilising multiple GPU cores, significant run time benefits can be achieved using explicit schemes. This benefit is most pronounced for very large models (in terms of number of cells).

TUFLOW GPU is currently limited to 2D only applications, and as such has not yet been dynamically linked to TUFLOW's 1D solver. However, the new Virtual Pipes feature allows the incorporation of inlet pits/gullies for modelling urban areas at a high resolution, and is a stepping stone to linking in the 1D solver, particularly for modelling full pipe networks. This feature is planned for release during 2016.

TUFLOW GPU is well-suited to broad-scale, first pass overland flow assessments, urban areas where high resolution detail is required to depict the urban topography and fine-scale modelling. In particular, the runtime benefits of this software are extremely powerful for broad-scale overland flow assessments and urban flooding that may have otherwise been too computationally intensive to model using a CPU type model.

TUFLOW GPU, whilst being a different numerical solver, uses the same model development framework as TUFLOW's 2D implicit solver, with all of the power and flexibility of TUFLOW's superior GIS functionality, scripting and scenario/event management available. However, due to efficiencies in maximising simulations speeds on GPU cards, the more simplistic solution scheme, and the young age of TUFLOW GPU, not all of the functionality of TUFLOW's features are available as yet. User information specific to TUFLOW GPU is provided in Chapter 10 of this document.

1.2.4 TUFLOW Advection Dispersion and Heat Balance (AD) Module

TUFLOW's AD (advection dispersion) module provides the capability to simulate constituent fate and transport in receiving waters. It is applicable to:

- Mixing in inland waterways;
- Fate of plumes;
- Flushing assessments;
- Advanced atmospheric heat exchange routines simulating thermal mixing and plumes;

TUFLOW's AD User Manual is a separate document that can be downloaded from www.tuflow.com.

1.2.5 Sediment Transport and Morphology (ST) Module

TUFLOW's original MOR (Morphology) module is currently being replaced with TUFLOW FV's more sophisticated sediment transport module algorithms. For further information please contact support@tuflow.com.

1.3 TUFLOW FV

TUFLOW FV is a 2D and 3D flexible mesh, finite volume numerical model that simulates hydrodynamic, sediment transport and water quality processes in oceans, coastal waters, estuaries, rivers and floodplains. The finite volume numerical scheme solves the conservative integral form of the non-linear shallow water equations (NLSWE). The equations can be solved in 2D (vertically averaged) and 3D. The key TUFLOW FV scheme features include:

- The ability to intrinsically handle shocks.
- Subcritical, supercritical and transitional flows.
- Local (and global) conservation of numerical precision.
- Robust wetting and drying.
- 1st and 2nd order spatial schemes.

TUFLOW FV is also dynamically linked with TUFLOW's 1D (ESTRY) solver's hydraulic structure routines, with full linkage planned for 2016.

The flexible mesh model structure allows users to modify mesh resolution spatially, seamlessly increasing the model resolution in areas of interest. This modelling approach reduces the number of computation cells in a model and hence reduces simulation times. Additionally, TUFLOW FV has been parallelised. This means users can fully capitalise on the computing power of multiple processor/thread computers. It is has been configured to run on Windows and Linux operating systems.

TUFLOW FV utilises different model development platforms compared to the more simplistic grid based approach of TUFLOW. Typically a third-party flexible mesh generator is utilised.

TUFLOW FV also has a range of modules, including:

- AD (advection dispersion) module that provides the capability to simulate constituent fate and transport in receiving waters. Is suited to investigations into salinity, temperature and sediment concentrations that require the 3D simulation of density driven currents. The AD scheme forms the core of subsequent sediment and water quality capabilities.
- ST (sediment transport) module that provides the ability to simulate cohesive and non-cohesive sediment transport, linked to hydrodynamic response via a morphological update routine to simulate evolution of bed features. Applications include: river, estuarine and coastal morphology; shoreline processes; and scouring and bank stability.
- Water quality modelling is available using TUFLOW FV in combination with the Aquatic Eco Dynamics (AED) model, developed by the University of Western Australia. This is currently a bespoke service, with a commercial version is currently being finalised.

TUFLOW FV is currently sold and documented as a separate product. The Scientific Reference and User Manual can be downloaded from www.tuflow.com.

For further information on TUFLOW FV please contact info@tuflow.com or sales@tuflow.com.

1.4 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:

- In areas of super-critical flow through the 1D and 2D domains, the results should be treated with caution, particularly if they are in key areas of interest. Hydraulic jumps and surcharging against obstructions can be complex 3D flow phenomena that are represented by 1D solutions as an occurrence between computational nodes and only approximately represented by 2D solutions.
- The Smagorinsky viscosity formulation is preferred over the constant viscosity formulation to model sub-cell turbulence (Barton, 2001). This is the default approach in TUFLOW. It is always good practice to carry out sensitivity tests to ascertain the importance of the viscosity coefficient(s) and formulation, which will be more influential where the bed friction is low (eg. in tidal reaches and coastal waters) and there are significant changes in velocity direction and magnitude causing sub-grid shear effects (eg. downstream of a constriction).
- 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 with water depths much greater than the cell size may start to violate the assumptions of the 2D equations. CFD (Computational Fluid Dynamics) codes that model turbulence and other terms more accurately may be needed in these situations. The influence of the viscosity (sub-grid scale turbulence) term can be particularly relevant. However, testing of models with very fine cell sizes (down to 0.1m) has indicated that reliable results can be obtained, particularly in urban areas where the flow depths are typically shallow.
- 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 3D flows that can occur through a structure, and regardless of the software used should be checked for their performance (Syme, 1998; Syme, 2001).
- 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. TUFLOW also has a range of options for modelling large structures in the 2D solution scheme.

1.4.1 UK Benchmarking Study

The "Desktop Review of 2D Hydraulic Modelling Packages" report (Néelz and Pender 2009) released in the United Kingdom by the Environment Agency highlighted the rapidly growing number of hydraulic modelling packages available for flood inundation estimation. A vast amount of documentation exists on appropriate applications of each modelling package however little of it discusses the influence that the choice of modelling package may have. The conclusions of the report recommended a series of benchmarking test cases to provide guidance on choosing the appropriate modelling package for future applications.

In response to the recommendations, a series of 10 2D flood inundation modelling benchmarking tests were conducted in 2010 using a variety of modelling packages. Additional testing was undertaken in in 2012 and published in 2013 due to the availability of new modelling packages and the further development undertaken on existing modelling packages. 15 software development organisations tested a total of 19 modelling packages. The results are documented in the report, "Benchmarking the Latest Generation of 2D Hydraulic Modelling Packages" (Néelz, S. and Pender, G. 2013).

TUFLOW was submitted for the initial phase of testing in 2010 with all three 2D schemes, TUFLOW's implicit and explicit GPU solvers, and TUFLOW FV's explicit solvers undergoing the more recent phase of testing in 2012. The results demonstrated consistency between each of the three TUFLOW engines and with other fully dynamic schemes. All three TUFLOW engines were found to be suitable for the following applications:

- Prediction of inundation extent;
- Prediction of maximum depth;
- Prediction of maximum velocity;
- Prediction of temporal variation in inundation extent;
- Prediction of temporal variation in depth; and
- Prediction of temporal variation in velocity.

The 10 tests are outlined in Table 1-1 below. For further information refer to "Benchmarking the Latest Generation of 2D Hydraulic Modelling Packages" (Néelz, S. and Pender, G. 2013).

Table 1-1 Table Summary of United Kingdom Environment Agency Benchmark Tests

Test Number	Description	Purpose
1	Flooding a disconnected water body	Assess basic capability to simulate flooding of disconnected water bodies on floodplains or coastal areas.
2	Filling of floodplain depressions	Tests capability to predict inundation extent and final flood depth for low momentum flow over complex topographies.
3	Momentum conservation over a small (0.25m) obstruction	Tests capability to simulate flow at relatively low depths over an obstruction with an adverse slope.
4	Speed of flood propagation over an extended floodplain	Tests simulation of speed of propagation of flood wave and the prediction of velocities at the leading edge of the advancing flood.
5	Valley flooding	Tests simulation of major flood inundation at the valley scale.
6A and 6B	Dam break	Tests simulation of shocks and wake zones close to a failing dam.
7	River to floodplain linking	Evaluates capability to simulate flood volume transfer between rivers and floodplains using 1D to 2D model linking.
8A and 8B	Rainfall and sewer surcharge flood in urban areas	Tests capability to simulate shallow flows in urban areas with inputs from rainfall (8A) and sewer surcharge (8B).

1.5 Modelling Environment

TUFLOW does not have its own graphical user interface, but utilises GIS and other third-party GUI software for the creation, manipulation and viewing of data.

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;
- Efficiency in producing high quality GIS based mapping for reports, brochures, plans and displays;
- Flexibility in choice of GIS package;
- Seamless handover of model inputs and results to clients requiring data in GIS format; and
- Better data quality control.

Section 2.1 outlines a variety of GUI software packages that may be utilised for constructing and visualising TUFLOW models.

A range of graphical interfaces are also available which allow the user to prepare spatial inputs, model control parameters, start TUFLOW simulations and view outputs from a single interface. Available Graphical interface options are discussed more on the TUFLOW website.

Regardless of the GIS package or interface used, the TUFLOW control files, GIS layers and modelling concepts are the same as outlined in this manual. This manual only focuses on TUFLOW's integration with GIS. For information and documentation on third-party TUFLOW GUIs, refer to the Graphical User Interfaces currently available.

2 Getting Started

Chapter Contents

2 Getting Started 2-1

2.1 The TUFLOW Modelling Concept 2-2

2.2 Graphical User Interface (GUI) Options 2-5

2.3 Installing and Running TUFLOW 2-6

2.3.1 TUFLOW Downloads and Installation 2-6

2.3.2 USB Locks (Dongles) and Licencing 2-6

2.3.3 Performing Simulations 2-6

2.4 Licence Free Simulations 2-7

2.4.1 Tutorial Models 2-7

2.4.2 Demo Models 2-7

2.4.3 Free Mode 2-8

2.5 Tips and Tricks 2-9

2.6 Folders and File Types 2-1

2.6.1 Suggested Folder Structure 2-1

2.6.2 File Types 2-2

2.6.3 Naming Conventions 2-7

2.1 The TUFLOW Modelling Concept

TUFLOW offers a different, but more powerful and efficient approach to modelling compared with other hydrodynamic modelling software. TUFLOW scripts (control files) allow modellers to readily and easily setup, modify and run numerous simulations, whether it be different calibration events, a batch of design events or various what-if scenarios investigating flood mitigation options. Combined with the power of GIS, the TUFLOW concept offers substantial benefits in terms of efficient workflow, modifications to models, quality control and user satisfaction, especially on investigations involving the examination of a multitude of events and "what-if" scenarios.

For those modellers preferring to work predominantly within a Graphical User Interface, there are also several third party options as discussed later in this chapter. For experienced modellers, the ability to be able to essentially script up a model, and use GIS in combination with these GUIs, offers a highly proficient arrangement by utilising the optimal software for different modelling tasks.

The fundamental software necessary for building and viewing TUFLOW models are:

- A text editor.
- Spreadsheet software.
- GIS software that can import/export .mif files or .shp files.
- 3D surface modelling software for the creation and interrogation of a DTM, and for importing 3D surfaces of water levels, depths, hazard, etc. This functionality is available in most GIS packages, sometimes as an add-on.
- GIS and/or a GUI for viewing results.

The above combination of offers a very powerful, workflow efficient and economical system for 2D/1D hydraulic modelling, driven by the unparalleled range of features and functionality available in TUFLOW, and the efficient modelling workflow that is generated using TUFLOW scripts or control files.

Suggested software packages include (but are not limited to) those listed in Table 2-1. Figure 2-1 illustrates an overview of the data input and output workflow:

- The GIS system is used to set up, modify, thematically map and manage all geographic data.
- Time-series and other non-geographically located data is tabulated using spreadsheet software.
- The text editor is used to create and edit TUFLOW simulation control files. The control files list all the simulation commands and file path references to the above mentioned GIS and tabular datasets.
- TUFLOW outputs results in text editor, spreadsheet, GIS and a variety of industry standard mapping formats for GUIs.

Table 2-1 Suggested Supporting Software

Software Type	Suggested Software
Text Editor	UltraEdit / TUFLOW Wiki UltraEdit Tips
	Notepad++ / TUFLOW Wiki Notepad++ Tips
	Textpad / TUFLOW Wiki Textpad Tips
	Other: Any text editor can be used for creating TUFLOW control files, including the Microsoft Windows default, Notepad. However, the above listed editors are recommended. They allow for advanced options, such as colour highlighting of TUFLOW control files and launching TUFLOW simulations from the editor – see downloads on this TUFLOW page.
Spreadsheet Software	Microsoft Excel / TUFLOW Wiki Excel Tips
	Libre Office
GIS Platforms	ArcGIS with Spatial Analyst TUFLOW Wiki ArcGIS Tips
	MapInfo Professional with Vertical Mapper or equivalent TUFLOW Wiki MapInfo Tips
	QGIS with TuPlot (time-series and profile plotting) and Crayfish (flood mapping and animation) plugins. TUFLOW Wiki QGIS Tips TUFLOW Wiki Crayfish Guidance
GUI Platforms (Additional to or as an alternative to using GIS)	12D Model / TUFLOW Wiki 12D Model Tips
	Blue Kenue
	Ensight
	FEWS
	Flood Modeller
	SMS / TUFLOW Wiki SMS Tips
	WaterRIDE
	XP2D

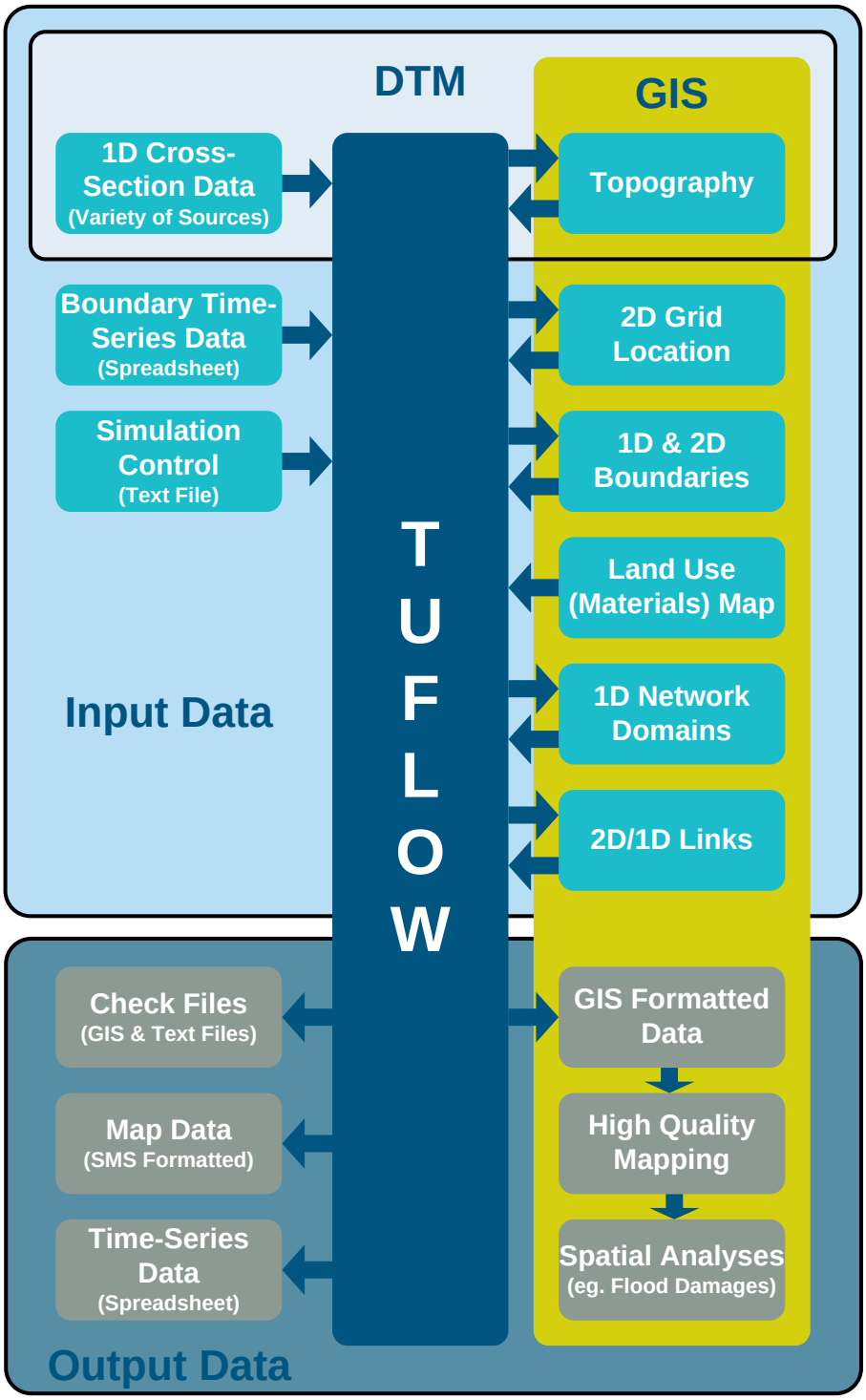


Figure 2-1 TUFLOW Data Input and Output Structure

2.2 Graphical User Interface (GUI) Options

The following GUI options are available if a complete Graphical User Interface (GUI) that allows the user to create, manage and view models and model output within the one interface is required, and/or to view and animate results. For general information and links visit TUFLOW Graphical User Interfaces.

- SMS TUFLOW Interface (available as of SMS Version 9.2 or later)
- XP-2D 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 TUFLOW 2D.
- Drainage 2D module available in 12D Model.
- Flood Modeller Pro (and the license free version Flood Modeller Free) available as part of the Flood Modeller Suite offers the ability to prepare GIS layers to be used in TUFLOW models. The 1D solution of Flood Modeller Pro has been dynamically linked to TUFLOW 2D.
- WaterRIDE for viewing, post-processing and querying TUFLOW results.

2.3 Installing and Running TUFLOW

2.3.1 TUFLOW Downloads and Installation

TUFLOW does not have an automatic installation, but instead requires the user to copy or unzip the downloaded files into a folder. Whilst this may seem "old-fashioned", this approach allows the modeller to have as many releases or versions of TUFLOW available as required, as there is often a need to run or re-run legacy models using older TUFLOW versions. All TUFLOW versions, manuals and release notes officially released are available from the All Downloads Page.

The TUFLOW software may even be located in a folder under the parent folder for the model, so that the version of TUFLOW used for that particular model is associated and archived with the model. It's also possible to force a model to only run on a particular release or build of TUFLOW (see Model TUFLOW Release or Model TUFLOW Build).

Section 11.4.1 provides more information on downloading and installing TUFLOW.

2.3.2 USB Locks (Dongles) and Licencing

A TUFLOW licence (eg. USB lock or dongle) is required to run TUFLOW, but is not required when using third party software such as a GIS, text editor or GUI (for further information refer to Section 11.5.1 and/or to the installation instructions on the TUFLOW Wiki). TUFLOW can be used licence free for the TUFLOW Tutorial Models and in Free or Demo Mode. In Free or Demo Mode there are limitations on the maximum number of 2D cells, 1D elements and simulation time.

For third-party USB locks that have TUFLOW licences please refer to the vendor's documentation for configuring the licence.

2.3.3 Performing Simulations

TUFLOW simulations are started by running the TUFLOW executable and passing the input TUFLOW control file. There are a number of ways of initiating simulations:

- Running a batch file. Batch files can be set up to loop through events and scenarios to run a multitude of simulations or to push simulations to different processors.
- Directly from the text editor – ideal for one off simulations especially whilst constructing a model.
- Directly from a GIS.
- Using Microsoft Explorer to right click and run.
- Via a GUI, such as the SMS TUFLOW Interface.
- From a Command (Console) Window.

Detailed descriptions on running TUFLOW from the above methods are provided in these instructions on the TUFLOW Wiki.

2.4 Licence Free Simulations

2.4.1 Tutorial Models

Tutorial models are available for download from www.tuflow.com. The purpose of the tutorial is to provide guidance on a number of the most commonly used features in the TUFLOW software. The tutorial documentation is provided within the TUFLOW Wiki. Documentation has been provided for developing models within ArcGIS, MapInfo, QGIS and for the SMS TUFLOW Interface.

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. An error will be generated if this command is included in any model other than the Tutorial model on the web.

The tutorial models are fully supported via the TUFLOW support services, therefore, should you have any queries please don't hesitate to contact support@tuflow.com.

2.4.2 Demo Models

A series of demonstration models were developed as part of three "challenges" issued prior to the 2012 Flood Managers Association (FMA) Conference in Sacramento, USA. The objective was to establish the variation in flood extents using different 2D software and modellers. Anyone could submit results, and the results were presented anonymously at the 2D Modelling Symposium held on the first day of the conference.

The three Challenge Models were:

- Challenge 1: An urban environment with a concrete lined main channel and numerous structures.
- Challenge 2: A coastal floodplain with two river entrances to the ocean during a flood.
- Challenge 3: A levied river within an alluvial fan in an arid, irrigated area.

The models developed using TUFLOW were created within a week and ran for various scenarios for all three challenges. The results of some 15 simulations were submitted and can be downloaded as pdf documents from www.tuflow.com. Models 1 and 3 were 1D/2D linked models, with Model 3 including some scenarios for soil infiltration using the Green-Ampt method. Challenge Model 2 was also simulated using TUFLOW's GPU solver and TUFLOW FV using a flexible mesh, providing an interesting benchmark between all three 2D solvers. Positive feedback was received on TUFLOW's hydraulic accuracy, and on the impressively short turnaround the TUFLOW models were created, simulated and results processed.

For those wishing to examine or run these models, they will be made available via the FMA Challenge models page on the TUFLOW Wiki. The models can be run without a license by specifying the command, Demo Model == ON, within the .tcf file.

The demo models are fully supported via the TUFLOW support services, therefore, should you have any queries please don't hesitate to contact support@tuflow.com.

2.4.3 Free Mode

As of the 2016-03 release, setting Demo Model == ON also allows users to setup and simulate models without a license so that users can trial or demo TUFLOW on small-scale models where:

- The total number of 2D cells does not exceed 100,000 – this is the number of cells in the bounding rectangle.
- The total number of active 2D cells does not exceed 30,000.
- The total number of 1D channels does not exceed 100.
- There is only one (1) 2D domain.
- Simulation time on the clock or CPU does not exceed 10 minutes.

Users operating in Free Mode can access TUFLOW support services under the same conditions as for a purchased licence (see the TUFLOW Licence Agreement), with the main provision being that TUFLOW support is not used as a substitution for TUFLOW training.

2.5 Tips and Tricks

Useful tips and tricks for a variety of third party software used in the development and visualisation of TUFLOW models have been collated on the TUFLOW Wiki. The guidance highlights how in-built features or user-developed functions can be used to enhance and streamline the modelling process. The Wiki is updated regularly. Please email any feedback to support@tuflow.com.

BMT WBM have also developed the MapInfo and TUFLOW Productivity Utilities (MiTools) to improve the efficiency of setting up and reviewing TUFLOW models when using MapInfo. The utilities enable 'automation' of many common repetitive tasks, saving valuable time and therefore money. The utilities also allow the efficient creation and visualisation of key TUFLOW model inputs/outputs within the MapInfo environment. Other tools provide data checking and quality assurance functionality, thus helping to minimise modelling errors. For further information or to arrange for a free 30 day evaluation license, please contact mitools@tuflow.com.

Utilities for ArcMap and QGIS are undergoing continuous development with the latest versions available for download at the following locations; ArcTUFLOW Toolbox and TUFLOW's QGIS TuPlot Plugin.

2.6 Folders and File Types

2.6.1 Suggested Folder Structure

Table 2-2 presents the recommended set of sub-folders to be set up for a 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.

Table 2-2 Recommended Sub-Folder Structure

Sub-Folder	Description
Locate folders below on the system network under a folder named "TUFLOW" 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 or model\gis folder (see below).
model\mi model\gis	GIS layers that are inputs to the 2D and 1D model domains. Also GIS workspaces. Model\mi\ - is typically used for MapInfo formatted GIS files model\gis\ - is typically used for other GIS formats (eg. shapefiles)
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" under the project folder eg. C:\Project12345\TUFLOW These files are reproducible and do not need to be as rigorously backed up (model output archiving is however still recommended at key project milestones. See also the Output Drive == Command	
results	The result files (use Output Folder).
check	GIS and other check files to carry out quality control checks (use Write Check Files).

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).** See also Section 4.3 for a discussion on absolute and relative file paths.

- Whilst TUFLOW readily accepts spaces and special characters (such as ! or #) in filenames and paths, other software may have issues with these. It is therefore recommended that spaces and other special characters are not used in the simulation path and filename without prior testing.
- Filenames and extensions are **not** case sensitive in any TUFLOW control files.

2.6.2 File Types

The most common file types and their extensions are listed in Table 2-3. These files are generally classified into the following categories:

- Control Files;
- Data Input Files;
- Data Output Files; and
- 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. As of the 2010 version of TUFLOW the original fixed field data input formats are no longer supported and an older version of TUFLOW will need to be used. If using fixed field formats you will need to use an older version of TUFLOW (pre 2010) and refer to older documentation.

Data Output Files contain the 2D and 1D hydraulic results of the simulation in the following formats:

- Mesh and results files for viewing the 2D and 1D domains in output formats supported by the following packages: SMS, WaterRIDE, Blue Kenue and 12D. A number of these software packages also permit animations of the results to be created.
- .csv (comma delimited) text output of time series data for direct input into spreadsheet software such as Microsoft Excel.
- GIS raster (gridded) format in .asc or .flt format for viewing results surfaces.
- GIS vector formats (.mif/.mid or .shp) for viewing 2D and 1D domain results.
- Text files that log the simulation.

In addition, several post-processing utilities are used for transferring data to GIS and other software (see Section 15).

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 model dataset which is used for the simulation calculations. The check files take the following forms:

- GIS formats (.mif/.mid or .shp) for viewing an echo of the model input data and also geographically locating any errors, warnings and checks;
- Text files for checking parameter and tabular inputs.

See also TUFLOW check files on the TUFLOW wiki.

Table 2-3 List of Most Commonly Used File Types

File	Extension	Description	Format
Control Files (Chapter 4)			
TUFLOW Simulation Control File	.tcf	Controls the data input and output for 2D or 2D/1D simulations. The filename (without extension) is used for naming all 2D domain files. Mandatory for all simulations.	Text
TUFLOW Boundary Conditions Control File	.tbc	Controls the 2D boundary condition data input. 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. 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 for a 1D or 2D/1D simulation.	Text
TUFLOW Operating Controls File	.toc	Contains operating rules that can be applied to hydraulic structures, pumps and other controllable devices modelled as 1D elements. Each set of operating rules is contained within a Control definition. More than one structure/device can use the same control definition.	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. The file extension can be anything; .trd, .erd and .rdf are most commonly used.	Text
Data Input (Chapters 5 to 8)			
ArcGIS Shapefile Layers	.shp .dbf .shx .prj	ArcGIS's industry standard for GIS layers. The .shp file contains information on the GIS objects coordinates. The .dbf file contains the attribute data information associated with the objects. Refer to the .tcf command GIS Format.	Binary

File	Extension	Description	Format
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. The .mid files are of similar format to .csv files, so they can be opened by Excel or other spreadsheet software. The files are text based and can be scripted by advanced users.	Text
TUFLOW Materials File	.tmf .csv	Sets the Manning's n values for different bed material categories in the 1D and 2D domains.	Text
TUFLOW Soils File	.tsoilf	Sets the infiltration method and infiltration parameters for different soil types in the 2D domains.	Text
Fixed Field Files	variety of extensions	New models do not require or support any fixed field input. However, older models (prior to the 2010-10 release) could utilise these formats. As the commands are no longer supported the fixed field documentation has been removed from this manual – see manuals prior to 2007 downloadable from www.tuflow.com.	Text
Data Output (Chapter 13)			
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 9-10 and Map Output Data Types for the different .dat file outputs.	Binary

File	Extension	Description	Format
SMS XMDF File	.xmdf	An alternative to using the .dat files described above. .xmdf files are much faster to access, are compressed by default for the 2016-03 release, and can contain all TUFLOW map output within a single file (rather than one file per output type as for the .dat format). See Table 9-10, Table 9-11 and Map Output Data Types for the different .xmdf file data sets available.	Binary
WaterRIDE	.wrb .wrc .wrr	WaterRIDE triangulated results file (.wrb) and/or grid based output (.wrr). The .wrc is a master file used when there are multiple file outputs.	
BlueKenue	.t3s .t3v	BlueKenue (National Research Council Canada) output format. The .t3s contains scalar data and the .t3v contains vector data.	
ESRI 2-5sci raster grid	.asc	Gridded data in the widely used ESRI 2-5sci format. This can be read in the majority of GIS platforms including ArcMap, QGIS and MapInfo.	
Binary Float Grid	.flt	Gridded data in the binary versions of the .asc format (see above). This data is recognised by most GIS packages and is much faster to read/write than the .asc format.	
12D Civil Solutions	.tmo	Format used by 12D for their TUFLOW interface.	
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 GIS.	Text
ArcGIS Shapefile Layers	.shp .dbf .shx .prj	Used for GIS based output of 1D and 2D time-series output within GIS. Refer to the .tcf command GIS Format.	Binary
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 (Chapter 12)			
TUFLOW Log File	.tlf	A log file containing information about the 2D/1D data input process and a log of the 2D simulation.	Text
TUFLOW Summary File	.tsf	A log file containing a concise summary of the simulation which can be regularly updated during the simulation.	Text
ESTRY Output File	.eof	Original ESTRY output file containing all 1D input data and results. This file is 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
ArcGIS Shapefile Layers	.shp .dbf .shx .prj	An alternative format to the MIF/MID files above. Refer to the .tcf command GIS Format.	Binary

2.6.3 Naming Conventions

As the bulk of the data input is via GIS data layers, efficient management of these datasets 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.

Different TUFLOW input files require different GIS attributes, for example an initial water level input file only requires a single attribute (with the attribute – initial water level), whereas, a 1D channel has a number of attributes (channel type, inverts etc.). Each of these file types is described in Table 2-4. **It is strongly recommended** that the prefixes described in Table 2-4 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. The .tcf command Write Empty GIS Files can be used to automate the creation of template files which use the recommended GIS data naming convention.

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 or roads. The sequential approach to reading datasets offers unlimited flexibility and increased efficiency. This layered approach also offers good traceability and quality control.

Refer also to Section 11.2 and 11.3 which provide further guidance on model naming conventions and methods in which the same control files may be used to simulate multiple flood events and scenarios, thereby reducing the potential number of files created.

Table 2-4 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 the 2d_bc prefix can be substituted with 2d_hx_ and 2d_sx_. Cell code values may also be defined in this layer.	0 8.2.1.1 8.2.1.2
2D Cell Codes	2d_code_	Optional GIS layers containing objects, typically polygons that define the cell codes.	6.7
2D Flow Constrictions	2d_fc_	Optional layers defining the adjustment of 2D cells to model structures such as bridges, box culverts, etc. Whilst this is still supported the newer and more flexible 2d_fcsh_ and 2d_lfcsh_ are preferred.	6.12.2
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.	9.4.4
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.	12.9
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.	7.6.1.2
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 defining the origin and orientation of 2D domains.	6.5
2D Longitudinal Profile Output Locations	2d_lp_	Optional layer(s) defining the locations longitudinal profile output from the 2D model domain	9.3.3
2D Land-Use (Materials) Categories	2d_mat_	Layers to define or change the land-use (material) types on a cell-by-cell basis.	6.9
2D Plot (Time-Series) Output Locations	2d_po_	Optional layer(s) defining the locations and types of time-series output from the 2D domains.	9.3.3

GIS Data Type	Suggested File Prefix	Description	Refer to Section
Direct Rainfall	2d_rf_	Optional layer(s) defining the polygons of sub-catchment areas for applying rainfall directly onto 2D domains or rainfall grids.	0
2D Source over Area	2d_sa_ 2d_sa_rf	Optional layer(s) defining the polygons of sub-catchment areas for applying a source (flow) or rainfall directly onto 2D domains.	0
2D Soil Infiltration	2d_soil_	Optional layer(s) that define the soil types on a cell-by-cell basis.	6.10
2D Elevations over an area	2d_za_	Optional layer(s) that define areas (polygons) of elevations at a constant height. This file is typically created by renaming the 2d_z__empty file.	6.8.2
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. These files are typically created by renaming the 2d_z__empty file.	0 6.8.5
2D Elevations as points	2d_zpt_	Layer(s) that define the elevations at the 2D cells mid-sides, corners and centres.	6.8.2
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).	6.8.4 6.8.5
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.	0
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.	6.12.2
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 vertical layers. Designed for modelling 2D flow under and over bridges, pipes and other obstructions across the waterway.	6.12.2.2

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).*	0
1D Bridge Losses	1d_bg_ 1d_lc_	Optional layer(s) that provide links to tabular data of loss versus height coefficients at a structure. The BG stands for **B**ridge **G**eometry and the alternative LC for **L**oss **C**oefficients. For medium to large models, create a folder called 'BG' under the 'Model' folder and place the 1d_bg_ tables in this folder along with all of the linked .csv files. These files are typically created by renaming the 1d_tab_empty file.	5.7.2
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.	07.6.1.1
1D Manholes	1d_mh_	Optional layer(s) defining the location and parameters of manholes.	5.12.5
1D Nodal Storage	1d_na_	Optional layer(s) that provide links to tabular data of storage surface area versus height at nodes. For medium to large models, create a folder called 'NA' under the 'Model' folder and place the 1d_na_ tables in this folder along with all of the linked .csv files.	5.11.3
1D Domain Network	1d_nwk_ 1d_nwke	Layer(s) that define the 1D or quasi-2D domain network of flow paths (channels), storage areas (nodes) and pit inlets (pits).	5.4 5.11
1D Water Level Lines for Mapping of 1D results into 2D formats	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.	9.5.2
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.	9.5.3

GIS Data Type	Suggested File Prefix	Description	Refer to Section
1D Tabular Input	1d_xs_ 1d_tab_	Optional layer(s) that provide links to tabular data of cross-sectional X-Z (chainage-elevation) values or H-W (height-width) values. For medium to large models, create a folder called 'XS' under the 'Model' folder and place the 1d_xs_ tables in this folder along with all of the linked .csv files.	5.10

3 Model Design

Chapter Contents

3 Model Design 3-1

3.1 Introduction 3-2

3.2 Model Schematisation 3-3

3.3 Model Resolution 3-4

3.3.1 2D Cell Size 3-4

3.3.2 1D Network Definition 3-5

3.4 Computational Timestep 3-6

3.4.1 2D Domains (Courant Number) 3-6

3.4.2 1D Domains (Courant Number) 3-6

3.4.3 1D/2D Models 3-7

3.4.4 Adaptive Timestep 3-7

3.5 Simulation Times 3-9

3.6 Eddy Viscosity 3-10

3.1 Introduction

This chapter of the Manual discusses key considerations that should be addressed during the design phase of a hydraulic modelling study, prior to the commencement of model build. Consideration of how the model is to be schematised and selection of key model parameters may have a significant influence on the accuracy, stability and simulation time of the model, and may avoid redundant work later on in the study.

3.2 Model Schematisation

General guidance on model schematisation and best practice procedures is freely available. These typically include discussion on data requirements, model conceptualisation, calibrating / sensitivity testing models, etc. Two such guidance documents are:

- Two Dimensional Modelling in Urban and Rural Floodplains Stage 1 & 2 Report (Babister, M. and Barton, C., 2012);
- Fluvial Design Guide (Environment Agency, 2010)

3.3 Model Resolution

3.3.1 2D Cell Size

The cell sizes of 2D domains need to be sufficiently small to reproduce the hydraulic behaviour, yet be large enough to minimise run times to meet project deadlines. At the start of a project, the modeller should determine the minimum cell size required to model the hydraulics accurately enough to meet the study objectives. Preferably at least three to four cells across the major flow paths is recommended.

Figure 3-1 below shows the different flow patterns through an urban area arising from the use of three different cell sizes. The larger 20m resolution is shown to be too coarse to depict the blockage caused by the buildings. The 10m resolution provides a better representation of the buildings however is not able to resolve the flow paths between the buildings. The 5m resolution in this example provides the best estimation of the flood behaviour.

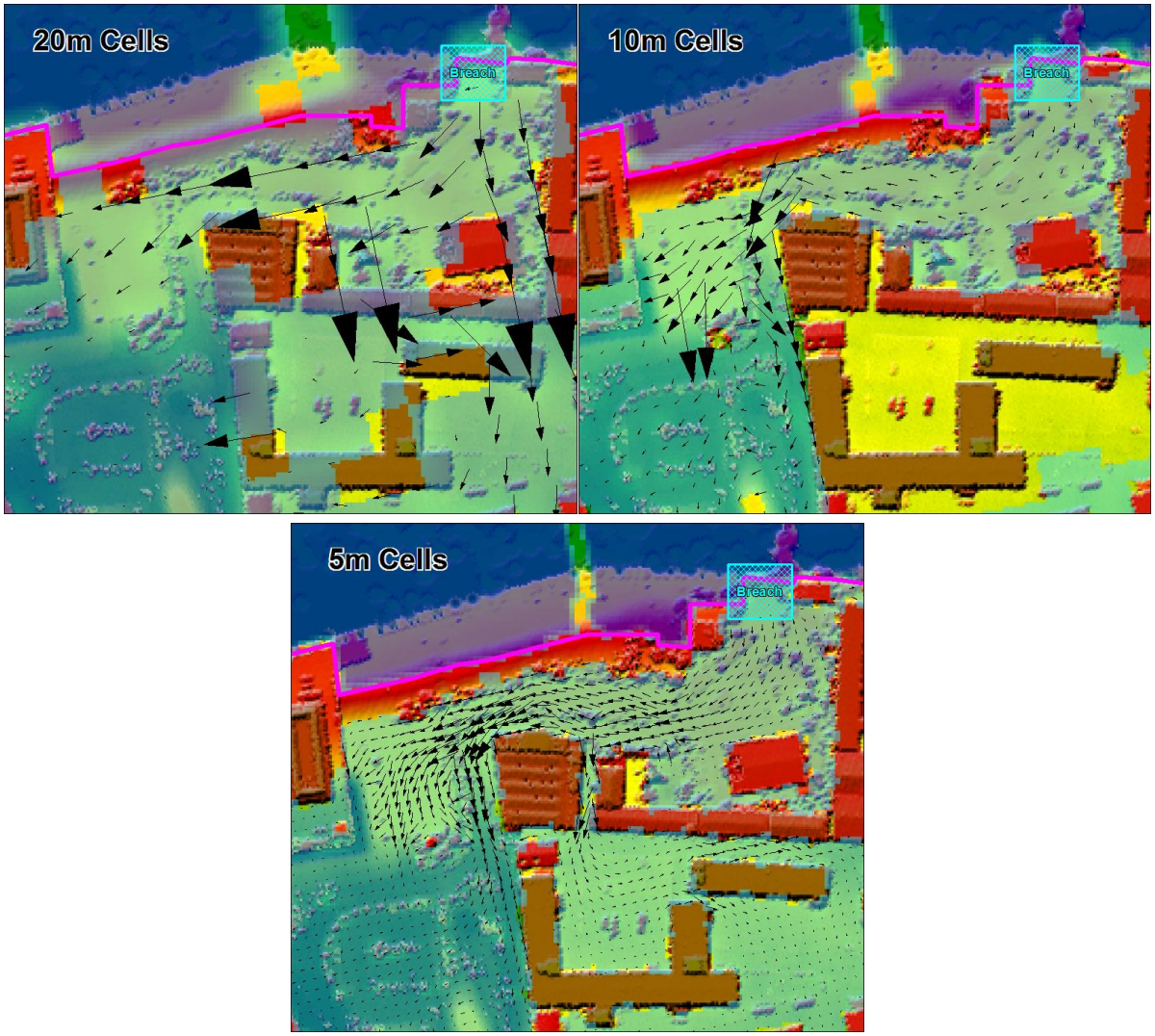


Figure 3-1 Impact of Cell Size on Model Results

(Tidal Thames Embayments Inundation Study, UK EA, Halcrow, TUFLOW, 2004)

Minor flow paths may be more coarsely or not represented if they play no significant role hydraulically in regard to meeting the modelling objectives. For example, minor drains across a floodplain may not affect peak flood levels; in which case, it may not be necessary to model them.

If it is not possible to model a major flow path with a sufficient cell resolution (see Figure 3-2) the flow path can be modelled as a 1D branch cut through the 2D domain (refer to Section 8.2). 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 or creek may be modelled in 1D and the floodplain in 2D.

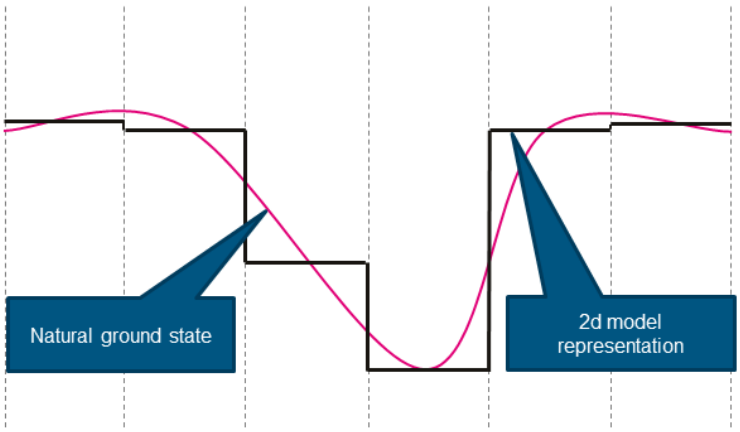


Figure 3-2 Example of a Poor Representation of a Narrow Channel in a 2D Model

3.3.2 1D Network Definition

The adequacy of the 1D domain is primarily dependent on the network representation that is adopted. Increased network detail will typically result in increased accuracy, though increased time and effort to establish the model. The end result may require a compromise between the level of detail and the manual effort required to create the model.

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.

3.4 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 simulation 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.

3.4.1 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.

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

Where:

$$\Delta t = timestep, s$$
$$\Delta x = length\ of\ model\ element, m$$
$$g = acceleration\ due\ to\ gravity, ms^{-\frac{1}{2}}$$
$$H = depth\ of\ water, m$$

As a general rule, the timestep (in seconds) is typically in the range of $^1/_2$ to $^1/_5$ of the cell size (in metres). For a 10m model the timestep will typical be in the range 2 – 5 seconds. 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 correct the instability. Refer to Section 14.3.2.

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 that the timestep is too high.

3.4.2 1D Domains (Courant Number)

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

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

The timestep 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 timestep. 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 timestep.

3.4.3 1D/2D Models

Different timesteps can be specified for the 1D and 2D domains, offering much greater flexibility in setting timesteps and model resolutions. Where the same timestep is used in both 1D and 2D domains, it is highly preferable that the 1D domains do not control the timestep, as typically 99% of the computational effort is usually in solving the 2D domains.

Specifying a 1D timestep in a linked 1D/2D model is optional. If a separate 1D timestep is not specified using the Timestep command, the smallest 2D timestep is used. 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. For example if the specified 2D timestep is 10 seconds and the 1D timestep is 3 seconds, the 1D timestep will be reduced to 2.5 seconds (this is reported in the simulation log file).

3.4.4 Adaptive Timestep

Adaptive timestepping using the .tcf command Maximum Courant Number has been introduced as part of the 2012-05 release. This is an **under-development** feature that at present is only available for 2D only models. Based on further successful testing, it may be extended to 1D/2D models. Cr_max would typically be in the range from 2 to 10, but higher values up to 60 or more have been successfully applied. Higher Cr-max values can be justified if consistent results are achieved using a smaller Cr_max or a fixed timestep. The Timestep command is only used to set the initial timestep if Maximum Courant Number is specified (Note: to avoid the simulation terminating early, specify an initial timestep that is less than the average adaptive timestep – this is required so that sufficient space can be allocated for storing time-series and other data written out at the end of the simulation). A Cr_max value of 0 or less switches adaptive timestepping off (which is the default). The maximum rate at which a timestep can increase is controlled by Timestep Maximum Increase. There is no limit to how quickly the timestep can decrease.

Note that adaptive timestepping may slow the simulation down if there are a few locations in the model that require a significantly reduce the timestep value below that normally used. Adaptive timestepping

can also be problematic for impact assessments if the timesteps in the before and after scenarios are sufficiently different to cause some unusual and unexpected impacts away from the area of interest!

3.5 Simulation Times

The simulation time of a model is dependent on a number of factors including:

- The area to be modelled;
- The cell size of the model;
- The number of cells that are wet;
- The duration of the simulation; and
- The computation engine (whether TUFLOW or TUFLOW GPU is used).

Each of the factors listed below should be considered during the development.

A simple spreadsheet has been created to estimate TUFLOW simulation times based on the cell size, catchment area and model run time (ie. event duration). The spreadsheet can be downloaded from the TUFLOW wiki. This spreadsheet is a guide as it has a number of assumptions including:

- The timestep;
- The number of calculations per second the computer can run; and
- The fraction of wet cells (this will change throughout the simulation);

However, it is still a useful guide. Reconsidering the model design is recommended if the spreadsheet is estimating excessively long runtimes.

The speed at which TUFLOW simulations will run is dependent on the hardware that you are running the simulation on. Newer and higher CPU frequency computers will run faster than older lower CPU frequency computers. A hardware benchmark model is available on the TUFLOW wiki, this page contains the runtimes for the same model on a range of different computers.

There are a number of other factors that can affect the performance of TUFLOW simulations such as:

- How frequently output is written, with more frequent output slowing the simulation;
- Whether the 32 bit or 64 bit version of TUFLOW is used - the 64bit version will be slightly faster.
- Whether the single or double precision version of TUFLOW is used - the single precision version will be faster. This is discussed in Section 11.4.2.

However, the primary influences will be the number of model cells, the duration of the simulation and the timestep.

As outlined in Chapter 10, the TUFLOW GPU module may be utilised to speed up the simulations for large 2D only models.

3.6 Eddy Viscosity

Two options exist for specifying eddy viscosity for the 2D domains to approximate the effect of fine-scale turbulence. The Viscosity Formulation and Viscosity Coefficient commands are used 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 m^2/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 the command Viscosity Coefficient.

The default Smagorinsky viscosity values are a coefficient value of 0.5 and constant value of 0.05m^2/s. The 0.5 Smagorinsky coefficient is dimensionless (the viscosity coefficient in m^2/s using the Smagorinsky approach is recalculated every timestep using the Smagorinsky formula and varies spatially and temporally). The viscosity coefficient can be output using the Map Output Data Types command.

The defaults were previously 0.2 and 0.1m^2/s. Testing on the UK EA 2D benchmark Test 6[1] models showed that the lower constant component produces improved results. For nearly all models the influence of this change is negligible or minor, as the lower constant component and higher Smagorinsky coefficient tend to cancel each other out. Further testing and research into flow patterns around fine-scale objects is planned.

Smagorinsky Formulation used by TUFLOW:

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

Where:

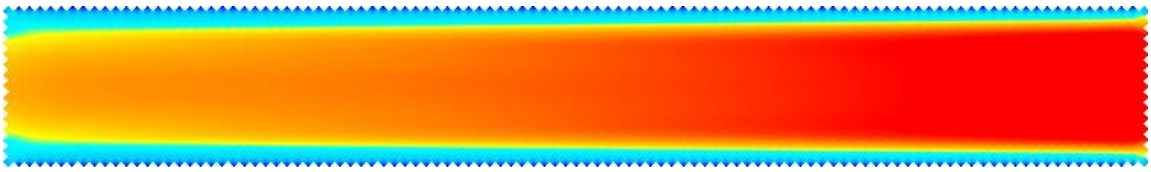
$u\ and\ v$ = Depth averaged velocity components in the X and Y directions
$x\ and\ y$ = Distance in the X and Y direction
μ = Horizontal diffusion of momentum (viscosity) coefficient
A_c = Area of Cell

[1] http://www.tuflow.com/Download/Publications/2012.06%20UK%20EA%202D%20Benchmarking%20Results.TUFLOW%20Products.pdf

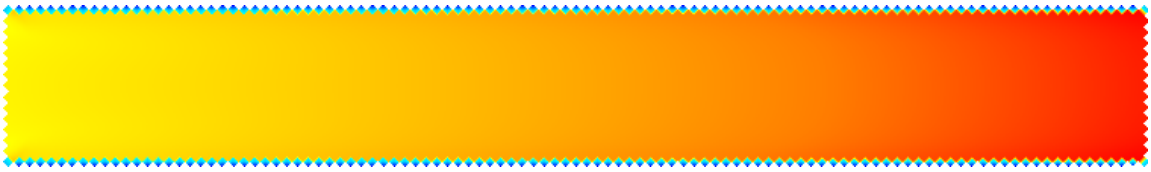
C_c = Constant Coefficient (default = 0.5)
C_s = Smagorinski Coefficient (default = 0.05)

The Smagorinsky approach features an enhanced treatment of the viscosity term at dry boundaries. The approach will produce the velocity contouring as shown in the bottom image of Figure 3-3. The top image shows slight flow separation along an oblique dry boundary that occurs without the enhanced treatment. With the enhanced treatment (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 cell-centred. This approach 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 without enhanced treatment of viscosity term



Correct velocity distribution using enhanced treatment of viscosity term at dry boundaries

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

4 Control Files and GIS Layers

Chapter Contents

4 Control Files and GIS Layers 4-1

4.1 Introduction 4-2

4.2 Control File Rules and Notation 4-3

4.3 Absolute and Relative File Paths 4-5

4.4 Units – Metric or US Customary/English) 4-6

4.5 TUFLOW Control File (.tcf file) 4-7

4.5.1 _TUFLOW_OVERRIDE Files 4-8

4.6 1D Commands 4-10

4.7 Geometry Control File (.tgc file) 4-11

4.8 Boundary Control File (.tbc file) 4-13

4.9 GIS Formats 4-14

4.9.1 “GIS” or “MI” Commands 4-14

4.9.2 “RowCol” or “MID” Commands 4-15

4.9.3 GIS Object Interpretation 4-15

4.10 XF Files 4-17

4.11 Fixed Field Formats 4-18

4.12 Run Time and Output Controls 4-19

4.1 Introduction

This chapter of the Manual discusses the TUFLOW model input files, which include the control files (text based input files) and GIS layers (.mif or .shp formatted files). The different control files will be discussed along with how different objects within the GIS layers are treated by TUFLOW. Further information on the GIS model input layers is discussed in Chapters 5 and 6.

4.2 Control File Rules and Notation

Control files, such as the tcf, .tbc, .tgc and .ecf 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.

Note: TUFLOW control files are NOT case sensitive.

An example of a command is:

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

This command 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.

Automatic colour coding of files for easy viewing is available for the following text editors: UltraEdit, Notepad++ and TextPad. Instructions outlining how these software can be configured can be found on the TUFLOW Wiki.

Commands 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 style of input is flexible bar a few rules. The rules are:

- A few characters are reserved for special purposes as described in Table 4-1;
- Command syntax is not case sensitive;
- Only one command can occur on a single line;
- A few commands rely on another command being previously specified. These are documented where appropriate.

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 also be acceptable, noting the use of the comment delimiter "!".

Blank lines are ignored. Spaces or indentations can occur at the start of the line. This is recommended when using the logic control, as outlined in Section 11.3. The second line in the following example is not required to be indented, however, it is recommended:

```
If Scenario == GPU
    GPU Solver == ON
End If
```

The notation used to document commands and valid parameter values in Appendix G are presented in Table 4-2.

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

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>`	A filename (can include an absolute or relative path, or a UNC path). See Section 4.3 for a more detailed description.
`[ {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 Maximums and Minimums command are: `[ ON | {ON MAXIMUMS ONLY} | OFF ]` Where the default is `ON MAXIMUMS ONLY` (which stores the maximums, but not the minimums).
`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, however, third party software may not allow this and as such is not recommended. If using spaces in filenames, batch files will require that the filename is enclosed in quotes.

4.3 Absolute and Relative File Paths

TUFLOW control files reference additional files, for example GIS files. The three methods that can be used are absolute file path, relative file path and UNC file path referencing. A model can use any or all of these methods. However, relative file paths are typically preferred. Take an example of reading a GIS layer containing initial water levels, the command is Read GIS IWL == <filepath>. The following are all valid:

Absolute file path: `Read GIS IWL == L:\Job\Job1234\TUFLOW\model\gis\2d_IWL_001_R.shp`

Relative file path: `Read GIS IWL == ..\model\gis\2d_IWL_001_R.shp`

UNC file path: `Read GIS IWL == \\server1\Job1234\TUFLOW\model\gis\2d_IWL_001_R.shp`

For the relative file path, the path is relative to the file that is referring to it. In the case above, if the command occurred in the .tcf file which is located in the TUFLOW\runs\ folder. The ..\ indicates to go up one level (from TUFLOW\runs\ to TUFLOW\) the model\ navigates into the TUFLOW\model\ folder, gis\ navigates from TUFLOW\model\ into TUFLOW\model\gis\). To go up more than one level simply use ..\ multiple times (eg. ..\..\ would navigate up two folders). If the file sits under the same folder then the filename can be specified. In the below example the Model_Events.tef would need to sit in the same location as the .tcf.

```
Event File == Model_Events.tef
```

The paths are relative to the current control file, a command in the geometry control file (TUFLOW\model\) would be relative to the .tgc location, not the .tcf location.

Using relative file paths has the advantage that the model can be moved or provided to another user and the control files do not need to be updated, if the model uses an absolute or UNC file path, all references will need to be updated if the model is moved.

Some large files, may still use an absolute or UNC file path particularly where large datasets are shared across models. This avoids the need to copy the dataset into multiple projects. An example is:

```
Read GIS IWL == \\server1\share\Lidar\East_Coast_5m.flt
```

If using LINUX at any point during the modelling tasks or pre/post processing, it is recommended to set Use Forward Slash == ON.

4.4 Units – Metric or US Customary/English)

The default unit settings for all inputs in a TUFLOW model are metric, however, it is possible to create models using US Customary units (also known as Imperial or English units) by specifying the .tcf command Units == US Customary (Units == Imperial or Units == English are also accepted are treated identically). The equivalent input and output units are listed in Table 4-3.

This manual uses the metric term for documentation purposes. If your model uses US Customary Units the metric term must be substituted for the equivalent US Customary unit as per Table 4-3 when reading this manual, unless otherwise stated.

Table 4-3 Model Units

Parameter	Metric Units	US Customary Units
Length	m	ft
Rainfall, initial loss and continuing loss	mm/hr	Inches/hr
Catchment area	km^2	$miles^2$
Constant eddy viscosity value	m^2/s	ft^2/s
Hazard categories (Z0 and ZUK0)	m and s	ft and s

If the Units == US Customary is specified, then any user defined values must also be specified in US Customary units. Model units can be checked by searching for the relevant parameter in the .tlf file (refer to Section 12.6).

4.5 TUFLOW Control File (.tcf file)

The **T**UFLOW **C**ontrol **F**ile (.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. Each TUFLOW simulation must have a .tcf file. An example of a simple .tcf file is shown further below.

The .tcf file must reference the following items as a minimum requirement:

- One .tgc file, using Geometry Control File for each 2D domain;
- One .tbc file, using BC Control File for each 2D domain;
- One .ecf file, using ESTRY Control File if there are any 1D domains (this file is optional – see below); and
- One materials file, using Read Materials File if material (land-use) polygons are being used. This file can be in either .tmf or .csv format.

Other mandatory or most commonly used .tcf commands are: BC Database, End Time, Map Output Data Types, Map Output Interval, MI Projection or SHP Projection, Maximums and Minimums, Output Folder, Start Time, Time Series Output Interval, Timestep, Write Check Files and Write Empty GIS 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 or SHP 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 .tcf commands are: BC Event Name; BC Event Text; Event File, Read File, Cell 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.

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

```
# 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.

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

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

Start Time (h) == 0.
End Time (h) == 12.
Timestep (s) == 5
```

```
Map Output Interval == 900 ! output interval in seconds
```

Note that it is possible to incorporate 1D (.ecf) commands into the .tcf file. 1D (.ecf) commands can be included in the .tcf file between Start 1D Domain and End 1D Domain block(s). This format is demonstrated in Module 2 of the TUFLOW tutorials which are available via the TUFLOW Wiki. For the new .tef file, 1D (.ecf) commands must be prefixed by "1D". Also note that ESTRY Control File is recognised every time if specified more than once (in previous releases only the last occurrence would apply).

4.5.1 _TUFLOW_OVERRIDE Files

These optional override files allow the user to apply .tcf commands after TUFLOW has finished processing the commands which are referenced within the .tcf file. This can be useful where you wish to change a parameter for all simulations that are started from a common folder. For example you could turn off check files, or change the output drive.

Two override file types are possible, one which is computer specific and one which is processed by all computers. If either or both are present, they are initialised after the .tcf file has finished processing its commands.

The first file that TUFLOW checks for is named "_TUFLOW_Override.tcf". If this file exists within the same folder as the simulation .tcf file, .tcf commands that are listed within the "_TUFLOW_Override.tcf" file overwrite the commands in the .tcf file.

The second file TUFLOW looks for must be named "_TUFLOW_Override_<computer_name>.tcf", where <computer_name> is the name of the computer running the simulation. If this file exists TUFLOW will process any commands within it after any commands from the "_TUFLOW_Override.tcf" file (if it exists). If you are unsure of your computer name, this is displayed in computer properties or in a TUFLOW Log File (.tlf) that has been processed on the computer, this is typically at the top of the .tlf (line 6).

```
Computer Name: COMP1234
```

With the computer name above the computer specific override file would be named "_TUFLOW_Override_COMP1234.tcf".

An override file specific to a particular computer can be particularly useful where, for example, different output drives or results folders, are to be used for runs using different computers. This will allow you to run TUFLOW simulations on different computers without having to change the .tcf file.

For example, if a run is started on one machine that only has a C drive, output can directed to the C drive just for that computer by using the command Output Drive == C.

Another example is if using the GPU solver and one machine only has a single GPU, while another has four GPUs. The command GPU Device IDs == 0, 1, 2, 3 can be specified in the override file specific to the machine with four GPUs.

Nearly all .tcf commands can be placed within the override files with the exception of Read GIS and some other similar commands that involve processing of data layers. It is recommended that the override files are only used for global changes to settings, similar to the examples above.

4.6 1D Commands

Commands specific to TUFLOW (ESTRY) 1D domains are detailed in 1D Commands, and can be placed either in the .tcf file (or in a Read File), or in their own file traditionally called an .ecf file. 1D commands can be located:

- Within an .ecf file and referenced within the .tcf file using the command ESTRY Control File. The command ESTRY Control File Auto can be used to force the .ecf file to have the same name as the .tcf file.
- Placed within a 1D domain block in the .tcf using Start 1D Domain and End 1D Domain commands.
- Placed anywhere in the .tcf file and preceded by "1D", for example, 1D Manhole Default Type. "1D" must be the first two characters.
- Any combination of the above, however, it is strongly recommended in the interests of manageability that a logical approach be adopted (that is in agreement with your colleagues!).

The .ecf file method is typically used if the 1D domain is large and complex, whereas the 1D domain block or preceding "1D" may be used where the 1D/2D linked model requires only a few 1D commands or they are placed in another file and referenced using Read File.

The 1D commands set simulation parameters and directs input from other data sources for all 1D domains. An example of some 1D commands are shown below.

Appendix B lists and describes the 1D commands and their parameters. Commands that are only relevant for 1D only models are indicated with a "1D Only" underneath the command in Appendix B.

Note: At present it is not an option to truly have a 1D only model. For 1D only models, a single 2D model, which can be made up of an inactive 2D cell, is still required.

```
# This is an example of a few 1D commands from an .ecf file
Timestep (s) == 3

Read GIS Network == ..\model\mi\1d_nwk_pipes.mif ! Read in the pipe network
Read GIS Manholes == ..\model\mi\1d_mh_manholes.mif ! Read in the manholes

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

Start Output (h) == 0.0
Output Interval (h) == 0.5
```

4.7 Geometry Control File (.tgc file)

2D domains are defined through a series of commands contained in **T**UFLOW **G**eometry **C**ontrol (.tgc) files. The .tgc file contains, or accesses from other files, information on the size and orientation of the grid, grid cell codes (whether cells are active or inactive), bed/ground elevations, bed material type or flow resistance value, and optional data such as soil type and 2D structure information.

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.

Rather than contain all the 2D grid information in one file, the .tgc file is a series of commands that **build** 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 dataset 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 Commands 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 that is read is 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.

Appendix C lists and describes .tgc commands and their parameters. 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  ! Domain origin and orientation
Cell Size == 10.  ! Set cell size to 10m

# Setup the domain code
Set Code == 0  ! Set everywhere inactive
Read GIS Code BC == ..\model\gis\2d_bc_my_model.mif   ! Sets active area using region

# Setup the base topography
Set Zpts == 100.  ! Set elevations everywhere to 100 m
Read Grid Zpts == grid\DEM_1m_All.flt  ! Sample elevations from DEM
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_landuse.mif   ! Read materials distribution
```

4.8 Boundary Control File (.tbc file)

The .tbc file or TUFLOW Boundary Conditions Control File (TBC) contains information regarding the location and type of boundaries within the model. These include (but are not limited to) upstream and downstream boundaries as well as 1D/2D or 2D/2D links. The GIS layers read into the .tbc file reference the Boundary Condition (BC) Database (see Section 7.5). The BC Database associates the GIS layers with a boundary condition such as a hydrograph or a depth-discharge curve.

Only one .tbc file per 2D domain is specified in the .tcf file using BC Control File**.**

A .tbc file must be specified, if no external boundaries are to be applied a blank file may be specified. This is rarely done, but can be used for models based only on initial water levels. Appendix C lists and describes .tbc commands and their parameters.

4.9 GIS Formats

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.

The format of input layers is solely controlled by the file extension (ie. .mif for the MIF format and .shp for the SHP format). TUFLOW requires that all GIS layers imported or exported by TUFLOW must be in the **same geographic projection.** The model projection is initialised using the MI Projection and/or SHP Projection commands. If a model has a mixture of .mif and .shp files as input, then both MI Projection and SHP Projection should be specified.

The default output format for GIS check layers and GIS outputs is the .mif format. To produce check and output GIS layers as .shp files, specify `GIS Format == SHP` in the .tcf file.

4.9.1 "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 and is now the preferred syntax). 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 format of the GIS layer is based on the filename's extension (.mif or .shp).

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.

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 affect. By default TUFLOW, checks that projection of each .mif file matches that specified by the MI Projection command and the projection of each shapefile matches that specified by the SHP Projection command. A MapInfo GIS layer must have a projection specified, whilst a Shapefile layer does not need to have a projection defined. Regardless of which GIS software used, it is strongly recommended that each file has a defined projection.

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 4-4 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.

4.9.2 "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 spatial data (.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.

The row and column numbering in the TUFLOW model will change if the model cell size, orientation or origin change. Whilst TUFLOW will continue to support the Read RowCol format, their use is only recommended for advanced users.

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 files may also be used. 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
```

4.9.3 GIS Object Interpretation

Table 4-4 outlines the compatible GIS objects and how they are interpreted by TUFLOW.

Table 4-4 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.

Object Type	TUFLOW Interpretation
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: 1D nodes within the region are selected. 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.

Object snapping is often used to relate point data with line and region data, for example with the Read GIS Z Shape command. TUFLOW supports point, end and vertex snapping. It does not support edge snapping.

4.10 XF Files

XF Files are a binary echo of selected input files automatically created by TUFLOW to vastly speed up the model initialisation process next time a simulation is carried out. If the original input data has a later save date later than its corresponding XF file, the original data is read and the XF file re-created. If the XF file is newer, it is used in preference to the original data.

XF files are always written to a folder named "xf" underneath 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. It is not possible to mix these (ie. to read an .xf4 single precision file into a double precision model).

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).

For commands that generate XF files (eg. Read RowCol Zpts), it is possible to refer directly to the .xf file instead of referring to the GIS layer, for example:

```
Read RowCol Zpts == ..\model\gis\2d_zpt_DTM.dbf.xf4
```

If different models utilise the same DEM(s), the DEM .xf file is reprocessed each time a different model is run. This can slow the model initialisation. To prevent this, the .tcf command XF Files Include in Filename adds unique text to the end of the .xf filenames for each model.

For example, if two models use the same DEMs for setting Zpt values, use this command as per below.

In Model 1's .tcf file:

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

In Model 2's .tcf file:

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

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

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

TUFLOW will add to the ~s1~ value to the end of any .xf filenames (refer to Section 11.3.3 on the automatic setting of scenarios and events as variables). Thus, when ~s1~ changes from one simulation to the next, the .xf filenames are unique to that scenario and will not need reprocessing.

4.11 Fixed Field Formats

Fixed field formats refers to lines in a text file that are formatted to strict rules regarding which columns values are entered in to. In early versions of ESTRY and TUFLOW (pre 1998) text input used fixed field format (reading GIS layers was first introduced in the late 1990s). Since the 2010 version of TUFLOW these fixed field formats are no longer supported. Should you require an older version of TUFLOW to run these models they can be downloaded from the previous releases section on the TUFLOW website. As these fixed field formats are no longer supported, the file formats are not documented in this version of the manual. Previous versions of the TUFLOW manual provide full documentation on fixed field input and can be downloaded from www.tuflow.com or by contacting support@tuflow.com.

4.12 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: Start Time, End Time and Timestep in the .tcf 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 3.4. For 2D/1D models, different timesteps may be used for both 2D and 1D schemes.

The output period 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. The output interval is defined by Map Output Interval and Time Series for the 2D domains, and Output Interval for the 1D domain.

The maximum and minimum hydraulic values can be determined using the Maximums and Minimums command for 2D domains (maximums and minimums are always output for 1D domains). **Maximum and minimum values are obtained by tracking the model results every calculation timestep, not the output interval.**

5 1D Domains

Chapter Contents

5 1D Domains 5-1

5.1 Introduction 5-4

5.2 Schematisation 5-5

5.3 Solution Scheme 5-6

5.4 1d_nwk Attributes (All Nodes and Channels) 5-7

5.5 Channels Overview 5-8

5.6 Open Channels 5-13

5.6.1 Inertial Channels 5-13

5.6.2 Non-Inertial Channels 5-13

5.7 Structures 5-17

5.7.1 Culverts and Pipes 5-17

5.7.2 Bridges 5-24

5.7.2.1 Bridges Overview 5-24

5.7.2.2 Bridge Cross-Section and Loss Tables 5-24

5.7.2.3 B Bridge Losses Approach 5-25

5.7.2.4 BB Bridge Losses Approach 5-26

5.7.3 Weirs 5-33

5.7.3.1 Weirs Overview 5-33

5.7.3.2 Original Weirs (W) 5-34

5.7.3.3 Advanced Weirs (WB, WC, WD, WO, WR, WT, WV, WW) 5-36

5.7.3.4 Advanced Weir Submergence Curves 5-39

5.7.3.5 Automatically Created Weirs 5-46

5.7.3.6 VW Channels (Variable Geometry Weir) 5-2

5.7.4 Spillways (SP) 5-4

5.7.5 Sluice Gates (SG) 5-4

5.7.6 Adjustment of Contraction and Expansion Losses 5-5

5.8 Special Channels 5-8

5.8.1 M Channels (User Defined Flow Matrix) 5-8

5.8.2 Q Channels (Upstream Depth-Discharge Relationship) 5-9

5.8.3 X Connectors 5-10

5.8.4 Legacy Channels 5-10

5.8.5 1d_nwk Attributes (M, P, Q, SG, SP Channels) 5-11

5.9 Operational Channels 5-16

5.9.1 .toc File Commands and Logic 5-16

5.9.1.1 Define Control Command 5-16

5.9.1.2 User Defined Variables 5-17
5.9.1.3 Logic Rules 5-19
5.9.2 Types of Operational Structures 5-22
5.9.2.1 Pumps (P and PO) 5-22
5.9.2.2 QO Channels 5-23
5.9.2.3 Gated Drowned Rectangular Culverts (RO) 5-23
5.9.2.4 Sluice Gates (SG and SGO) 5-24
5.9.2.5 Spillways with Gates (SPO) 5-27
5.9.2.6 Weirs (WBO, WCO, WDO, WOO, WRO, WTO) 5-29
5.10 Cross-Sections 5-30
5.10.1 Type "XZ" Optional Flags 5-34
5.10.1.1 Relative Resistances 5-34
5.10.2 Type "HW" Optional Flags 5-37
5.10.2.1 Flow Area (A) 5-37
5.10.2.2 Wetted Perimeter (P) 5-37
5.10.3 Parallel Channel Analysis 5-37
5.10.4 Effective Area versus Total Area 5-39
5.10.5 Mid Cross-Sections 5-39
5.10.6 End Cross-Sections 5-40
5.10.7 Interpolated Cross-Section Protocols 5-40
5.11 Nodes 5-42
5.11.1 Storage Calculated from Channel Widths 5-47
5.11.2 Storage above Structure Obverts 5-48
5.11.3 Storage Nodes (User Defined NA Tables) 5-48
5.11.4 Using Nodes to Define Channel Inverts 5-50
5.11.5 Automatically Connecting Nodes to 2D domains 5-50
5.12 Pipe Networks 5-51
5.12.1 Pipes 5-51
5.12.2 Pits and Pit Channels 5-51
5.12.3 Pit Inlet and Depth/Stage vs Discharge Databases 5-56
5.12.3.1 Road Crossfall Options 5-59
5.12.4 Connecting Pits and Nodes to 2D Domains 5-59
5.12.5 Manholes 5-61
5.12.5.1 Automatically Assigned Manholes 5-61
5.12.5.2 Manually Assigned Manholes (1d_mh Layer) 5-62
5.12.5.3 Digitising Culverts Connected to Manholes 5-64
5.12.5.4 Engelund Manhole Loss Approach 5-64
5.12.5.5 Fixed Manhole Loss Approach 5-67
5.12.5.6 Discussion on Approaches to Modelling Pipe Junction Losses 5-67

5.13 Boundaries and 1D / 2D Links 5-69

5.14 Presenting 1D Domains in 2D Output 5-70

5.1 Introduction

This chapter of the Manual discusses features specifically related to 1D model domains. 2D domain features are discussed separately in Chapter 6 and 1D/2D linking is discussed in Chapter 8.

5.2 Schematisation

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

- Channels represent the conveyance of the flow paths (see Section 5.4). The channels are flow and velocity computation points in the 1D model.
- Nodes represent the junctions of channels and the storage capacity of the network (see Section 5.11). These are water level computation points in the 1D model; and

Both channels and nodes are created using one or more 1d_nwk GIS layers. 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. **The digitising of nodes is optional.** In most models these are automatically created.

Multiple 1d_nwk GIS layers can be specified. Subsequent 1d_nwk layers are used to modify the network at individual objects. For example, if a culvert is to be upgraded in size, 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 in the 1d_nwk layer depend on the channel or node type. Table 5-1 and Table 5-15 present the available channel and node types. Section 5.4 presents links to all tables of attributes within the 1d_nwk layer for all channel and node types.

If using more than 100,000 channels see Maximum 1D Channels.

5.3 Solution Scheme

The scheme is based on a numerical solution of the 1D unsteady St Venant fluid flow equations (momentum and continuity) including the inertia terms. 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).

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

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

Where:

$u = depth\ and\ width\ averaged\ velocity$

$\zeta = water\ level$

$t = time$

$x = distance$

$A = cross\ sectional\ area$

$B = width\ of\ flow$

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

$n = Manning's\ n$

$f_l = Form\ (Energy)\ Loss\ coefficien\ t$

$R = Hydraulic\ Radius$

$g = accelerati\ on\ due\ to\ gravity$

5.4 1d_nwk Attributes (All Nodes and Channels)

In previous versions of the TUFLOW Manual one table covering the 1d_nwk attributes for all types of channels and nodes was provided. Rightly or wrongly (there was much debate on this one!), this table has been reworked as individual tables for each type of channel and node. Links to these tables are provided below. (If you would prefer one long table covering all types channels and nodes please let us know.)

Open Channels (S, G, Blank) 1d_nwk Attributes

Culverts and Pipes (C, I, R) 1d_nwk Attributes

Bridges (B, BB) 1d_nwk Attributes

Weirs (W, W*) 1d_nwk Attributes

Other Channels (M, P, Q, SG, SP) 1d_nwk Attributes

Nodes 1d_nwk Attributes

Pits 1d_nwk Attributes

1d_nwke (Extended 1d_nwk) Attributes

Note: The names of 1d_nwk attributes may have changed from previous TUFLOW releases to reflect new features and changes. The name of the attribute is irrelevant 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. Character, 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 correctly.

5.5 Channels Overview

1d_nwk channels can represent open channels, hydraulic structures such as bridges, culverts/pipes, weirs, and operational structures (eg. pumps and gates), and also other flow controls such as a user defined flow matrix.

A channel is digitised as a line or polyline. To connect channels the ends of the channels must be snapped. Channel flow direction is positive in the direction the line/polyline is digitised. This is best visualised in the GIS using a line style that has arrows or other symbolism indicating the line direction.

A channel is defined by a length, a Manning's n value, a table of hydraulic properties (wetted perimeter, flow area, hydraulic radius) versus elevation and other parameters depending on the type of channel. Table 5-1 below lists the available channel types as well as additional options that may be appended to selected channel types.

The hydraulic properties table for channels can be defined at a cross section positioned midway along the channel (for some structures and open channels) or can be derived from cross-sections located at the channel ends (for open channels only). The exceptions are:

- For culverts (C and R types) the attribute information supplied within a 1d_nwk layer (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 metres high based on the invert and width values.

Tables of cross-section profiles, cross-section hydraulic properties and bridge loss coefficients are accessed using links within 1d_xs and 1d_bg GIS layers. Tables can also be used to define nodal surface areas (refer to Section 5.11.3). This allows these data to be entered in a comma delimited format using .csv files that can be managed and edited in spreadsheet software such as Microsoft Excel.

Modellers often keep the different data sets separate as numerous .csv files are often needed. Separate folders underneath the model folder (same level as the mi or GIS folder) are often used to store all of the .csv files and the GIS layer.

- 1d_xs for XZ cross-section profiles in a model\xs folder
- 1d_bg for bridge loss coefficient tables in a model\bg folder

The Read GIS Table Links command is used for linking tabular data to channels. The method for linking cross-sectional and bridge losses is as follows:

- Lines or polylines are linked to channels. The method depends on whether the object has two or more vertices. The logic is:
    - For lines with two points (the start and end – no intermediate vertices) the line only needs to cross a channel– 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.

- Lines with three or more vertex must have one of the vertices snap to a vertex on the channel line. If both types are specified, the snapped sections are given preference over any two-point line that crosses the channel line.

- Other objects (regions and points) are not used.

The attributes and the method for determining which data to extract from the source file is outlined in

Table 5-14 for 1d_xs and

Table 5-6 for 1d_bg. Using the Column_1 attribute, several tables can be located in the one source file if desired.

Table 5-1 1D Channel (Line) Types

Channel/Node	Type	Description
Open Channels (Section 5.6)		
Open Channel	**S**	Open channel that incorporates all flow regimes. Similar to Normal (Blank) and Gradient (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 open channel type 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.
Structures (Section 5.7)		
Bridge Section 5.7.2.	**B**	Bridge structure – energy loss coefficients supplied by the user.
	BB	Bridge structure (introduced for Build 2016-03-AA) – only pier loss and submerged deck loss coefficients required (all other losses automatically calculated). BB bridges will also recognise bridge definition inputs to automatically generate loss coefficients (currently under development).
Culverts Section 5.7.1.	**C**	Pipe or Circular culvert.
	I	Irregular shaped culvert.
	R	Box or Rectangular culvert.
Gates Section 5.9.2.4	**RG**	Reserved for Radial Gates.
	SG	Sluice Gate.

Channel/Node	Type	Description
Pump Section 5.9.2.1	**P**	Pump.
Spillways Section 5.7.4 and 5.9.2.5	**SP**	Gated or ungated spillways.
Weirs Section 5.7.3	**W**	Weir structure (original weir channel).
	WB	Broad-crested weir.
	WC	Crump weir
	WD	User-defined weir.
	WO	Ogee-crested weir.
	WR	Rectangular weir (sharp-crested).
	WT	Trapezoidal / Cippoletti weir.
	WV	V-notch weir
	WW	Similar to the original W weir channel, but with more user options.
Special Channels (Section 5.8)		
Normal	(leave blank)	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.**
Gradient	**G**	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.
Matrix Flow Channel	**M**	User defined flow channel using a flow matrix. The flow through the structure is dependent on the water levels upstream and downstream.
Depth-Discharge Channel	**Q**	User-defined stage discharge channel. The flow through the structure is only dependent on the upstream conditions, such as user defined spillways. If downstream levels are influential then an M channel (see above) may be required.

Channel/Node	Type	Description
Connector	**X**	Connects the end of one channel to another. This is particularly useful for connecting a side tributary or pipe into the main flow path. 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. **Note: 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 Flags		
Adjust Structure Losses	**A**	Uses the equations and methodology in Section 5.7.6 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 5-4), unless it is a zero length channel (ie. channel length less than 0.01m).
Energy	**E**	For structures specifies the use of energy level for the flow calculations. The default is to use energy (E), unless the global .ecf command Structure Flow Levels == WATER is used, in which case the default is to use water level (H). See H below.
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 5.7.6. For example, to fix the losses for a circular culvert specify a Type attribute of "CF".
Water Surface	**H**	For structures specifies use of water level for the flow calculations. The default is to use energy level unless Structure Flow Levels == WATER has been specified. For example, if a broad-crested weir is to use water surface levels rather than energy levels, specify WBH (a space can be used so "WB H" may be preferred for clarity).
Non-inertial channel	**N**	Open Channel (S), Normal (blank) Gradient (G) and channels can be specified as non-inertial by including an "N" in the Type attribute. A non-inertial channel has the inertia term suppressed from the momentum equation.
Operational Control	**O**	"O" flag is required for structures that are to be operated using an operating control definition (see Section 5.8). For example, an operated pump would have a Type attribute of "PO" (or "OP")

Channel/Node	Type	Description
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 (from upstream to downstream). For example a "RU" channel could be used to represent a flap gated rectangular culvert.
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 5.7.3.6 for more details. **Note that prior to the 2013-12 release, a variable weir channel was specified as a WV channel type. As of the 2013-12 release, WV channels are processed as a V-notch weir. Variable weir channels must be specified as type "VW".**
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 5-10. Some of these attributes are used to specify the weir parameters.

5.6 Open Channels

5.6.1 Inertial Channels

An open channel that includes the inertia term is specified as a series of lines or polylines in one or more 1d_nwk GIS layers with an attribute type of 'S', S signifying a sloping channel that can handle steep, super-critical flows. S channels are typically all natural channels and artificial channels such as concrete lined open drains. They automatically test for the occurrence of upstream controlled flow and automatically switch between the two regimes, and is the preferred type for all open channels. Other open channel types, Normal (type '"blank") and Gradient (type "G") are kept for backward compatibility and discussed in Section 5.8.4. Table 5-2 lists the 1d_nwk attributes that are required for open channels.

The hydraulic properties table for open channels is typically provided in the form of cross-sectional data referenced within a 1d_xs GIS layer and using the command Read GIS Table Links but can also specified from external sources (see Section 5.10).

Lines or polylines within the 1d_xs GIS layer may be digitised midway along the channel or snapped to the channel ends. The treatment of the cross-sectional data is different depending on the digitisation. It is also possible to automatically create interpolated cross-sections. The attributes of the 1d_xs GIS layer is described in

Table 5-14 and further information on cross-sections is provided in Section 5.10.

5.6.2 Non-Inertial Channels

In order to bypass the Courant stability condition, a special channel flag (N) is included, known as non-inertial channel or a friction-controlled channel. This is valid for the S (open channel) and also the superseded blank and G gradient channels. To apply this to an S type channel the channel type is SN. For a non-inertial channel, the inertial terms are ignored (eliminating inertial effects) and the stability control procedure is automatically applied. Although rarely required, the suppression of the inertia terms can be useful for stabilising very short S channels with high velocities.

Table 5-2 Open Channels: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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.	Char(12)
2	Type	As described in Table 5-1. **S:** Steep Channel **G:** Gradient Channel **Blank:** Normal Channel	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes).	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 the upstream and downstream nodes connected to the channel. If set to No ("N" or "n") or False ("F" or "f"), the channel width and length does not contribute to the node's storage. See Section 5.11.1 for further discussion.	Char(1)
5	Len_or_ANA	If greater than zero, sets the length of the channel in metres. 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 Len_or_ANA is -99999, the length from the MIKE 11 link channel is used.	Float
6	n_nF_Cd	The Manning's n or Manning's n multiplier for the channel. If not using materials or Manning's values in the cross-section the Manning's n values is specified using this attribute. If using materials or Manning's n to define the bed resistance from XZ tables (see Sections 5.10.1.1.2 and 5.10.1.1.3), n_nF_Cd is a multiplier and is typically 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.	Float
7	US_Invert	**G, S Channel Type:** The upstream bed or invert elevation of the channel in metres. The bed of	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		the channel cross-section is used if -99999 is specified for G and S channels. If a manually created node exists at the upstream end of the channel, and -99999 is specified, the upstream invert is set to the DS_Invert value of the node (or pit), provided this value is greater than -99999 – for example, see Table 5-16. **Blank Channel Type:** Sets the upstream and downstream 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. **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 most GIS platforms including MapInfo for the invert attributes.**	
8	DS_Invert	Sets the downstream invert of the channel using the same rules as described for the US_Invert attribute above.	Float
9	Form_Loss	Additional form losses (factor of dynamic head) due to bends, bridge piers, etc. This method is preferred instead of increasing Manning's n to account for losses. For S channels, this only applies when not in upstream controlled friction mode.	Float
10	pBlockage	Not used.	Float
11	Inlet_Type	This attribute is used to manage MIKE 11 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 Inlet_Type (River name), Conn_1D_2D (Topo ID) and Conn_No (XSect ID or Chainage) attributes.	Char(256)
12	Conn_1D_2D	See description for Inlet_Type above for MIKE 11 cross-section data. Used to reference the Topo ID from the MIKE 11 cross-section data if desired. If Conn_1D_2D is "$LINK", searches the active MIKE 11 network (.nwk11) file for the link cross-section details.	Char(4)

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		This attribute is also used to manage Flood Modeller cross-section data. If a Flood Modeller database (.pro file), finds the processed data based on the label specified in the Conn_1D_2D attribute.	
13	Conn_No	See description for Inlet_Type and Conn_1D_2D above for MIKE 11 cross-section data. If being used for a MIKE 11 cross-section, this attribute should match the Chainage specified to the nearest integer. For links, Conn_No must equal or fall within the upstream and downstream chainages of the link.	Integer
14	Width_or_Dia	Not used.	Float
15	Height_or_WF	Not used.	Float
16	Number_of	Not used.	Integer
17	HConF_or_WC	Not used.	Float
18	WConF_or_WEx	Not used.	Float
19	EntryC_or_WSa	Not used.	Float
20	ExitC_or_WSb	Can be used to apply a form loss coefficient per unit length of channel for an S type channel. For example a value of 0.0001 v2/g / metre or (0.1 per km) for a 200m channel length the extra form loss would be 0.0001 x 200 or 0.02 v2/2g. This can be used to account for irregularities in the bed form not accounted for by Manning's n value. If using it is recommended that this is calibrated. This was introduced in the 2016-03 TUFLOW release, prior to this it was not used.	Float

5.7 Structures

Hydraulic structures in the 1D domain are modelled by replacing the momentum equation with standard equations describing the flow through the structure. The structures available are described in the following sections. A discussion on the choice of a 1D or 2D representation of the structure is presented in Section 6.12.1.

A channel is flagged as a hydraulic structure using the Type attribute as described in Table 5-1. Except for culverts, a structure has zero length, ie. there is no bed resistance. A length can be applied to weir channels, however, this is only used in the calculation of the storage (nodal area).

5.7.1 Culverts and Pipes

Culvert or pipe channels can be either rectangular, circular (pipe) or irregular 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 5-1, Figure 5-2 and Table 5-4 present the different flow regimes which can be modelled. The regimes which occur during a simulation are output to the .eof file next to the velocity and flow output values, and to the _TSF GIS layer (see Sections 12.7 and 13.2.3).

For all culvert types the length, upstream and downstream inverts, Manning's n, bend loss, entrance and exit losses and number of barrels are entered using the 1d_nwk attributes (see Table 5-3). For type "C" circular or type "R" rectangular culverts, the dimensions are also specified within the 1d_nwk attributes. For an "I" irregular shaped culvert, the cross-sectional shape is specified in the same manner as for open channels using a 1d_xs GIS layer (refer to Section 5.10 and

Table 5-14) and the command Read GIS Table Links. The line or polyline is digitised across the 1d_nwk channel line.

The four culvert 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 entry loss coefficient. The standard value for this coefficient is 0.5. Variations to this value may be applied based on manufacturer specifications.
- 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".

For benchmarking of culvert flow to the literature see "TUFLOW Validation and Testing (Huxley, 2004) which is available at www.tuflow.com.

Table 5-3 Culverts and Pipes: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. When automatically creating nodes (default) ".1" and ".2" are added to the channel names for the upstream and downstream node names respectively. IDs over 10 characters long are not recommended as the appending of .1 and .2 can cause duplicate node ID's to be created.	Char(12)
2	Type	The culvert type: • "C" for a circular culvert; • "R" for a rectangular culvert; • "I" for an irregular shaped culvert Optional type flags may also be used to implement additional flow controls such as unidirectional flow ("U") or operated flow conditions ("O"). Culvert type details are outlined in Table 5-1.	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes)	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 5.11 for further discussion.	Char(1)
5	Len_or_ANA	The length of the culvert in metres. If the length is less than zero, except for the special values below, the length of the line/polyline is used. If Length is -99999, the length from the MIKE 11 link channel is used.	Float
6	n_nF_Cd	The Manning's n value of the culvert.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		If using materials to define the bed resistance from XZ tables (only for Irregular culvert, see Section 5.10.1.1.2), n_nF_Cd 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.	
7	US_Invert	The upstream bed or invert elevation of the culvert in metres. If a culvert 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. Interpolate Culvert Inverts can also be used to switch this feature ON or OFF.	Float
8	DS_Invert	Sets the downstream invert of the culvert using the same rules as for described for the US_Invert attribute above.	Float
9	Form_Loss	Specifies 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.	Float
10	pBlockage	**C, R Channel Type:** The percentage blockage (for 10%, enter 10) of the culvert. 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. Note that when applying a % Blockage to C culverts, the invert level remains unchanged and only the soffit level is reduced by the calculated decrease in diameter. **I Channel Type:** Not used.	Float
11	Inlet_Type	Not used.	Char(256)
12	Conn_1D_2D	Not used.	Char(4)
13	Conn_No	Not used.	Integer
14	Width_or_Dia	**C Channel Type:** The pipe diameter in metres. **R Channel Type:** The box culvert width in metres. **I Channel Type:** Not used.	Float
15	Height_or_WF	**R Channel Type:** The box culvert height in metres.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		C, I, Channel Type: Not used.	
16	Number_of	The number of culvert barrels. If set to zero, one barrel is assumed.	Integer
17	HConF_or_WC	**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. **C Channel Type:** Not used.	Float
18	WConF_or_WEx	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.	Float
19	EntryC_or_WSa	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 5-15), this value is adjusted according to the approach and departure velocities (see Section 5.7.6). 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 5.12.5). The entry loss value used in the computation can be viewed over time using the _TSL output layer (see Section 13.2.3).	Float
20	ExitC_or_WSb	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 5-15), this value is adjusted according to the approach and departure velocities (see Section 5.7.6). 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 5.12.5). The entry loss value used in the computation can be viewed over time using the _TSL output layer (see Section 13.2.3).	Float

Table 5-4 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 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.

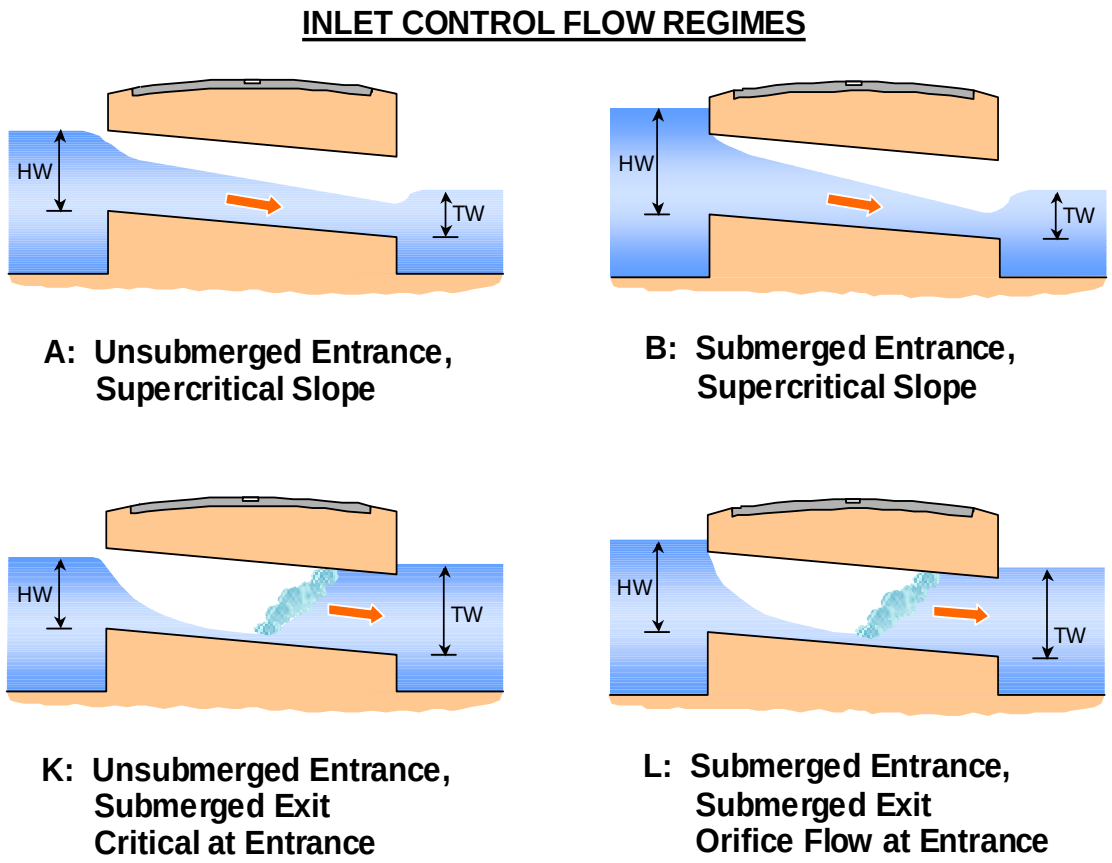


Figure 5-1 1D Inlet Control Culvert Flow Regimes

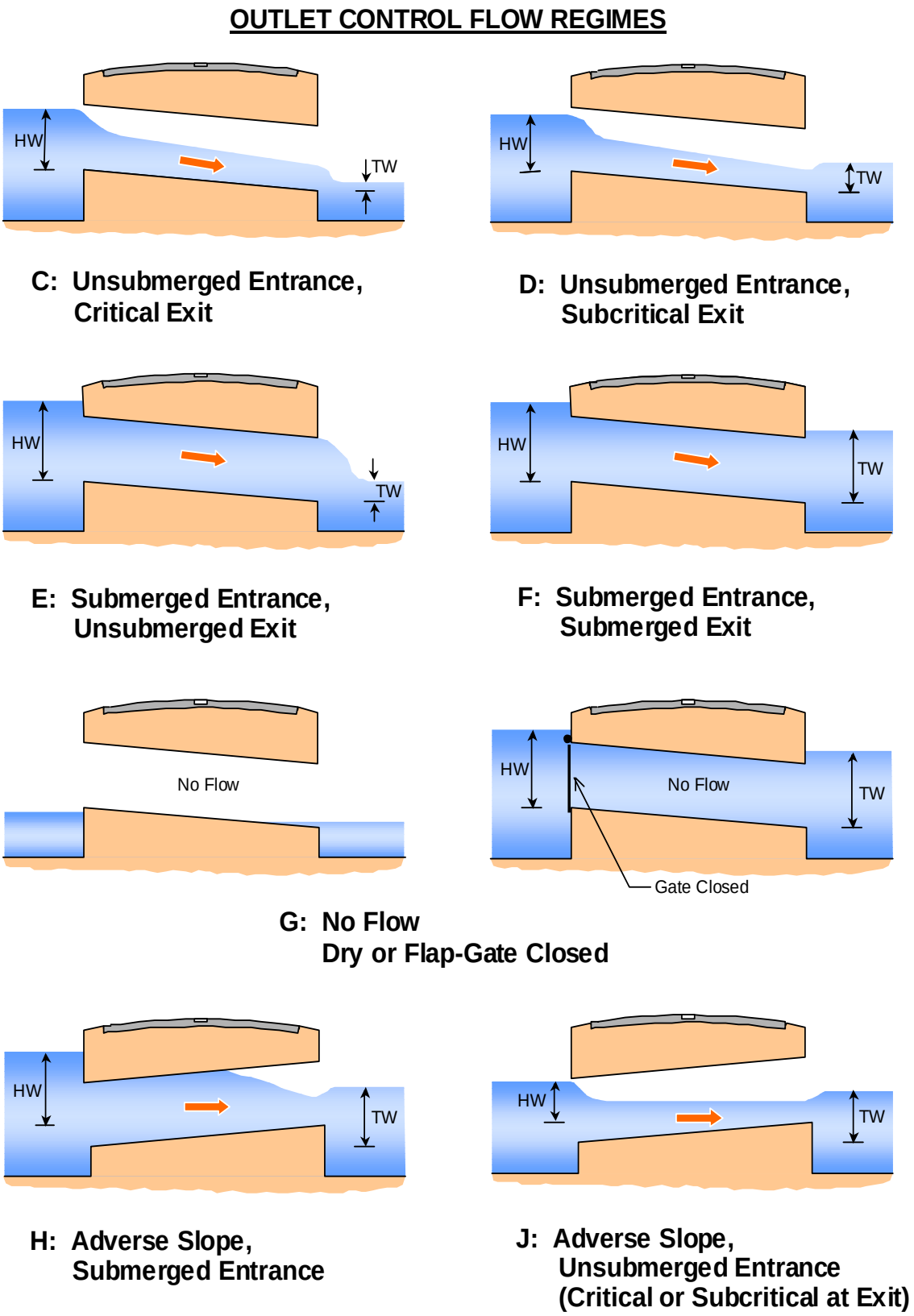


Figure 5-2 1D Outlet Control Culvert Flow Regimes

5.7.2 Bridges

5.7.2.1 Bridges Overview

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 5.11.1). The bridge opening cross section is described in the same manner to a normal channel.

As of the TUFLOW 2016-03 release two types of bridge channels can be specified. The original bridge channel is denoted as "B", while the new bridge channel is of type "BB".

B bridges require the user to specify an energy loss versus elevation table, usually derived from loss coefficients in the literature such as "Hydraulics of Bridge Waterways" (Bradley, 1978). The energy loss table can be generated automatically via the 1d_nwk Form_Loss attribute if the energy loss coefficient is constant up to the underside of the bridge deck.

BB bridges automatically calculate the form (energy) losses associated with the approach and departure flows as the water constricts and expands, and also automatically apply bridge deck losses associated with pressure flow. The only user specified loss coefficients required for BB bridges are the pier losses and the deck losses once fully submerged. If the pier loss coefficient is constant through the vertical the coefficient can simply be specified via the 1d_nwk Form_Loss attribute as described further below. BB bridges will also support the automatic generation of pier losses based on a bridge definition input feature currently under development, thereby not requiring the user to pre-calculate the energy loss coefficients.

Two bridge flow approaches 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. This is discussed further below. Method B is the approach recommended with Method A provided for legacy models.

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

5.7.2.2 Bridge Cross-Section and Loss Tables

The cross-sectional shape of the bridge is specified in the same manner as for open channels using a 1d_xs GIS layer (refer to Section 5.10 and

Table 5-14) and the command Read GIS Table Links. The line or polyline is digitised midway across the 1d_nwk channel line (do not specify as an end cross-section, ie. a cross-section line snapped to an end of the bridge channel). As per the open channel, the cross-section data can be in offset-elevation (XZ) or height-width (HW) format.

Bridge structures are modelled using a height varying form or energy loss coefficient. A table (referred to as a Bridge Geometry "BG" or Loss Coefficient "LC" Table) of backwater or form loss coefficient versus height is required. The interpretation of loss coefficients provided by the user differs depending on whether the bridge channel is of a B or BB type as discussed in the following sections.

BG Tables can be entered using .csv files via a 1d_bg GIS layer (see

Table 5-6) using the command Read GIS Table Links. A line or polyline is digitised crossing the 1d_nwk channel in the same manner as for the 1d_xs GIS layer used to define the cross-sectional shape of the bridge. The line does not have to be identical to the cross-section line.

Where the loss coefficient is constant through to the bridge deck (eg. no losses such as a clear spanning bridge, or pier losses only – see BB bridges), the BG table can automatically be created by specifying a positive non-zero value for the Form_Loss attribute in the 1d_nwk layer (see Table 5-5). How the Form_Loss attribute is interpreted differs between B and BB bridge channels as discussed in the following sections.

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 or BBW channel can be specified (see Section 5.7.3.5).

5.7.2.3 B Bridge Losses Approach

The coefficients for B bridges are usually obtained from publications such as "Hydraulics of Bridge Waterways" (Bradley, 1978), 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) make up the primary contributors to K_b.
- The backwater coefficient K_b input into the LC 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 (via consideration of the upstream and downstream cross-section areas), in which case the losses for the B channel should be fixed. This is the default setting or can be manually specified using the "F" flag (ie. a "BF" channel) in the 1d_nwk Type attribute, or use Structure Losses == ADJUST EXCEPT BG TABLES (the default).

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.

The value of 1.5625 is derived from the following equation presented in Waterway Design - A Guide to the Hydraulic Design of Bridges, Culverts and Floodways (Aust Roads, 1994):

$$Q = C_d b_N Z \sqrt{2gdH}$$

Where:

Q = Total discharge (m^3/s)
C_d = Coeffecient of discharge (0.8 for a surcharged bridge deck)
b_N = Net width of waterway (m)
Z = Verticle distance under bridge to mean river bed (m)
dH = Upstream energy (or water surface) level minus downstream water surface level (m)

Assuming $V = \frac{Q}{b_N Z}$ and $dh = K\frac{V^2}{2g}$, the equation rearranges to give $K = \frac{1}{C_d^{\,2}}$, where a C_d value of 0.8 equates to a K value of 1.5625.

5.7.2.4 BB Bridge Losses Approach

BB bridges were introduced for Build 2016-03-AA. They differ from B bridges in that the losses due to flow contraction and expansion, and the occurrence of pressure flow is handled automatically. The only loss coefficients required to be specified are those due to any piers, and the bridge deck when it is submerged and not under pressure flow.

BB bridges will also in a future release support the bridge definition input feature currently being developed where the variation in pier and deck loss coefficients with height are automatically determined, thereby removing the need for the user to manually generate a LC table.

The entrance and exit losses are adjusted very timestep according to the approach and equations in Section 5.7.6. This approach yields similar results to the approach for determining contraction and expansion losses in publications such as Hydraulics of Bridge Waterways (Bradley, 1978).

Pressure flow is handled by transitioning from the equation described in the previous section (to derive the K value of 1.5625 from a coefficient of discharge value of 0.8) to a fully submerged situation where a deck energy loss is applied. The 1d_nwk HConF_or_WC attribute can be used to vary Cd (default

value is 0.8) and the WConF_or_WEx attribute to set the submerged deck loss coefficient. When pressure flow results the "P" flag will appear in the .eof file and _TSF layer.

The parameters used by the BB bridge routine are:

- CD is the Bridge Deck surcharge coefficient (Default = 0.8).
- DLC is the Deck loss coefficient (Default = 0.5) and only applies when no LC table exists and an automatically generated table using the 1d_nwk Form_Loss attribute is created.
- ELC is the unadjusted entry loss coefficient (Default = 0.5).
- XLC is the unadjusted exit loss coefficient (Default = 1.0).

New .ecf command "Bridge Zero Coefficients == <CD>, <DLC>, <ELC>, <XLC>" can be used to change the above default values to the user's own default values.

The above values can also be changed for a bridge using the following 1d_nwk attributes. If the attribute value is zero then the default value or the value specified by Bridge Zero Coefficients is used.

- CD = HConF_or_WC
- DLC = WConF_or_WEx
- ELC = EntryC_or_WSa
- XLC = ExitC_or_WSb

BB bridges have their entrance and exit losses adjusted every timestep according to the approach and departure velocities as per the equations below (also see Section 5.7.6).

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

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

where: V = Velocity (m/s)
C = Energy Loss Coeffecient

LC tables for BB bridges should therefore only be the losses due to piers and bridge decks. The LC table should not include any losses for contraction and expansion.

If no LC table exists for the BB bridge, and the 1d_nwk Form_Loss attribute is greater than 0.0001, a LC table is automatically generated using Form_Loss for the pier losses and the WConF_or_WEx for the Deck Loss coefficient (DLC).

Note: If a LC table exists, the Form_Loss value is added to all the LC table loss values.

Other notes are:

- BB bridges are only available if Structure Routines == 2013 (the default).
- The unadjusted entry and exist losses (ELS and XLC) cannot be below 0 or greater than 1, and will be automatically limited to these values.
- _TSF and _TSL layers contain the following flags/values for BB bridges:
  - For normal flow (“ ” or “D” if drowned out): fixed / adjusted components
  - For Pressure (“P”) flow: Deck surcharge Coefficient / 0.0
  - Other flags:
    - “U” for upstream controlled flow – only occurs when downstream water level is below bridge bed level.
    - “Z” for zero or nearly zero flow.

Table 5-5 Bridges: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. Digitised nodes can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	“B” or “BB” as specified in Table 5-1.	Char(4)
3	Ignore	If a “T”, “t”, “Y” or “y” is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes).	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 5.11 for further discussion.	Char(1)
5	Len_or_ANA	Only used in determining nodal storages if the UCS attribute is set to “Y” or “T”. Not used in conveyance calculations.	Float
6	n_nF_Cd	Not used.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
7	US_Invert	Sets the upstream and downstream 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. **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.**	Float
8	DS_Invert	Sets the downstream invert of the channel using the same rules as for described for the US_Invert attribute above.	Float
9	Form_Loss	If a LC table exists, for BB bridges adds the value specified to the loss coefficients in the LC table. Not added to LC tables for B bridges. If no LC table exists, and the value is greater than zero, TUFLOW automatically generates a LC table of constant loss coefficient up until the bridge deck (ie. the top of the cross-section). The interpretation of the LC table generated from the Form_Loss value differs depending on whether a B or a BB bridge as follows: For B bridges (with no LC table): • The Form_Loss (LC value) is not subject to adjustment when using Structure Losses == ADJUST. • Above the underside of the bridge deck (the top of the cross-section) a value of 1.5625 is used, with the 1.5625 less the Form_Loss value is subject to adjustment when using Structure Losses == ADJUST. For BB bridges (with no LC table): • The Form_Loss (LC value) is only used to apply pier losses. Contraction/expansion losses and bridge deck pressure flow losses are automatically calculated. • Once the bridge deck is fully submerged (and no longer under pressure flow) the WConF_or_WEx attribute is used to set the additional losses due to the bridge deck.	Float
10	pBlockage	Not used. Reserved for future builds to fully or partially block B channels. The 1d_xs Skew attribute can be used to partially block cross-sections of these channels – see Table 5-14.	Float
11	Inlet_Type	This attribute is used to manage MIKE 11 1D cross-section data. If attribute is not blank, TUFLOW searches the active cross-section	Char(256)

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		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 Inlet_Type (River name), Conn_1D_2D (Topo ID) and Conn_No (XSect ID or Chainage) attributes.	
12	Conn_1D_2D	See description for Inlet_Type above for MIKE 11 cross-section data. Used to reference the Topo ID from the MIKE 11 cross-section data if desired. If Conn_1D_2D is "$LINK", searches the active MIKE 11 network (.nwk11) file for the link cross-section details. This attribute is also used to manage Flood Modeller cross-section data. If a Flood Modeller database (.pro file), finds the processed data based on the label specified in the Conn_1D_2D attribute.	Char(4)
13	Conn_No	See description for Inlet_Type and Conn_1D_2D above for MIKE 11 cross-section data. If being used for a MIKE 11 cross-section, this attribute should match the Chainage specified to the nearest integer. For links, Conn_No must equal or fall within the upstream and downstream chainages of the link.	Integer
14	Width_or_Dia	Not used.	Float
15	Height_or_WF	Not used.	Float
16	Number_of	Not used.	Integer
17	HConF_or_WC	B bridges: Not used. BB bridges: Bridge deck pressure flow contraction coefficient (C_d). If set to zero the default of 0.8 or that specified by Bridge Zero Coefficients is used.	Float
18	WConF_or_WEx	B bridges: Not used. BB bridges: Bridge deck energy loss coefficient (*DLC*) for fully submerged flow. If set to zero the default of 0.5 or that specified by Bridge Zero Coefficients is used.	Float
19	EntryC_or_WSa	B bridges: Not used. BB bridges: Unadjusted entrance energy loss coefficient (*ELC*). If set to zero the default of 0.5 or that specified by Bridge Zero Coefficients is used.	Float
20	ExitC_or_WSb	B bridges: Not used.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		BB bridges: Unadjusted exit energy loss coefficient (*XLC*). If set to zero the default of 1.0 or that specified by Bridge Zero Coefficients is used.	

Table 5-6 1D Bridge Geometry Table Link (1d_bg) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
1	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(50)
2	Type	**"BG" or "LC":** Bridge energy loss coefficients (second column) versus elevation (first column) for bridge structures.	Char(2)
3	Flags	No optional flags.	Char(8)
4	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(20)
5	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(20)
6	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(20)
7	Column_4	Optional. Defines the fourth column of data.	Char(20)
8	Column_5	Optional. Defines the fifth column of data.	Char(20)
9	Column_6	Optional. Defines the sixth column of data.	Char(20)
10	Z_Increment	Not used.	Float
11	Z_Maximum	Not used.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
12	Skew (in degrees)	Not used.	Float

5.7.3 Weirs

5.7.3.1 Weirs Overview

A range of weir types are available as listed in Table 5-7. 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 5.11.1).

All weirs have three flow regimes of zero flow (dry), upstream controlled flow (unsubmerged) and downstream controlled flow (submerged).

Table 5-7 Weir Types

Weir Type	Description
W	The original ESTRY weir based on the broad-crested weir formula with the Bradley submergence approach (see Weir Approach == METHOD A). The weir shape is defined either as a rectangle using the 1d_nwk Width attribute or as a cross-section. This was the only weir option prior to the 2013-12 release.
The weir types below were introduced for the TUFLOW 2013-12 release to offer a greater range of weir types/equations, with more user flexibility to define submergence curves and other parameters.	
WB	Broad-crested weir. A rectangular section shape is assumed.
WC	Crump weir
WD	User-defined weir
WO	Ogee-crested weir
WR	Rectangular weir (sharp crested)
WT	Trapezoidal weir or Cippoletti weir
WV	V-notch weir. Note, prior to the 2013-12 release a variable weir channel could be specified as a WV channel. As of the 2013-12 release WV channels are processed as a V-notch weir, so any WV channels will need to be changed to VW.
WW	Similar to the original W weir channel, but has more options allowing the user to customise the weir sub-mergence curve and other parameters. Can be based on either a rectangular shape using the 1d_nwk Width attribute or on a cross-section.

5.7.3.2 Original Weirs (W)

For a "W" type weir, a standard weir flow formula is used as per the equation below. The weir is assumed to be broad-crested. Weirs with different characteristics should be modelled using one of the other weir types listed in Table 5-7 and discussed in Section 5.7.3.3.

$$Q_{weir} = \frac{2}{3} CW \sqrt{\frac{2g}{C_f}} H^{\frac{3}{2}}$$

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

Where:

Q_{weir} = Unsubmerged flow over the weir (m^3/s)
$V_{approach}$ = Velocity approaching the weir (m/s)
C = Broad-crested weir coefficient of 0.57
W = Flow width (m)
C_f = Weir calibration factor (default of 1.0 – refer to 1d_nwk "Height_or_WF" attribute)
H = Depth of water approaching the weir relative to the 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, although use of the other weir types listed in Table 5-7 is recommended.

For benchmarking of unsubmerged and submerged weir flow to the literature, see Huxley (2004), which is available at www.tuflow.com.

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).

For submergence of W weir channels, it is recommended that Weir Approach is set to METHOD A or METHOD C. Both METHOD A and METHOD C utilises the Bradley submergence approach (METHOD C is a slight enhancement that only affects WW weir channels). The Bradley Submergence approach (Bradley, 1978, Figure 24) is handled by fitting the equation below to Bradley's submergence curve reproduced in Figure 5-3 and applying the submergence factor to the weir equation above.

Once the percentage of submergence exceeds 70%, the submergence factor applied is given by the equation below. The Bradley curve (as digitised) and the resulting curve from the equation below are shown alongside the submergence curves used for other weir types in Figure 5-4 and Figure 5-5. The

submergence factor transitions the flow from weir flow to zero flow as the water level difference (*dH*) approaches zero.

$$C_{sf} = 1 - \left(1 - \frac{dH}{H}\right)^{20}$$

Weir Approach == METHOD B was introduced to provide additional stability for 1D flow over roadways connected to 2D domains, however, it can cause excessive head drops under drowned flow conditions and is therefore not recommended or to be used with caution. METHOD B only applies to the W weir – it has no influence over the new weirs introduced in the 2013-12 release as documented in the following sections.

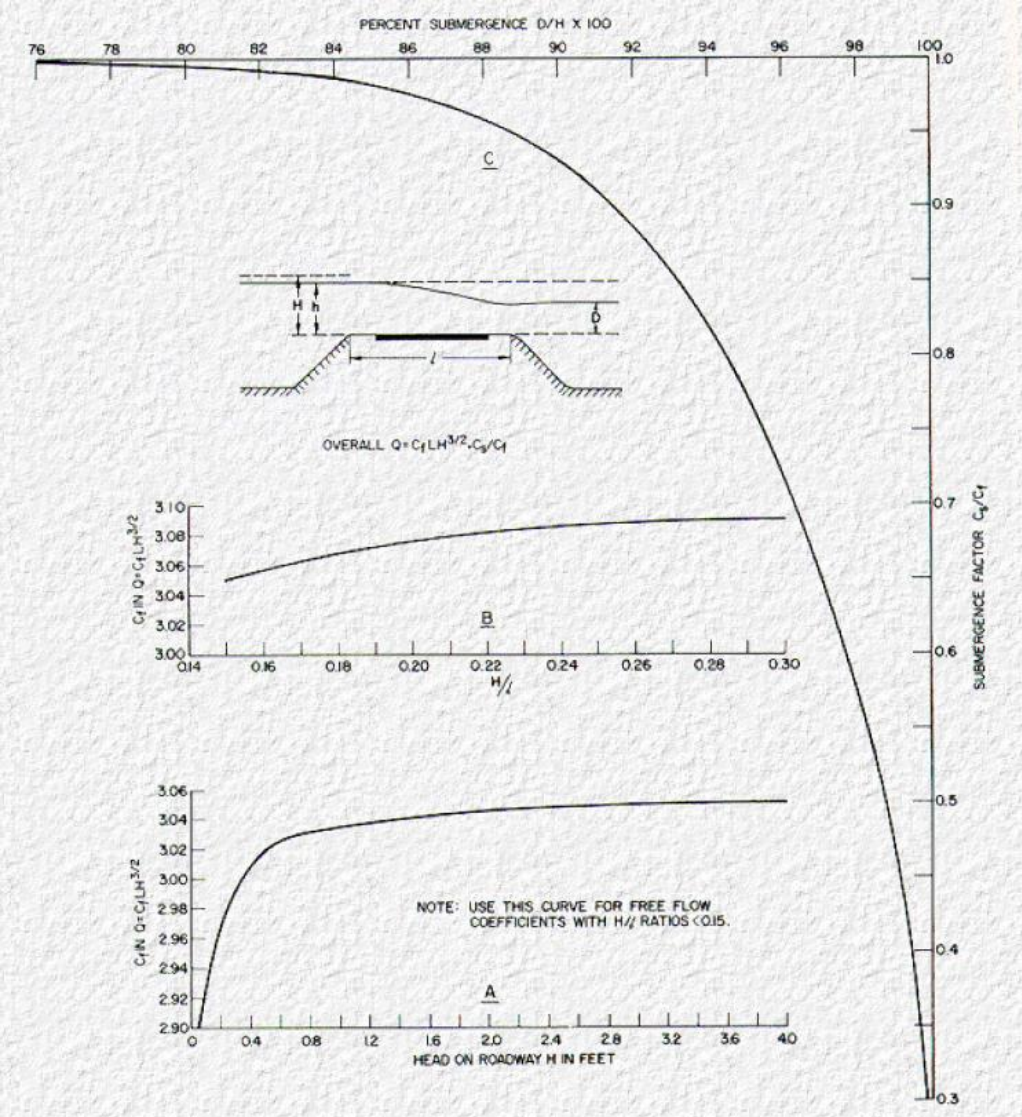


Figure 5-3 Bradley Weir Submergence Curve, Bradley 1978

5.7.3.3 Advanced Weirs (WB, WC, WD, WO, WR, WT, WV, WW)

All weirs other than the original 'W' type weir channel, as listed in Table 5-7 were introduced for the TUFLOW 2013-12 release. These weirs offer greater variety, flexibility and can be customised by the user. As of the 2016-03 release most of these weirs can be operated – see Section 5.9.

The weir flow when upstream controlled is determined by the following equation.

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

Where:

Q = Flow over the weir (m^3/s)
C_d = Weir coefficient
C_{sf} = Weir submergence factor
C_f = Weir calibration factor (default of 1.0 – refer to 1d_nwk "Height_or_WF" attribute)
W = Flow width (m)
H = Upstream water surface or energy depth relative to the weir invert (m) – see note below
Ex = Weir flow equation exponent

Notes

1. The default values for C_d are provided in Table 5-8, and documented further below for weirs where C_d is recalculated each timestep.
2. The approach taken for calculating the weir submergence factor C_{sf} each timestep is documented below.
3. The weir calibration factor, C_f, is by default 1.0 and should only be changed should there be a good justification.
4. For weirs where the flow width, *W*, varies, eg. a V-notch WV weir, the formula for that weir takes into account the varying width.
5. Whether water surface depth or the energy level is used for *H* depends on the Structure Flow Levels setting, which can be changed on a structure by structure case using the E or H flag (see Table 5-1).
6. The default values for *Ex* are provided in Table 5-8.

Table 5-8 presents the weir coefficient C_d and weir flow exponent Ex used for each weir. Some of these values are derived from dimensional forms of the weir equations. Values other than the default values shown in Table 5-8 may be used by altering the attributes of the 1d_nwk layer. Refer to Table 5-9 for further information.

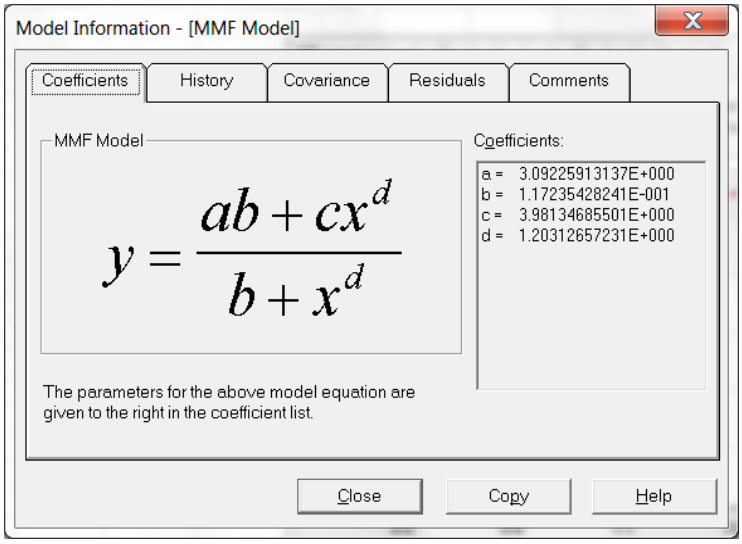
Note that C_d for WO and WV weirs is recalculated every timestep as described in the following sections. It is possible to override this by specifying a non-zero positive value for the "HConF_or_WC" attribute in the 1d_nwk layer. For WD weirs the user must specify a non-zero positive value.

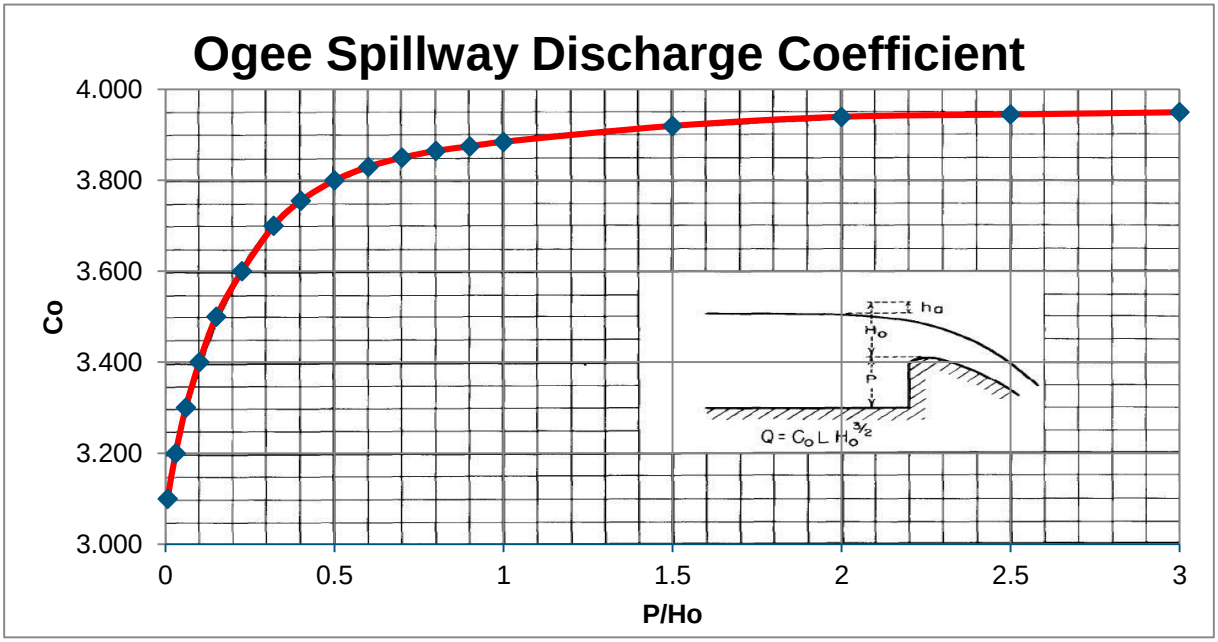
Table 5-8 Default Attribute Values for the Weir Equation for Different Weir Flows

Channel Type	C_d (HConF_or _WC)	Ex (WConF_or _WEx)	a (EntryC_or _WSa)	b (ExitC_or _WSb)	Default Submergence Curve (Figure 5-4 and Figure 5-5)
SP	0.75	1.5	6.992	0.648	Ogee / Nappe (USBR 1987 and Miller 1994)
WB	0.577	1.5	8.55	0.556	Broad-crested from Abou Seida & Quarashi, 1976 (Miller 1994)
WC	0.508	1.5	17.87	0.59	Crump H1/Hb=1.5 (Bos 1989)
WD	User Defined	1.5	3.0	0.5	User Defined default settings
WO	Recalculated every timestep	1.5	6.992	0.648	Ogee / Nappe (USBR 1987 and Miller 1994)
WR	0.62	1.5	2.205	0.483	Sharp Crest Thin Plate from Hagar, 1987a (Miller 1994)
WT	0.63	1.5	2.205	0.483	Sharp Crest Thin Plate from Hagar, 1987a (Miller 1994)
WV	Recalculated every timestep	2.5	2.205	0.483	Sharp Crest Thin Plate from Hagar, 1987a (Miller 1994)
WW	0.542	1.5	21.15	0.627	Bradley 1978 Broad-crested (Bradley 1978)

5.7.3.3.1 Ogee Crest Weir (WO)

For WO (Ogee crest) weirs, the chart developed in USBR Small Dams, 1987 is used by fitting the relationship presented and plotted on the USBR curve below. The relationship falls within ±0.5% of the curve. Note that the relationship below is for US Customary Units, which is converted to metric if running the simulation in metric units. For setting the value of *P* (the height of the weir crest above the bed), see the discussion for the 1d_nwk US_Invert attribute in Table 5-9.





5.7.3.3.2 V-Notch Weir (WV)

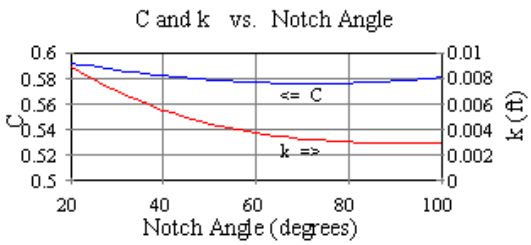
For WV (V-notch) weirs, the approach taken is to use the formulae derived by LMNO Engineering as shown below. For metric models the flow is calculated in ft^3/s and converted to m^3/s.

Equations

V-notch weir equations have become somewhat standardized. ISO (1980), ASTM (1993), and USBR (1997) all suggest using the Kindsvater-Shen equation, which is presented below from USBR (1997) for Q in cfs and heights in ft units. All of the references show similar curves for C and k vs. angle, but none of them provide equations for the curves. To produce automated calculations, LMNO Engineering used a curve fitting program to obtain the equations which best fit the C and k curves. Our equations are shown below. The graph shown is from our fits. If you compare it to the graphs shown in the references, it looks nearly identical which implies that our fits are very good.

$$Q = 4.28\ C\ \tan\left(\frac{\theta}{2}\right)(h+k)^{5/2}$$

where Q = Discharge (cfs)
C = Discharge Coefficient
θ = Notch Angle
h = Head (ft)
k=Head Correction Factor (ft)



$$C = 0.607165052 - 0.000874466963\ Ø + 6.10393334\times10^{-6}\ Ø^2$$

$$k\ (ft.) = 0.0144902648 - 0.00033955535\ Ø + 3.29819003\times10^{-6}\ Ø^2 - 1.06215442\times10^{-8}\ Ø^3$$

where Ø is the notch angle in degrees

5.7.3.4 Advanced Weir Submergence Curves

Weir submergence factors C_{sf} were sought from two sources: "Discharge Characteristics" (Miller, 1994) and "Discharge Measurement Structures" (Bos, 1989). The submergence charts for each weir type, relating the weir submergence factor to the ratio between downstream and upstream water level were reproduced from the literature in Excel and are shown in Figure 5-5.

Two methods were utilised to fit equations to these curves. These are:

- The Rational Function expressed as: $C_{sf} = \frac{a+b\left(H_d/H_u\right)}{1+c\left(H_d/H_u\right)+d\left(H_d/H_u\right)^2}$
- The Villemonte equation, expressed as: $C_{sf} = \left(1-\left(H_d/H_u\right)^a\right)^b$

For each submergence curve, the above equations were solved to obtain values for each of the variables that produced the best fit with the curves provided in the literature. After a comparison of the results, the equation from Villemonte was chosen for a number of reasons:

- The Rational Function was found to be sensitive to variables and therefore required a greater number of decimal places. Villemonte was found to provide accurate results with variables requiring only 2 decimal places.

- The Villemonte equation contains only two variables, compared to four used in the Rational Function, making it simpler and less susceptible to error.
- The Villemonte equation may be solved exactly at the extremities of the curves, ie. where ${H_d}/{H_u} = 1$ and $C_{sf} = 0$, and when ${H_d}/{H_u} = 0$ and $C_{sf} = 1$. The Rational Function required further manipulation through inclusion of additional points to achieve this outcome.

The default variables a and b used to determine the submergence factor C_{sf} for each weir type are presented in Table 5-8. Figure 5-5 shows the submergence curves produced using the default values in Table 5-8 to calculate C_{sf}.

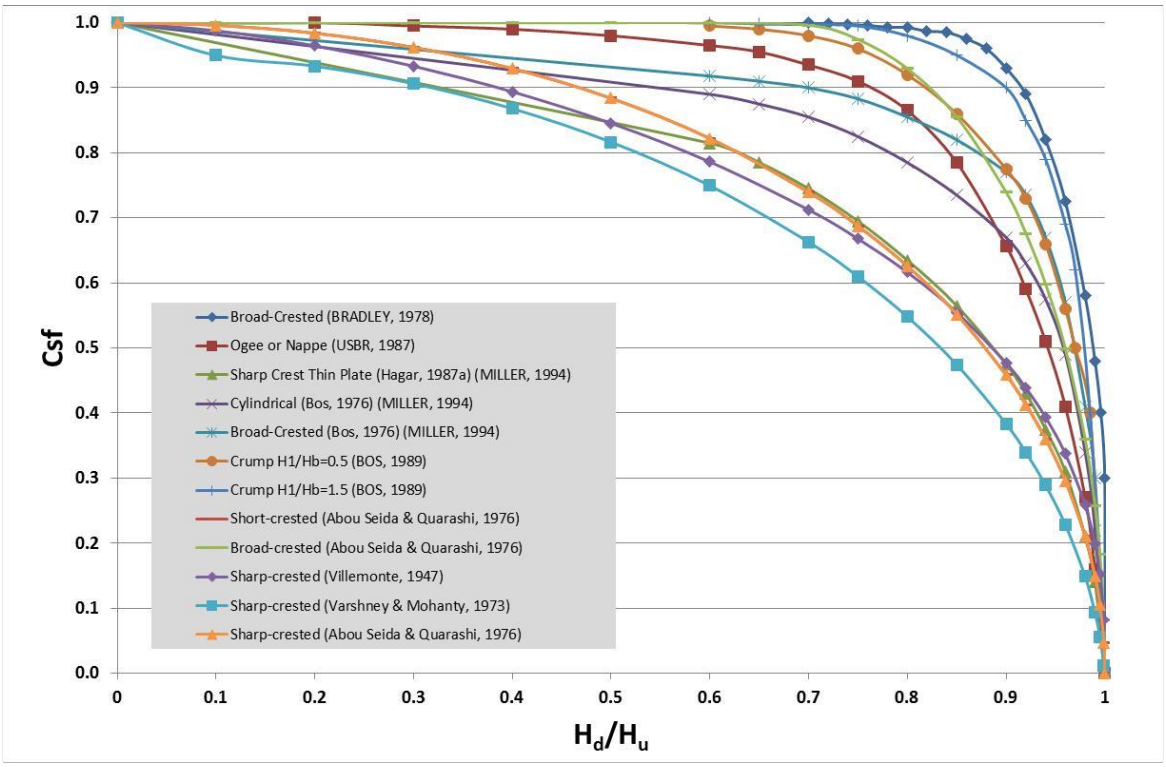


Figure 5-4 Weir Submergence Curves from the Literature

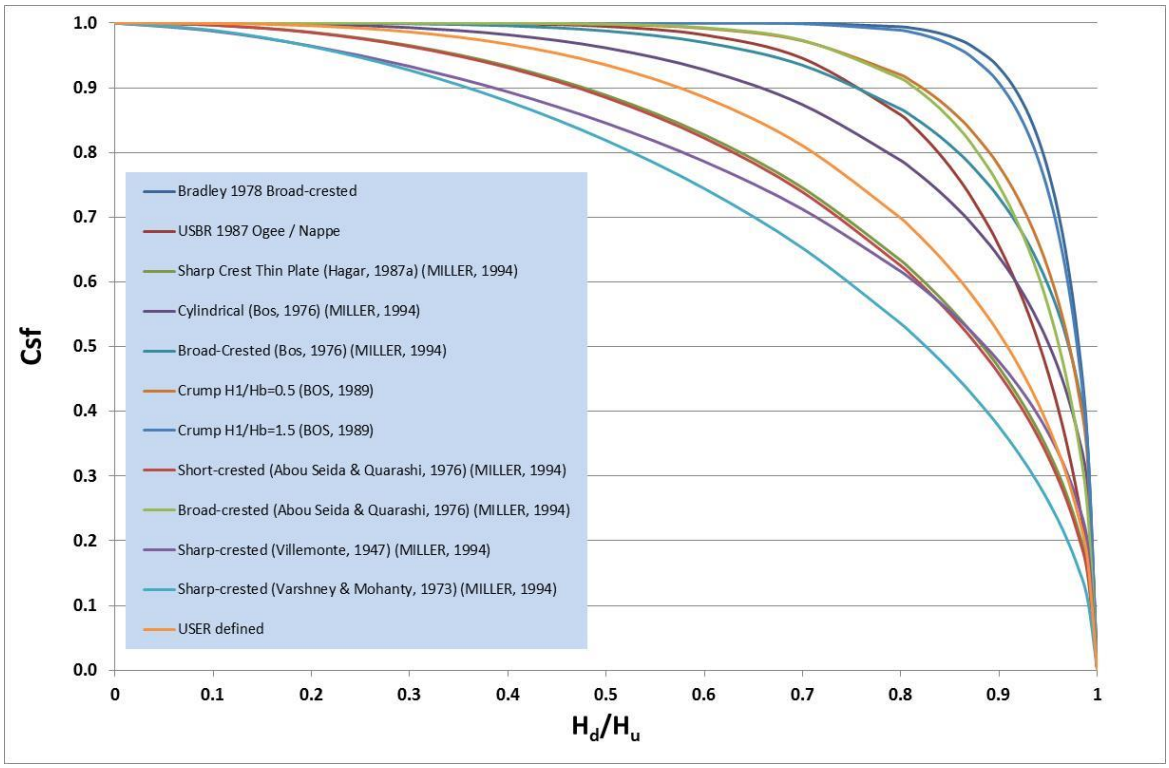


Figure 5-5 Weir Submergence Curves using Villemonte Equation

Table 5-9 Weirs: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. Digitised nodes can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	The weir channel type as specified using the flags in Table 5-1 and Table 5-7. For example, a V-notch weir would be entered as "WV".	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes).	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 5.11.1 for further discussion.	Char(1)
5	Len_or_ANA	Only used in determining nodal storages if the UCS attribute is set to "Y" or "T". Not used in conveyance calculations.	Float
6	n_nF_Cd	Not used.	Float
7	US_Invert	**All Weir (excluding WO) Channel Types:** Sets the weir invert. Note that the invert is taken as the maximum of the US_Invert and the DS_Invert attributes. For W and WW weirs use -99999 to use the bed of the cross-section as the invert or weir crest. The maximum of US_Invert and DS_Invert is used in conjunction with the Width_or_Dia attribute to define a rectangular section 5m (16.4ft). The automatic height given to the weir is 5m (16.4ft) so that the generation of node storage areas from channel widths are within a realistic range of elevations. Use Depth Limit Factor to allow water levels to exceed the 5m (16.4ft) range if required. Note: For W and WW weirs if a cross-section for the channel exists, the cross-section profile will prevail over the automatic rectangular shape. **WO Channel Type:** The absolute difference in height between the US_Invert and	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		DS_Invert is used to set the height of the weir above its sill (usually denoted as P), which is used for recalculating the weir's discharge coefficient each timestep. If the US_Invert and DS_Invert are the same value the primary upstream channel bed will be used to set the value of P.	
8	DS_Invert	See comments above for US_Invert.	Float
9	Form_Loss	Not used.	Float
10	pBlockage	**W Channel Type:** Not used. Reserved for future builds to fully or partially block W channels. The 1d_xs Skew attribute can be used to partially block cross-sections of these channels - see Table 5-14. **WB, WC, WD, WO, WR, and WS Channel Type:** The weir width is adjusted proportionally by the % blockage. **WT Channel Type:** The base width of the weir is adjusted proportionally by the % blockage. **WV Channel Type:** The V-notch angle is adjusted proportionally by the % blockage.	Float
11	Inlet_Type	This attribute is used to manage MIKE 11 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 Inlet_Type (River name), Conn_1D_2D (Topo ID) and Conn_No (XSect ID or Chainage) attributes.	Char(256)
12	Conn_1D_2D	See description for Inlet_Type above for MIKE 11 cross-section data. Used to reference the Topo ID from the MIKE 11 cross-section data if desired. If Conn_1D_2D is "$LINK", searches the active MIKE 11 network (.nwk11) file for the link cross-section details. This attribute is also used to manage Flood Modeller cross-section data. If a Flood Modeller database (.pro file), finds the processed data based on the label specified in the Conn_1D_2D attribute.	Char(4)
13	Conn_No	See description for Inlet_Type and Conn_1D_2D above for MIKE 11 cross-section data.	Integer

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		If being used for a MIKE 11 cross-section, this attribute should match the Chainage specified to the nearest integer. For links, Conn_No must equal or fall within the upstream and downstream chainages of the link.	
14	Width_or_Dia	**All Weir (excluding WT and WV) Channel Types:** The weir width in metres. For W and WW weirs, this attribute is ignored if a cross-section for the channel exists. See discussion for the US_Invert attribute above. **For operational weirs**, the width of the weir when fully open. **WT Channel Type:** The width at the base of the trapezoidal weir. **WV Channel Type:** Angle of the V-notch in degrees. Must be between 20º and 100º.	Float
15	Height_or_WF	**For non-operated weirs**, this value can be used as a weir coefficient adjustment factor to be primarily used for model calibration or sensitivity testing. The weir coefficient is multiplied by this value. The resulting weir coefficient can be viewed in the .eof file and over time in the _TSL GIS layer. If zero or negative an adjustment factor of 1.0 (ie. no adjustment) is applied. See Section 5.7.3. **For operational weirs**, the height of the weir above the crest when fully up.	Float
16	Number_of	Not used.	Integer
17	HConF_or_WC	**W Channel Type:** Not used. **All Weir (excluding W) Channel Types:** Weir coefficient, C_d, in its dimensionless form. If less than or equal to zero the default value for the weir type in Table 5-8 is used. Note that for WO and WV weirs the default is to recalculate C_d every timestep. Entering a value greater than zero (0) will override this and apply a fixed C_d. For WD weirs the user must specify a non-zero positive value. Note that published weir coefficients may be based on other non-dimensional or dimensional forms of the weir equation, therefore care should be taken in ensuring the coefficient is compatible with the form of the weir flow equation presented in Section 5.7.3.3.	Float
18	WConF_or_WEx	**W Channel Type:** Not used.	Float

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
		All Weir (excluding W) Channel Types: Weir flow equation exponent Ex in the weir flow equation presented in Section 5.7.3.3). If less than or equal to zero the default value for the weir type in Table 5-8 is used. The default value is 1.5 for all weir types except for WV which is 2.5.	
19	EntryC_or_WSa	**W Channel Type:** Not used. **All Weir (excluding W) Channel Types:** Sets the submergence factor "a" exponent in the Villemonte Equation for calculating the weir submergence factor C_{sf} (refer to equations in Section 5.7.3.3 and 5.7.3.4). If less than or equal to zero the default value for the weir type in Table 5-8 is used.	Float
20	ExitC_or_WSb	**W Channel Type:** Not used. **All Weir (excluding W) Channel Types:** Sets the submergence factor "b" exponent in the Villemonte Equation for calculating the weir submergence factor C_{sf} (refer to equations in Section 5.7.3.3 and 5.7.3.4). If less than or equal to zero the default value for the weir type in Table 5-8 is used.	Float

5.7.3.5 Automatically Created Weirs

Weirs representing overtopping of structures such as culverts and bridges may be automatically created without the need to digitise a separate line or polyline within a 1d_nwk layer. The structure must be digitised within a 1d_nwke layer (as opposed to a 1d_nwk layer) and a "W" specified alongside the original structure type. For example, to model a bridge and a weir representing overtopping of the road deck, specify type "BW". The weir crest level and dimensions are specified within the additional attributes contained within a 1d_nwke layer and are explained in Table 5-10. The original W weir approach is adopted for calculating the flow (see Section 5.7.3.2).

The weir's shape is assumed to be two rectangles on top of each other. The lower rectangle is reduced in width according to the percent blockage applied to the rail (ie. the EN4 attribute in Table 5-10), and its height is the EN3 attribute. The upper rectangle is the full flow width and extends indefinitely in the vertical.

Alternatively, the flow over a structure can be manually digitised as a separate 1d_nwk weir channel parallel to the original bridge or culvert structure (ie. the weir is connected to the ends of the bridge/culvert). Any of the available weir types can be used in this instance.

Table 5-10 1D Model Network (1d_nwke) OPTIONAL Attribute Descriptions

No.	Default GIS Attribute Name	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. If < 0.001, uses width 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, depending on the EN4 attribute entry.	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

5.7.3.6 VW Channels (Variable Geometry Weir)

The VW (variable weir) channel allows the modeller to vary the cross-section geometry of a W weir over time using a trapezoidal shape. To set up a VW channel follow the steps below.

Note that prior to the 2013 release, a variable weir channel was specified as a WV channel type. As of the 2013 release, WV channels are processed as a V-notch weir. Variable weir channels must be specified as type "VW".

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

- ID = ID of the channel;
- Type = "VW";
- Len_or_ANA = Nominal length in m (only used for calculating nodal storage if UCS is on);
- US_Invert = -99999 (the invert level is specified in the .csv file discussed below);
- DS_Invert = -99999 (the invert level is specified in the .csv file discussed below);
- Inlet_Type = relative path to a .csv file containing information on how the weir geometry varies; and
- 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).

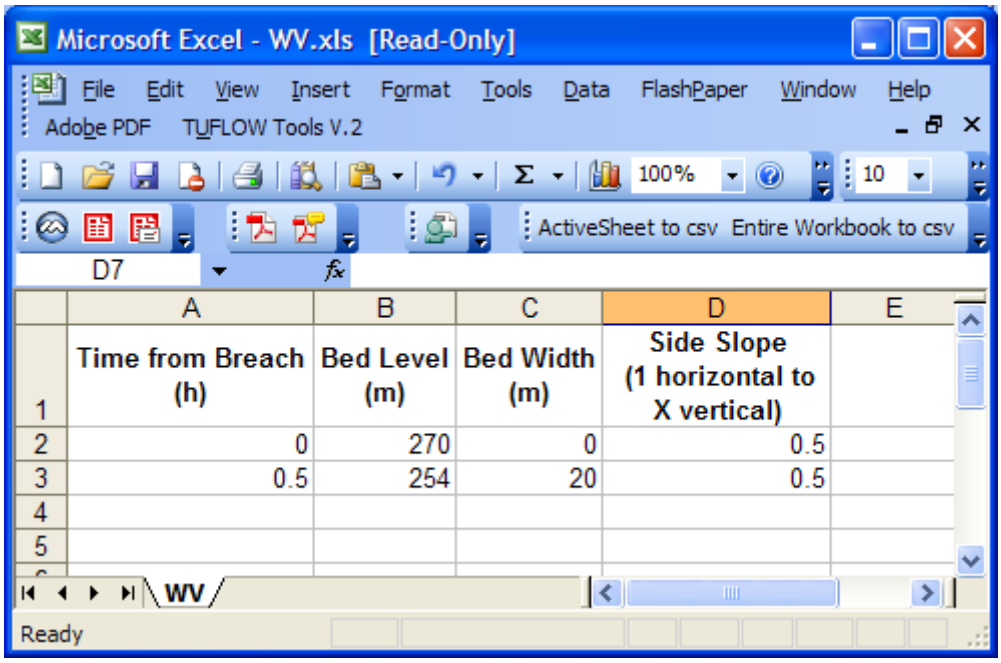
The .csv file must be structured as follows (also see example below):

1. TUFLOW searches through the sheet until more than 4 numbers are found at the beginning of a row (Row 2 in the example below).
2. 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.
3. 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.

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.

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.

The original W weir approach is adopted for calculating the flow (see Section 5.7.3.2).



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

5.7.4 Spillways (SP)

Spillways ('SP') were introduced for the TUFLOW 2013-12 release, and may also be used in operational mode as a gated spillway (see Section 5.9). Spillways may also be simulated and operated as Q or QO channels where the user provides the stage discharge relationships (see Section 5.8.2 and Section 5.9.2.2). The 1d_nwk attributes are presented in Table 5-11.

Spillways use the same equation as for advanced weirs (Section 5.7.3.3). For ungated spillways (ie. SP, non-operated spillways) the same parameters as for Ogee Weirs are the default (see Table 5-8), except for Cd, which is fixed with the default value of 0.75. For Ogee Weirs, C_d is recalculated every timestep (see Section 5.7.3.3.1). The 1d_nwk attributes in Table 5-8 can be used to modify the flow equation parameters for SP channels in a similar manner for advanced weirs.

SPO channels also use the same equation when the gate is not affecting the flow (see more information on SPO channels refer to Section 5.9.2.5).

SP and SPO channels can also drown out as per the submergence curves for advanced weirs.

5.7.5 Sluice Gates (SG)

For sluice gates refer to Section 5.9.2.4. The same approach applies as for SGO operated gates, except that the gate is assumed to be in a fixed position based on the 1d_nwk Height_or_WF attribute value. The 1d_nwk attributes are presented in Table 5-11.

5.7.6 Adjustment of Contraction and Expansion Losses

The energy losses associated with the contraction and expansion of flow lines into and out of a structure, can 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. In this situation, there is effectively no entrance (contraction) or exit (expansion) losses and the losses need to be reduced to zero. 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. This is the most common case where the application of the full entrance and exit loss coefficients (typically 0.5 and 1.0) will overestimate the energy loss through the structure. The full values are only representative of the situation where the approach and departure velocities are close to zero, for example, a culvert discharging from a lake into another lake where the velocity transitions from still water to fast flowing and to still water.

The entrance and exit losses are adjusted according to the equations below to take into account the change in velocity caused by the structure. The first equation is empirical, while the second equation to adjust exit losses can be derived from first principles.

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

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

where: V = Velocity (m/s)
C = Energy Loss Coeffecient

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

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

The adjustment of losses feature is available to structures that require entrance and exit loss coefficients, namely culverts and bridges. For culverts, the adjusted entrance loss coefficient only applies where the flow is not inlet controlled (ie. Regimes C, D, E, F, H and J in Figure 5-2), and the adjusted exit loss is

only influential where the flow is downstream controlled (ie. Regimes D, F, H and J (subcritical at exit) in Figure 5-2). For bridges, the application varies as discussed below.

If Structure Losses == ADJUST EXCEPT BG TABLES (default), the adjustment of the losses is only applied to culverts and B bridges with automatically generated loss tables using the 1d_nwk Form_Loss attribute (see Table 5-5). There is no adjustment for B bridges with a user specified BG or LC energy loss table, because bridge energy losses are most often based on coefficients from publications such as Hydraulics of Bridge Waterways (Bradley, 1978) that have already taken into account the effects of contraction and expansion.

For BB bridges (Build 2016-03-AA or later), the LC table should only represent pier and submerged deck losses (see Section 5.7.2.4), as the adjustment of entrance and exit losses every timestep as per the equations above is always applied irrespective of the Structure Losses setting. The equations above conform with the approach for determining contraction and expansion losses in publications such as Hydraulics of Bridge Waterways.

For B bridges and culverts, if Structure Losses is set to "ADJUST", or "A" has been specified in the 1d_nwk Type attribute (eg. BA, CA, IA or RA), the entrance and exit losses are adjusted according to the equations above. For B bridges, because the entrance and exit losses are combined as one loss coefficient, the entrance and exit loss components are proportioned one-third / two-thirds respectively when applying the above equations. For the new BB bridge, entrance and exit losses are always adjusted as per the equations above and the Structure Losses setting is not relevant (also see Section 5.7.2.4).

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 whose 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").
- X channels can be used to connect additional channels and ensure that these are not considered the primary channel.

TUFLOW has no requirement of a minimum loss coefficient value for stability, and therefore allows the adjusted coefficient to approach zero allowing this feature to correctly model 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_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 for BB bridges entered using the Form_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.
- The ends of culverts and bridges that are connected to 2D SX or HX cells as the approach or departure velocity needs to be derived in some manner from the 2D velocity field. It is intended to offer this option in future releases, plus it is important not to be duplicating energy losses by applying exit losses to a 1D structure and simulating the same energy losses due to the flow expansion in the 2D domain – for further information see this pdf presentation.

If Structure Losses is set to FIX, or "F" has been specified in the 1d_nwk Type attribute (eg. BF, CF, IF or RF), the loss coefficients for B bridges and culverts are not adjusted. Fixing the entrance and exit losses for BB bridges is not available – use a B bridge instead.

The variation in time of the loss coefficients can be viewed using the _TSL output layer (see Section 13.2.3).

If there is a manhole at the culvert end, a manhole energy loss approach (see Section 5.12.5) is used instead of the culverts contraction/expansion loss, and the above description does not apply.

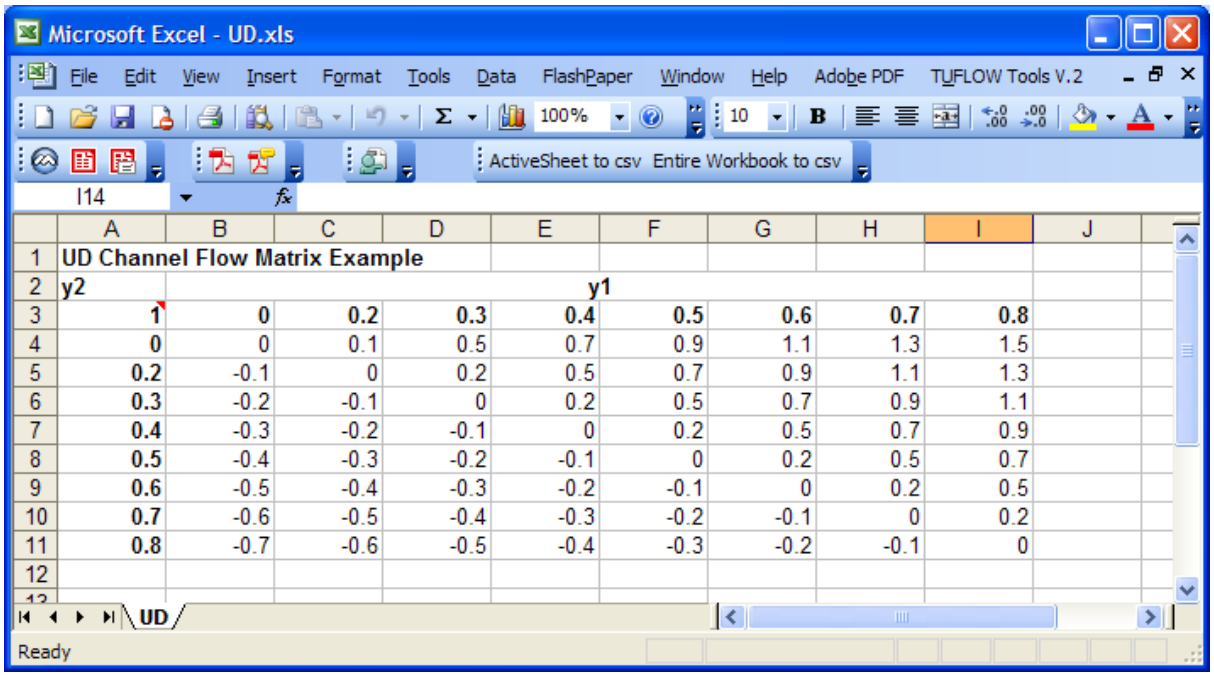
This pdf of a presentation provides further information on this topic.

5.8 Special Channels

5.8.1 M Channels (User Defined Flow Matrix)

M channels allow the modeller to define the flow through a channel (usually a structure) based on a user specified flow matrix. To set up M channels follow the steps below:

1. In the 1d_nwk layer, populate the required attributes as shown in Table 5-11.
2. Create the flow matrix as shown in the image below and export the file to .csv using the Save csv tool. The .csv file is referenced in the 1d_nwk Inlet_Type attribute. Notes using the example in the below image 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) followed by upstream depth values (in the direction that 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. Flows are listed in the adjacent columns relating to the above upstream depth value (Row 3).
   d) Note: 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 in the flow area matrix (the value in the flow matrix is however 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, TUFLOW will calculate the area based on the channel width multiplied by the pBlockage and the average of the upstream and downstream depths.



	A	B	C	D	E	F	G	H	I
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

The default method for interpolating into the matrix is Method B. The previous method (pre Build 2012-05-AB) utilises Method A which can be enabled using the .ecf command M Channel Approach == Method A or Defaults == Pre 2012. For further information refer to the release notes for the 2012 release or contact support@tuflow.com.

5.8.2 Q Channels (Upstream Depth-Discharge Relationship)

Q channels are used to model flow through a channel using an .ecf Depth Discharge Database. The Depth Discharge Database is the same as the Pit Inlet Database used for Q pits, with the same database used for both Q channels and Q pits. Refer to Section 0.

In the 1d_nwk layer, the following attributes can be used to set up a Q channel (also see Table 5-11).

- ID = Unique Channel ID.
- Type = "Q".
- US_Invert = Elevation corresponding to zero depth in the depth-discharge curve.
- Inlet_Type = The depth discharge curve in the Depth Discharge Database or Pit Inlet Database. This is analogous to a Q pit (see Section 0). Note that the flow is automatically adjusted for being drowned out using the Bradley relationship for weirs (see Section 5.7.3 or Figure 5-3), and if the flow reverses the same depth discharge curve is used.
- Width_or_Dia = For Q channels can be used as a flow multiplier – this is useful if the depth-discharge curve is a unit flow, ie. flow per unit width. Therefore, if the discharge is unit flow specify the width of the flow, otherwise specify a value of 1.0 (noting that a zero value is treated as 1.0).
- Number_of = The number of parallel Q channels (a zero value is interpreted as one channel).

5.8.3 X Connectors

X connectors are used for connecting a side tributary or pipe into the main flow path. They are digitised as a line or polyline within a 1d_nwk GIS layer with type "X". No other attributes are required.

Use of an X connector has the advantage of allowing different end cross-sections (see Section 5.10.6) or WLLs (see Section 9.5) to be specified for the side channel, rather than using the end cross-section on the main channel. They can also be used in pipe networks to ensure that the angle of the inlet and outlet culverts has been digitised appropriately as this influences the manhole losses calculated when using the Englehund loss approach (see Section 5.12.5.4).

The direction of the X Connector must be digitised starting from the side channel and ending 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.

5.8.4 Legacy Channels

For backwards compatibility, gradient (type 'G') and normal (type 'blank') channels remain supported in the current release of TUFLOW. Sloping (type 'S') open channels are the preferred method of modelling open channels as it incorporates the flow regimes covered by normal and G channels, and include the additional ability of handling super-critical flow. Refer to Section 5.6 for further information.

A normal flow channel is defined by its length, bed resistance and hydraulic properties. The channel can wet and dry, however, for overbank areas (eg. tidal flats or floodplains) G or S 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 for the reasons given above.**

A gradient channel has been 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. They are similar to normal channels, 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.

5.8.5 1d_nwk Attributes (M, P, Q, SG, SP Channels)

The table below covers the 1d_nwk attributes for all channels not covered in other 1d_nwk attribute tables.

Table 5-11 1D Model Network (1d_nwk) Attribute Descriptions (Special Channels)

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. Digitised nodes can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	The channel type as specified using the flags in Table 5-1. For X (connectors), no other attributes are required.	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes).	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 5.11.1 for further discussion.	Char(1)
5	Len_or_ANA	Only used in determining nodal storages if the UCS attribute is set to "Y" or "T". Not used in conveyance calculations.	Float
6	n_ nF_Cd	**M, P, Q Channel Type:** Not used. **SG, SP Channel Type:** Discharge coefficient for the structure if using a fixed coefficient. If the value is less than or equal to zero, the default Cd value of 0.6 for SG and 0.75 for SP is used.	Float
7	US_Invert	**M or P Channel Type:** Invert level of channel. **Q Channel Type:** US_Invert sets the level from which the upstream depth is to be calculated for interpolation into the depth-discharge curve.	Float

		SG Channel Type: The higher of US_Invert and DS_Invert is used to set the structure crest or invert. **SP Channel Type:** Sets the spillway crest	
8	DS_Invert	**M Channel Type:** Invert level of channel is taken as the maximum of US_Invert and DS_Invert. **Q Channel Type:** Used to set to the invert of the joining channel downstream, otherwise not used. **SG Channel Type:** See comments above for US_Invert. **SP Channel Type:** Sets the level of the gate seat (if SP is operated, ie. a SPO channel).	Float
9	Form_Loss	**M, P, Q, SP Channel Type:** Not used. **SG Channel Type:** If no weir is specified in the Type attribute, it is assumed that the gate seats on to the bottom of the channel. For this case the flow calculations where the gate is not surcharged uses the Form_Loss attribute to apply an energy loss to the structure to represent contraction/expansion losses.	Float
10	pBlockage	**Q Channel Type:** Not used. **M Channel Type:** Percentage blockage of channel (only used if no flow area matrix is provided). Refer to Inlet_Type attribute below. **P, SG, SP Channel Type:** The channel's dimensions are reduced as per the pBlockage value.	Float
11	Inlet_Type	**M Channel Type:** The relative path to the flow matrix file (must be a .csv file), and optionally flow area matrix file. See Section 5.8.1. **P, SG, SP Channel Type:** Only used to specify the name of the <control_id> for structures controlled using operating rules (ie. type "PO", "RGO", SGO" or "SPO"). The <control_id> is referenced within the .toc file and specifies which rules are used to operate the channel. Refer to Section 5.8. **Q Channel Type (for QO see further below):** For non-operated Q channels the name of depth-discharge curve in a depth discharge database (identical approach to inlet Q pits). A Q channel is distinguished from a Q pit, in that a Q channel is a polyline in the 1d_nwk layer and a Q pit is a point object. A depth-discharge	Char (max 256)

		database is the same as a pit inlet database as used for Q pits, and the same database file can be used for both Q channels and Q pits. The new .ecf command Depth Discharge Database and the existing command Pit Inlet Database perform the same function. Note that the depth discharge curve is depth (not water elevation) and the depth is taken as the upstream water level less the Q channel's invert. **QO Channel Type:** For operated Q channels the filename of a .csv file containing a flow matrix table in the same format as used for M channels. See Section 5.8.1.	
12	Conn_1D_2D	Not used.	Char
13	Conn_No	Not used.	Integer
14	Width_or_Dia	**P Channel Type:** The diameter of the pump offtake. **M Channel Type:** Width of channel (only used if no flow area matrix is provided). Refer to Inlet_Type attribute above. **Q Channel Type (for QO see further below):** A multiplier applied to the flow interpolated from the depth-discharge curve. Particularly useful if the discharge values are per unit width (ie. m^2/s or ft^2/s). If set to zero (0), the multiplier is set to one (1). **QO Channel Type:** Used to determine the flow area at different stages by multiplying by the upstream depth. The area is used to determine the velocity that is output, otherwise, it does not play a role in the hydraulic calculations. **SG, SP Channel Type:** The width of the gate/spillway. The flow area is assumed to be rectangular in shape.	Float
15	Height_or_WF	**M, Q Channel Type:** Not used. **P Channel Type:** Can be used to set the pump flow capacity for operated pumps (ie. a PO channel). The pump capacity can also be set and changed during the simulation using the operating control commands. Useful where the operating control definition is generic, ie. pump control is used for more than one pump of different flow capacities. **SG Channel Type:** The vertical height of the gate in its fully open position. If an underlying weir is specified, the weir is assigned a coefficient adjustment factor of 1.0, which cannot be modified using this attribute. **SPO Channel Type:** For an operated (gated) spillway (SPO), sets the vertical height of the gate in its fully open position. The fully open height can also be changed	Float

		during the simulation using the operating control commands (this is useful where the operating control definition is generic, ie. non-structure specific). Not used if channel is non-operational, ie. a SP channel.	
16	Number_of	**P, SG, SP Channel Type:** The number of parallel channels. If set to zero, one barrel is assumed. **M, Q Channel Type:** Number of parallel channels (flow and area matrices are multiplied by this value). If zero is set to one.	Integer
17	HConF_or_WC	**M, Q Channel Type:** Not used. **SP Channel Type:** Weir flow coefficient, C_d, in its dimensionless form as in the equation below. If less than or equal to zero the default value for spillways in Table 5-8 is used. Entering a value greater than zero (0) will override this and apply a fixed C_d. Note that published coefficients may be based on other non-dimensional or dimensional forms of the equation below, therefore care should be taken in ensuring the coefficient is converted to comply with the form below if this is the case. $$Q = \frac{2}{3} C_{sf}\ C_d\ B\ \sqrt{2g}\ H^{Ex}$$ **RG, SG Channel Type**: As for the weir channel in Table 5-9 if a weir has been specified (eg. a "SG WB" channel).	Float
18	WConF_or_WEx	**M, Q Channel Type:** Not used. **SP Channel Type:** Weir flow equation exponent Ex in the equation for HConF_or_WC. If less than or equal to zero the default value for spillways in Table 5-8 is used, ie. 1.5. **RG, SG Channel Type:** As for the weir channel in Table 5-9 if a weir has been specified (eg. a "SG WB" channel).	Float
19	EntryC_or_WSa	**M, Q Channel Type:** Not used. **SP Channel Type:** Sets the submergence factor "a" exponent in the equation below for calculating the submergence factor C_{sf}. If less than or equal to zero the default value for spillways in Table 5-8 is used. $$C_{sf} = \left(1 - \left({H_d}/{H_u}\right)^a\right)^b$$	Float

		RG, SG Channel Type: As for the weir channel in Table 5-9 if a weir has been specified (eg. a "SG WB" channel).	
20	ExitC_or_WSb	**M, Q Channel Type:** Not used. **SP Channel Type:** Sets the submergence factor "b" exponent in the weir flow equation above for calculating the weir submergence factor C_{sf}. If less than or equal to zero the default value for spillways in Table 5-8 is used. **RG, SG Channel Type:** As for the weir channel in Table 5-9 if a weir has been specified (eg. a "SG WB" channel).	Float

5.9 Operational Channels

Gated rectangular culverts, pumps, sluice gates, gated spillways, weirs and Q channels can be operated using logical scripts. An "O" Type flag is required within the 1d_nwk layer for structures that are to be operated using an operating control definition. For example, an operated pump would have a Type attribute of "PO" or "OP".

Operating rules are contained within a .toc file (**T**UFLOW **O**perations **C**ontrol) with each set of rules contained within a control definition. More than one structure/device can use the same control definition. The .toc file is referenced using Read Operating Controls File via the .ecf file, or via the .tcf file within a Start 1D Domain block or by preceding the command with "1D".

The operating rules for a control can only occur within a .toc file. More than one .toc file can be set up and accessed should there be a need to break the control definitions into several files (for example, all pump controls could be placed in one file and sluice gate controls in another).

Operational structure time-series data is output to the _1d_O.csv file. The file reports the time varying status of the structure and the resulting flow rate. Values of user defined variables and other information are also output to this file. This is discussed in greater detail within section 13.2.2.

5.9.1 .toc File Commands and Logic

5.9.1.1 Define Control Command

A .toc file can only contain Define Control and End Define blocks.

Each Define Control must include a keyword indicating the type of structure/device as per below:

```
Define [ Culvert | Pump | Q Channel | Sluice | Spillway | Weir ] Control == <control_id>
    ….
End Define
```

Where <control_id> is a unique control definition name, noting that more than one structure can refer to the same operational control definition. For example, several pumps of different capacities may utilise the same operational control logic. The 1d_nwk Inlet_Type attribute is used to link a control definition with the structure.

Within the control definition, commands specific to the type of structure/device can be used to adjust the structure/device's operation. The commands available for each type of control are described below.

Each Define Control block consists of three sections:

4. The default settings for the control's commands. These are usually placed at the top of the definition and prior to the logical rules. These default settings are used at the start of the simulation/operation and during the operation unless changed by the logical rules.
5. User defined variables as described further below.
6. One or more logical rules as described further below.

An example of a definition control with the three sections is provided below.

```
!_____________________________________________________________
Define Pump Control == P_8_to_1          ! Define the set of commands for "P_8_to_1"
    ! Default Settings
    Pump Operation == OFF                ! Set the pump to off

    ! Set User Variables
    t == TIME 24h                        ! Sets the variable 't' to the simulation time on a 24hr clock
    wday == DAY of WEEK                  ! Sets the variable 'wday' to a day of the week

    ! Provide Logic
    IF wday >= MON AND wday < =FRI       ! If the day of the week is Mon - Fri
        IF t > 6 AND t < 18              ! If the simulation time is between 0600 and 1800hrs
            Pump Capacity == 0.1         ! Sets the pump flow to a constant 0.1 m3/s
            Pump Operation == ON         ! Sets the pump to on
        ELSE
            Pump Operation == OFF        ! Set the pump to off
        END IF

    ELSE
        IF t > 10 AND t < 16             ! If the simulation time is between 1000 and 1600hrs
            Pump Capacity == 0.1         ! Sets the pump flow to a constant 0.1 m3/s
            Pump Operation == ON         ! Sets the pump to on
        ELSE
            Pump Operation == OFF        ! Set the pump to off
        END IF
    END IF
End Define
```

5.9.1.2 User Defined Variables

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

```
<variable> == <variable_value>
```

Where <variable> is the variable name, while <variable_value> must conform to one of the options in Table 5-12. Any characters can be used for the variable name, but it is recommended to use only letters, numbers and underscores and to avoid spaces. Note that variables can be redefined at any point within the control definition. Also note that these variables only apply to the control. For global variables, use the Set Variable feature.

Table 5-12 Variable Value Types

Variable Value	Description
constant	Sets the variable to the value of <constant>. Must be a number.
Time of Model	Sets the variable to the simulation time in hours.
Time in 24H	Sets the variable to the simulation time in hours on a 24 hour clock (ie. will always be between 0 and 24 hours). A simulation time of zero is equivalent to midnight.
Day of Week	Sets the variable to the day of the week where Sunday is 1 and Saturday is 7. The keywords “Sun”, “Mon”, “Tue”, “Wed”, “Thu”, “Fri” and “Sat” can also be used when using the variable in a logic rule.

Variable Value	Description
Period No Change	Sets the variable to the time in hours since there was last a change in operation.
H1D <node_id>	Sets the variable to the water level at the 1D node named <node_id>.
Q1D <node_id>	Sets the variable to the flow in the 1D channel named <channel_id>.
H2D <x>,<y>	Sets the variable to the water level at the 2D cell located at the XY coordinates <x>,<y>
H2D <2d_po_ID>	Sets the variable to the water level at the 2D plot output location given by the plot output ID.
HU	Sets the variable to the 1D water level at the upstream node of the channel, based on the digitised direction of the channel.
HD	Sets the variable to the 1D water level at the downstream node of the channel based on the digitised direction of the channel.
dHUD	Sets the variable to the difference in water level between the upstream and downstream nodes based on the digitised direction of the channel. Will be negative if flow is in opposite direction to digitised direction.
H1	Sets the variable to the upstream water level of the channel based on the flow direction.
H2	Sets the variable to the downstream water level of the channel based on the flow direction.
dH12	Sets the variable to the difference in water level between the upstream and downstream nodes based on the flow direction of the channel. Will always be positive.
YU	Sets the variable to the depth above the structure invert of the upstream node based on the digitised direction of the channel.
YD	Sets the variable to the depth above the structure invert of the downstream node based on the digitised direction of the channel.
Y1	Sets the variable to the upstream depth relative to the structure invert based on the flow direction.
Y2	Sets the variable to the downstream depth relative to the structure invert based on the flow direction.

5.9.1.3 Logic Rules

The logic rules consist of using If…End If blocks using the construct below.

```
If <condition_1> [ [ and | or ] <condition_2> ]  [ [ and | or ] <condition_3> ]….
      …enter one or more commands or variable definition lines
[ Else If  <condition_1> [ [ and | or ] <condition_2> ]  [ [ and | or ] <condition_3>
]….
      …enter one or more commands or variable definition lines]
[ Else If…repeat as needed]
      …enter one or more commands or variable definition lines]
[ Else
      …enter one or more commands or variable definition lines]
End If
```

<condition_1> must be a conditional operation that includes one of the symbols "=", ">", ">=", "<" or "<=". The left and right sides of the condition must be a single value or a variable name. Optionally the variable may be operated on by the following:

- A "+", "-", "*" or "/" and a constant value. For example, a condition could be "x + 2 < 3"; or
- Specifying "HIGHER" or "LOWER" to compare the current value of the variable to its value at the start of the current period of no change in operation. For example "x == LOWER" will be true if the current value of 'x' is less than its value at the last time there was a change in operation.

If more than one condition is to be applied, the conditions must be separated by either an "and" or "or". <condition_2>, <condition_3>, etc. have the same format as for <condition_1> above.

If…End If blocks can be nested inside other If…End If blocks. Indenting is strongly recommended to make the control file easier to read.

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

```
!_____________________________________________________________
Define Pump Control == P_8_to_1          ! Define the set of commands for "P_8_to_1"
    ! Default Settings
    Pump Operation == OFF                ! Set the pump to off

    ! Set User Variables
    t == TIME 24h                        ! Sets the variable 't' to the simulation time on a 24hr clock
    wday == DAY of WEEK                  ! Sets the variable 'wday' to a day of the week

    ! Provide Logic
    IF wday >= MON AND wday < =FRI       ! If the day of the week is Mon - Fri
        IF t > 6 AND t < 18              ! If the simulation time is between 0600 and 1800hrs
            Pump Capacity == 0.1         ! Sets the pump flow to a constant 0.1 m3/s
            Pump Operation == ON         ! Sets the pump to on
        ELSE
            Pump Operation == OFF        ! Set the pump to off
        END IF

    ELSE
        IF t > 10 AND t < 16             ! If the simulation time is between 1000 and 1600hrs
            Pump Capacity == 0.1         ! Sets the pump flow to a constant 0.1 m3/s
            Pump Operation == ON         ! Sets the pump to on
        ELSE
            Pump Operation == OFF        ! Set the pump to off
        END IF
    END IF
End Define
```

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

```
!_____________________________________________________________
Define Pump Control == PG_Pump          ! Define the set of commands for "PG_Pump"
    ! Default Settings
    Pump Capacity == 19.3               ! Sets to flow capacity of the pump to 19.3m3/s
    Pump Operation == OFF               ! Set the pump to off
    Period Startup/Shutdown (min) == 10 ! Set a 10min period in which the pump takes to startup/shutdown

    ! Set User Variables
    h_Reservoir == H2D 275300, 286250   ! Set the variable to the water level at the 2D cell located
                                        ! at the specified coordinates

    ! Provide Logic
    IF h_Reservoir > 337.4              ! When the water level at the 2D cell exceeds 337.4mAHD...
        Pump Operation == ON            ! Set the pump to on
    ELSE IF h_Reservoir < 337.3         ! When the water level at the 2D cell is lower than 337.3mAHD...
        PUMP Operation == OFF           ! Set the pump to off
    ELSE                                ! For all other situations...
        Pump Operation == NO CHANGE     ! Make no change to the operation of the pump
    END IF

End Define

!_____________________________________________________________
Define Q Channel Control == Q_Release                       ! Define the set of commands for "Q_Release"

    ! Default Settings
    Period Opening/Closing (min) == 600                     ! Sets a 600min period in which the gate takes to open/close
    Gate Opening == CLOSE                                   ! Sets the gate to closed

    ! Set User Variables
    h_Reservoir == H2D 275300, 286250                       ! Set the variable to the water level at the 2D cell located
                                                            ! at the specified coordinates
    period_no_change == PERIOD NO CHANGE                    ! Define a variable for 'PERIOD NO CHANGE'
    wait_period_rise == 1                                   ! Sets the variable 'wait_period_rise' to a constant of 1
    wait_period_fall == 1                                   ! Sets the variable 'wait_period_fall' to a constant of 1

    ! Provide Logic
    IF period_no_change < wait_period_rise                  ! If the last time there was a change in operation of the
                                                            ! gate is less than the constant for variable 'wait_period_rise'
        Gate Opening == NO CHANGE                           ! Make no change to the operation of the gate
    ELSE IF h_Reservoir > 337.5 AND h_Reservoir == HIGHER   If the water level at the 2D cell is greater than 337.5mAHD and 
        Gate Opening == ++10                                ! Increase the gate opening by 10m
        wait_period_rise == 1                               ! Set the variable 'wait_period_rise' to a constant of 1
        wait_period_fall == 3                               ! Set the variable 'wait_period_fall' to a constant of 3
    ELSE IF period_no_change < wait_period_fall             ! If the last time there was a change in operation of the
                                                            ! gate is less than the constant for variable 'wait_period_fall'
        Gate Opening == NO CHANGE                           ! Make no change to the operation of the gate
    ELSE
        Gate Opening == --10                                ! Decrease the gate opening by 10m
        wait_period_rise == 1                               ! Sets the variable 'wait_period_rise' to a constant of 1
        wait_period_fall == 1                               ! Sets the variable 'wait_period_fall' to a constant of 1
    END IF

End Define
```

5.9.2 Types of Operational Structures

5.9.2.1 Pumps (P and PO)

Pumps can be modelled as a "P" or "PO" type channel. In non-operational mode (P channel), the pump flow is interpolated from a head discharge curve in the Depth Discharge Database defined via a head difference versus flow relationship – see Section 0. In operational mode, PO, the pump flow can be varied using functions such as: switching on and off over a start up and shutdown period; and changing the pump capacity and/or discharge curve according to time, day of the week, hydraulic conditions and other variables. Pumps do not contribute to any model storage.

In the 1d_nwk layer, the following attributes can be used to set up the pump (also see Table 5-11).

- ID = Unique Channel ID.
- Type = "P" or "PO".
- US_Invert = Intake elevation of the pump.
- DS_Invert = Outlet elevation of the receptor.
- Inlet_Type = The pump discharge curve in the Depth Discharge Database. This curve is a head difference versus discharge curve, therefore, for a pump this curve would usually have greater flows for smaller head differences. If the head difference is negative (ie. the receptor water level is below the intake water level) the discharge used is that for a zero head difference.
- Width_or_Dia = Diameter of the pump's outlet pipe/hose. Used to trigger dry conditions (see below) and for calculating the velocity.
- Height_or_WF = For PO pump the initial operating pump capacity (subject to not being overridden by an operational control command).
- Number_of = Number of pumps.

P and PO pumps are simulated as dry (zero flow) if the upstream node is dry or the upstream water level is below the intake elevation plus the output pipe diameter, ie the upstream (intake) water level is below US_Invert + Width_or_Dia.

P pumps always produce a flow in the direction the P channel is digitised based on that interpolated from the pump discharge curve; the exception being when dry as described above.

PO pumps are typically operated on a time basis or based on hydraulic conditions elsewhere in the model. Operational control commands specific to the Define Pump Control command are provided below. Subject to not being overridden by an operational control command, PO pumps are assumed to be OFF at the start of the simulation.

- Pump Operation turns the pump on or off.
- Period Startup/Shutdown sets the time taken to start the pump up or shut it down.
- Pump Capacity sets the flow capacity of the pump. It is possible to set a constant flow rate or a head-discharge curve referenced within the Depth Discharge Database.
- Pump Number sets the number of pumps in parallel.

5.9.2.2 QO Channels

For QO channels, depth discharge curves for different structure openings is used to vary and control the discharge. The relationships are contained in a csv file in a similar format as used for M channels (see Section 5.8.1). The vertical axis is the depth above the channel's invert and the horizontal axis is the percentage opening as shown in the example below. In the example, Column A contains the depth above invert values and Row 2 the % opening values. Note that the value in cell A2 is a flow multiplier. If this value is empty, negative or set to zero (0), a multiplier of one (1) is used. The flow through the channel is interpolated from this table at each timestep based on the structure's opening and upstream depth above the invert.

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

In the 1d_nwk layer, the following attributes can be used to set up a QO channel.

- ID = Unique Channel ID.
- Type = "QO".
- US_Invert = Elevation corresponding to zero depth in the depth-discharge curve.
- Inlet_Type = The .csv file containing the matrix of flows for different openings and upstream water level as per discussion above.
- Width_or_Dia = Not used other than to estimate the velocity and contribute to nodal storage.

The following commands are specific to the Define Q Channel Control command used for QO channels.

- Gate Speed sets the speed at which the gate moves.
- Period Opening/Closing sets the time taken to transition from zero to 100% opening or vice versa.
- Gate Opening sets the position the opening to be operated towards. This can be specified incrementally or as an absolute value.

5.9.2.3 Gated Drowned Rectangular Culverts (RO)

Rectangular culverts with a gate on the exit can be operated using the .toc commands below for a Define Culvert Control block. The command is applicable for operated rectangular (RO) culverts only, and uses the equation below as implemented into TUFLOW for a project based in Florida. The equation

below is for UK Customary (English) Units, but RO culverts can be used in metric or UK Customary units. Note that the equation is for submerged culverts only, therefore the user must ensure that the culvert is drowned at all times.

$$Q = C_d A_0 \sqrt{\frac{2g(H-h)}{\left(\frac{A_0}{A_G}\right)^2 + 2C_d^{\,2}\left(1 - \frac{A_0}{A_G} + \frac{gn^2 L}{1.49^2 R_0^{\,4/3}}\right)}}$$

Criteria: $TW \geq Hc$ and $HW \geq$ Hc

where Q = discharge (cfs);
C_d = discharge coefficient;
A_0 = area of the culvert flowing full (ft^2);
A_G = the flow area underneath of the opening gate (ft^2);
g = acceleration due to gravity (32.2 ft^2/s);
H = head water stage (ft);
h = tail water stage (ft);
n = Manning friction coefficient;
Hc = Height of the box culvert
L = length of the box culvert (ft);
R_0 = hydraulic radius of the full pipe (ft);
HW = H – culvert invert elevation at head water side (ft);
TW = h – culvert invert elevation at tailwater side (ft);
Go = gate opening (ft);
Hg = gate height (ft).

Discharge coefficient of 0.75 is used for STA 3/4 culverts.

Gate Type sets the type of gate arrangement.

Gate Speed sets the speed at which the gate moves.

Period Opening/Closing sets the time taken to fully open a closed gate or to fully close an open gate.

Gate Height Fully Open sets the height (not elevation) of the gate when fully open above the gate's seat for vertically moving gate. If not set, the 1d_nwk "Height" attribute is used.

Gate Width Fully Open sets the width of the gate(s) when fully open for horizontally moving gates. If not set, the 1d_nwk "Width_or_Dia" attribute is used.

Gate Opening sets the position the gate is to be operated towards. This can be specified incrementally or as an absolute value.

Cd sets the discharge coefficient C_d in the equation above.

5.9.2.4 Sluice Gates (SG and SGO)

Sluice gates can be operated using the .toc commands further below for a Define Sluice Control block. The approach to calculating the flow through the gate is based on that documented in the HEC-RAS 4.0

Reference Manual as described below. For non-operated sluice gates (SG) the gate is assumed to be in a fixed position based on the 1d_nwk Height_or_WF attribute value.

For a free flowing sluice gate, ie. upstream controlled Q is calculated using:

$$Q = C_d W B \sqrt{2gH_1}$$

Where

Q = Discharge
C_d = Discharge coefficient upstream controlled flow (default = 0.6)
W = Width
B = Height of gate opening above crest level
H_1 = Upstream energy level – Crest level

For downstream controlled flow:

$$Q = C_s W B \sqrt{2g\Delta H}$$

Where

C_s = Submerged discharge coefficient (default = 0.8)
ΔH = Upstream energy level – Downstream level

Transition between upstream controlled and full submergence downstream controlled flow:

$$Q = C_d W B \sqrt{2g3\Delta H}$$

The transition between downstream and upstream controlled flow equations is based on the degree of submergence calculated as the tailwater depth above the spill crest divided by the upstream energy depth. For a ratio below 0.67 upstream controlled flow applies, above 0.8 downstream controlled flow and in between the transition equation applies.

Note that by default the energy level is used for calculating H_1 and ΔH, however, this can be changed globally to water surface level using Structure Flow Levels == WATER or by using the "E" or "H" optional flag for the Type attribute (see Table 5-1).

When the flow is not in contact with the gate one of the following options apply:

- One of the advanced weir types (see Section 5.7.3.3) can be specified for the sill by adding the weir type to the 1d_nwk Type attribute (eg. "SGWB" or "SG WB" are accepted). The weir equation for this weir type is applied when the gate is not controlling the flow. It is recommended that one of the rectangular weir shapes is used (ie. WB, WD, WO or WR).
- If no weir is associated with the SG channel, the flow is calculated using a zero length rectangular channel with adjusted entrance and exit losses as per a zero length culvert.

In the 1d_nwk layer, the following attributes can be used to set up the SG or SGO channel.

- ID = Unique Channel ID.

- Type = “SG” or “SGO”.
- US_Invert and DS_Invert: The higher value is used for the sill crest.
- Width_or_Dia = Width of the gate.
- Height_or_WF = The height above the sill crest of the gate when fully opened (subject to not being overridden by an operational control command).
- Number_of = Number of (identical) parallel gates.

The following lists commands specific to the Define Sluice Control commands.

- Gate Speed sets the speed at which the gate moves.
- Period Opening/Closing sets the time taken to fully open a closed gate or to fully close an open gate.
- Gate Height Fully Open sets the height (not elevation) of the gate when fully open above the gate’s seat for vertically moving gate. If not set, the 1d_nwk “Height” attribute is used.
- Gate Opening sets the position the gate is to be operated towards. This can be specified incrementally or as an absolute value.
- Cd Gate sets the discharge coefficient of the gate, C_d.
- Cd Gate Submerged sets the submerged discharge coefficient, C_s.

5.9.2.5 Spillways with Gates (SPO)

Gated spillways can be operated using the approach documented in the USACE Hydraulic Design Criteria Sheet 312 for Vertical Lift Gates on Spillways.

For flow over the spillway unaffected by a gate the following equation applies:

$$Q = \frac{2}{3} C_d W H \sqrt{2gH}$$

Where Q = Discharge
C_d = Discharge coefficient (default = 0.75)
W = Width of the spillway (rectangular cross-section assumed)
H = Upstream energy level – Crest level

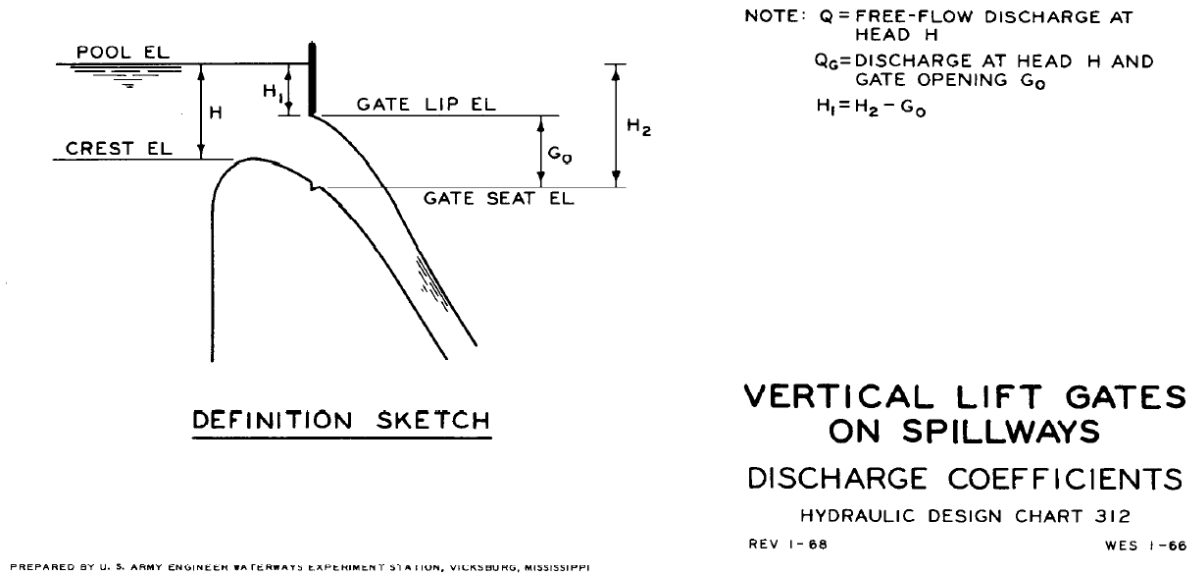
NOTE: C_d prior to the 2016-03 release was based on $Q = C_d W H \sqrt{2gH}$ (as per Sheet 312), with a default C_d value of 0.5. As of the 2016-03 release, SPO channels now use the same formula as SP and weir channels and use a default value of 0.75.

The ratio of the gated discharge to the ungated discharge is derived as:

$$\frac{Q_G}{Q} = \frac{C_G}{C_d}\left(\frac{H_2^{\frac{3}{2}} - H_1^{\frac{3}{2}}}{H^{\frac{3}{2}}}\right)$$

Where C_G = Discharge coefficient (default = C_d)
$H_1 and\ H_2$ = See diagram below from Sheet 312

If *QG* is less than *Q*, *QG* is used for the flow through the structure. The structure is also tested for submergence using the same setting as for Ogee Weirs (see Section 5.7.3.4).



Note that by default the energy level is used for calculating the *H* values, however, this can be changed globally to water surface level using Structure Flow Levels == WATER or by using the "E" or "H" optional flag for the Type attribute (see Table 5-1).

In the 1d_nwk layer, the following attributes can be used to set up a SPO channel. Note that the 1d_nwk values can be overridden by their equivalent operational control command.

- ID = Unique Channel ID.
- Type = "SPO".
- US_Invert = Spillway crest level.
- DS_Invert = Gate seat level (see GATE SEAT EL in diagram from Sheet 312). If DS_Invert is higher than US_Invert an ERROR 1050 results.
- Width_or_Dia = Width of the gate.
- Height_or_WF = The height of the gate above the gate seat when fully opened.
- Number_of = Number of (identical) parallel spillways.

The following section lists commands specific to the Define Spillway Control commands.

Gate Speed sets the speed at which the gate moves.

Period Opening/Closing sets the time taken to fully open a closed gate or to fully close an open gate.

Gate Height Fully Open sets the height (not elevation) of the gate when fully open above the gate's seat for vertically moving gate. If not set, the 1d_nwk "Height" attribute is used.

Gate Seat Vertical Offset sets the difference in height between the spillway crest and the seat of the gate, ie. the CREST EL minus GATE SEAT EL in the diagram above from Sheet 312.

Gate Opening sets the position the gate is to be operated towards. This can be specified incrementally or as an absolute value. This value is G_O in the diagram above from Sheet 312.

Cd Spillway sets the discharge coefficient of the spillway, C_d. Default value is 0.75.

Cd Gate sets the discharge coefficient of the gate, C_G. By default assumed to be the same as Cd Spillway see Sheet 312.

5.9.2.6 Weirs (WBO, WCO, WDO, WOO, WRO, WTO)

Weirs can be operated to simulate structures such as fabri (inflatable) dams for the WB, WC, WD, WO, WR and WT weir types.

In addition to the 1d_nwk attributes for non-operated weirs (see Section 5.7.3.3) the following criteria is used to set the limiting dimensions via the 1d_nwk attributes as follows:

- Width_or_Dia = The width of the weir when fully open, or if the width remains unchanged throughout.
- Height_or_WF = The height of the weir when fully up. The invert of the weir (when fully down) is set by the maximum of the US_Invert and DS_Invert attributes. Note that for operational weirs this attribute cannot be used to set the Weir Calibration Factor (a value of 1.0 is used).

The following are the commands specific to Define Weir Control blocks for operating weirs.

- Weir Height sets the height above the weir crest to operate towards.
- Weir Width sets the width of the weir to operate towards.
- Weir Height Speed sets the speed of the weir in the vertical.
- Weir Width Speed sets the speed of the weir in the horizontal.
- The generic commands Operation and "Period Opening/Closing ==" also apply.

5.10 Cross-Sections

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 Flood Modeller .pro files.

Cross-section profile and hydraulic properties data are accessed using a 1d_xs GIS layer and the .ecf command, Read GIS Table Links. Type "XZ" is specified if accessing a cross-section profile (distance versus elevation) and a type "CS" or "HW" is used if accessing a hydraulic properties table (elevation versus width).

Table 5-14 presents the attributes required for the 1d_xs GIS layer. A number of optional flags are available for both "XZ" and "CS" or "HW" and are explained in more detail in Section 0.

External sources are defined using the XS Database and M11 Network commands. Cross-sections are extracted using the 1d_nwk attributes Inlet_Type, Conn_1D_2D and Conn_No as described in Table 5-2. The hydraulic properties table is automatically created from the external source. The conversion from these sources preserves all the hydraulic properties listed in Table 5-13 including any vertical variation in bed resistance (Manning's n) values.

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 allow the water level to exceed ten times the depth of the CS or NA table before an instability is triggered. See Depth Limit Factor for further details.

Table 5-13 Channel Cross-Section Hydraulic Properties from External Sources

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.

Table 5-14 1D Cross-Section Table Link (1d_xs) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
1	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(50)
2	Type	Two characters defining the type of table link. **"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. **"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	Char(2)
3	Flags	Optional flags are as follows:	Char(8)

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
		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 for a Manning's n value. **"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 5.10.1.1.4. **"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. **"E" or "T":** Specify an "E" to use effective area or a "T" to use total area when calculating the flow area (see Section 5.10.3 and Flow Area). If neither is specified, the global value set using Flow Area is used. **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. **"E":** Effective flow width (Column 6)	
4	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(20)
5	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(20)
6	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(20)
7	Column_4	Optional. Defines the fourth column of data.	Char(20)
8	Column_5	Optional. Defines the fifth column of data.	Char(20)
9	Column_6	Optional. Defines the sixth column of data.	Char(20)

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
10	Z_Increment	Optional. Sets the height increment in metres 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
11	Z_Maximum	Optional. Sets the maximum elevation in metres 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
12	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 √2.	Float

5.10.1 Type "XZ" Optional Flags

5.10.1.1 Relative Resistances

Varying the resistance across an XZ (offset-elevation) cross-section is possible by using either a relative resistance factor (R flag), different material ID values (M flag) or different Manning's n values (N 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). The reason for this is that material or n values can be correctly sampled from a GIS layer at the survey points. 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. This feature is particularly useful when developing a linked 1D/2D model where the 1D cross-sections are typically trimmed to the top of bank to avoid double counting of floodplain storage.

5.10.1.1.1 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 5.10.4), 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.

5.10.1.1.2 Material Values (M)

If using material values (M), the Manning's n value to be applied is taken from a Materials Definition File (see Read Materials File). If the Position "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 5.10.1.1.3).

When using materials, the n_nF_Cd 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

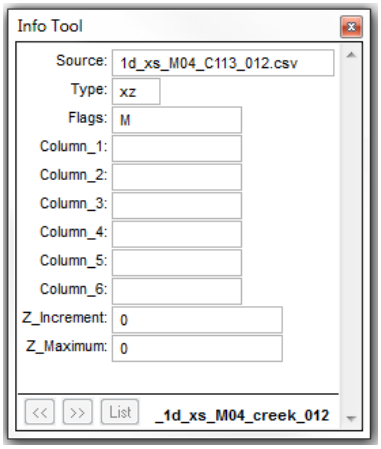
n_nF_Cd value in the 1d_nwk layer to, say, 1.1 to increase all Manning's n values across the cross-section by 10%.

A 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. The negative value must be the negative of a valid Material ID in the Materials Definition File (see Read Materials File and Section 6.9).

Example

In the following example, the TUFLOW Tutorial Model (discussed in Section 2.4) will be modified to demonstrate how Manning's n values may be assigned to cross-sections based on values within the Materials Definition File.

An "M" flag is added to the 1d_xs layer referencing the cross-sectional data of the open channel.



An additional third column is then added to the .csv source file, containing one or more Material ID values from within the Materials Definition File. In the figure below, a Material ID of 10 has been assigned to the whole cross-section.

	A	B	C
1	Cross-section written from SMS		
2	X	Z	M
3	0	41.86	10
4	0.660668	41.8524	10
5	1.54971	41.7583	10
6	6.59121	41.1738	10
7	6.74283	41.1564	10
8	9.27349	40.8554	10
9	9.27492	40.8552	10
10	9.55985	40.6679	10
11	9.56812	40.6635	10
12	9.56985	40.6623	10
13	11.7945	38.9385	10
14	11.7945	38.9385	10
15	13.2116	38.9397	10
16	13.218	38.9397	10

This correlates to a Manning's n value of 0.08 as shown in the Materials Definition File.

	A	B	C
1	Material ID	Manning's n	Infiltration Parameters
2	1	0.06	
3	2	0.022	
4	3	3	
5	4	0.03	
6	10	0.08	
7			

When using the “M” flag to define material values, the “n_nF_Cd” attribute in the 1d_nwk becomes a multiplier (refer to Table 5-2). In most cases, this should be set to 1, as has been carried out for this example.

Once the model has been compiled, a check of the Manning’s n values applied to each cross-section may be viewed in the _ta_tables_check.csv (refer to Table 12-2).

Section XS00015 [xzM from Cols 1,2,3 in C:\TUTORIAL\Test_Models\TUFLOW\model\xs\1d_xs_M04_C113_012.csv]													
Point	Distance	Elevation	Manning n		Elevation	Depth	Width	Eff Width	Eff Area	Eff Wet Pe	Radius	Vert Res F	K (n=0.080)
1	0	41.86	0.08		38.938	0	0.001	0.001	0	0.001	0	1	0
2	0.661	41.852	0.08		39	0.062	4.825	4.825	0.28	4.795	0.059	1	0.5
3	1.55	41.758	0.08		39.1	0.162	5.119	5.119	0.78	5.045	0.154	1	2.8
4	6.591	41.174	0.08		39.2	0.262	5.413	5.413	1.31	5.297	0.246	1	6.4
5	6.743	41.156	0.08		39.3	0.362	5.707	5.707	1.86	5.552	0.335	1	11.2
6	9.273	40.855	0.08		39.4	0.462	6.001	6.001	2.45	5.81	0.421	1	17.2
7	9.275	40.855	0.08		39.5	0.562	6.295	6.295	3.06	6.071	0.504	1	24.2
8	9.56	40.668	0.08		39.6	0.662	6.589	6.589	3.71	6.299	0.588	1	32.5
9	9.568	40.663	0.08		39.7	0.762	6.883	6.883	4.38	6.53	0.671	1	41.9
10	9.57	40.662	0.08		39.8	0.862	7.177	7.177	5.08	6.772	0.75	1	52.5
Bed	11.794	38.938	0.08		39.9	0.962	7.471	7.471	5.81	6.984	0.833	1	64.3
Bed	11.794	38.938	0.08		40	1.062	7.765	7.765	6.58	7.209	0.912	1	77.3
13	13.212	38.94	0.08		40.1	1.162	8.059	8.059	7.37	7.444	0.99	1	91.5
14	13.218	38.94	0.08		40.2	1.262	8.331	8.331	8.19	7.668	1.068	1	106.9
15	13.961	38.94	0.08		40.3	1.362	8.602	8.602	9.03	7.895	1.144	1	123.5
16	16.444	38.943	0.08		40.4	1.462	8.874	8.874	9.91	8.125	1.219	1	141.4
17	16.445	38.943	0.08		40.5	1.562	9.145	9.145	10.81	8.355	1.294	1	160.4
18	18.362	40.104	0.08		40.6	1.662	9.416	9.416	11.74	8.589	1.366	1	180.7
19	18.364	40.106	0.08		40.7	1.762	10.192	10.192	12.7	8.838	1.437	1	202.3
20	19.137	40.65	0.08		40.8	1.862	11.526	11.526	13.79	9.216	1.496	1	225.5
21	19.142	40.653	0.08		40.9	1.962	13.168	13.168	15.02	9.728	1.544	1	250.7
22	19.148	40.653	0.08		41	2.062	15.215	15.215	16.43	10.432	1.575	1	278.1
23	22.566	40.942	0.08		41.1	2.162	17.28	17.28	18.06	11.314	1.596	1	308.3
24	28.302	41.411	0.08		41.2	2.262	19.356	19.356	19.89	12.346	1.611	1	341.7
					41.3	2.362	21.442	21.442	21.93	13.489	1.626	1	379

5.10.1.1.3 Manning’s n Values (N)

If using Manning’s n values (N), the n value is specified directly, noting that the n_or_n_Cd 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.

5.10.1.1.4 Position Flag (P)

The position values are used to indicate whether an XZ point is left bank (1), mainstream (2) or right bank (3). 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.

5.10.2 Type "HW" Optional Flags

5.10.2.1 Flow Area (A)

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.

5.10.2.2 Wetted Perimeter (P)

The wetted perimeter in metres. If omitted, the area is calculated based on the elevations and widths assuming a symmetrical channel.

5.10.3 Parallel Channel Analysis

To calculate total conveyance, a cross-section needs to be sub-divided into panels for which the velocity is uniformly distributed. Conveyance for each panel is calculated using the Manning's equation:

$$K = \frac{1.0}{n} AR^{2/3}$$

Where:

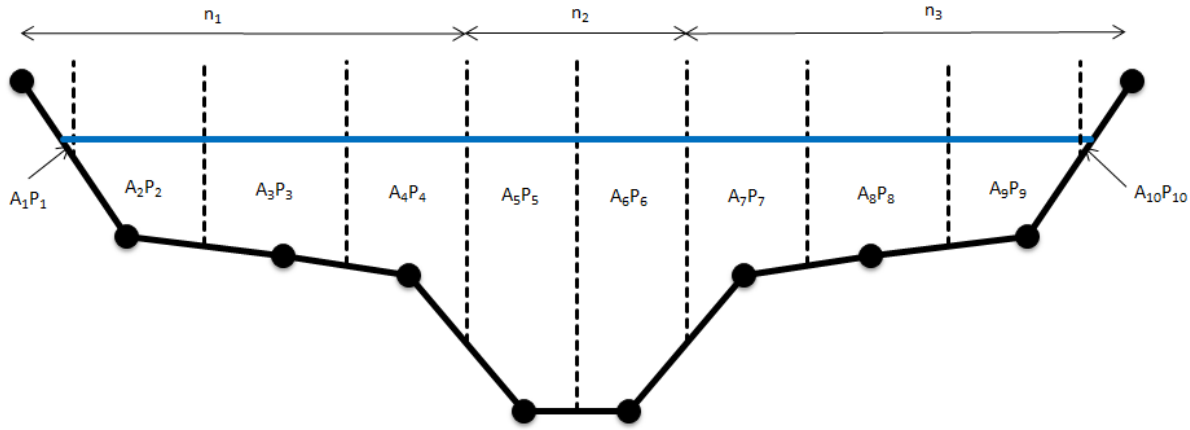
K = conveyance of panel
n = Manning's n roughness coefficient
A = Flow Area (m^2)
R = Hydraulic Radius (m) – area / wetted perimeter

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.

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.

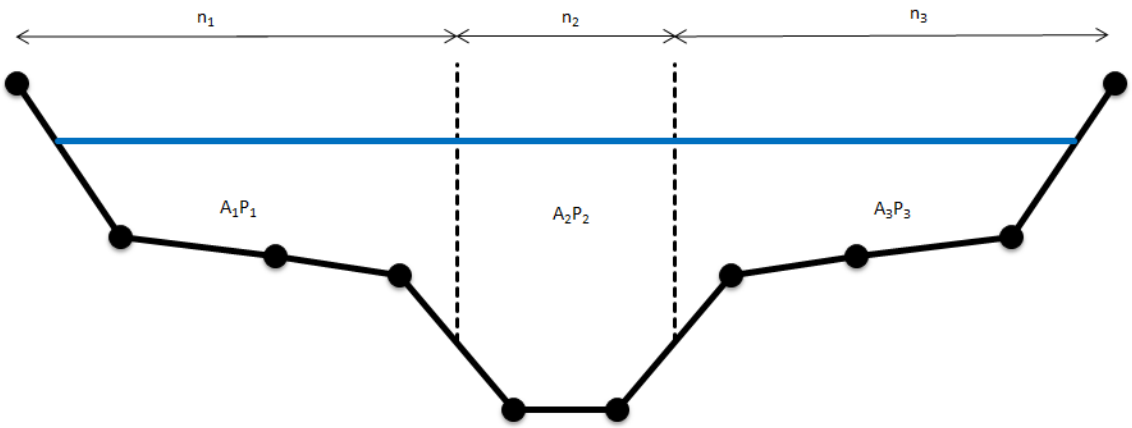
Figure 5-6 illustrates the ALL PARALLEL method of conveyance calculation.



$$K_{total} = K_1 + K_2 + K_3 + K_4 + K_5 + K_6 + K_7 + K_8 + K_9 + K_{10}$$

Figure 5-6 'All Parallel' Conveyance Calculation Method

Conveyance Calculation == CHANGE IN RESISTANCE may be used as an alternative to the default ALL PARALLEL approach. In this case, 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 (R flag), material type (M flag) or Manning's n values (N flag)). This is illustrated in Figure 5-7:



$$K_{total} = K_1 + K_2 + K_3$$

Figure 5-7 'Change in Resistance' Conveyance Calculation Method

It should be noted that differences in results are expected between the two methods of conveyance calculation. The total number of panels for each calculation method will be different as demonstrated in Figure 5-6 and Figure 5-7, thereby influencing the total conveyance.

The ALL PARALLEL approach has been chosen as the current default conveyance calculation method for ESTRY. This is not to imply that this method produces the more accurate result, rather it has been chosen as it generally does not cause conveyance reducing with height warnings.

5.10.4 Effective Area versus Total Area

For XZ (offset elevation) Cross-Sections, the flow area is calculated as an effective area (E flag) or a total area (T flag). Use of the flag will override the global setting set by Flow Area where the default is approach is to use the effective 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. 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 averaged depth and width 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.

5.10.5 Mid Cross-Sections

Cross-sections may be specified using lines digitised within a 1d_xs layer partway along the channel. The upstream and downstream invert levels of the channel are both assigned the invert level of the cross-section if a value of -99999 has been specified within the 1d_nwk channel (refer to Table 5-2). If either of these attributes is greater than -99999, the invert of the channel is set to the GIS attribute value rather than that of the cross-section bed elevation.

The mid cross-section approach is the only approach available for structures such as bridges, weirs and irregular shaped culverts. It can also be used for open channels, however the digitisation of cross-section lines within a 1d_xs layer that have been snapped to the channel ends (as described in Section 5.10.6 below) has added advantages and is recommended.

5.10.6 End Cross-Sections

Cross-sections for open channels (S channels and the superseded G channels) can be specified using lines digitised within a 1d_xs layer at the channel ends, rather than a single cross-section midway along the channel as described above. This approach has the following benefits:

- The upstream and downstream inverts can be based on the beds of the cross-sections, thereby saving some effort to enter this information within the 1d_nwk file. To do this, set the US_Invert and DS_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 some other 1D models often have the cross-sections at the channel ends, therefore, this makes it easier to use these external data sources.

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

- The 1d_xs cross-section lines must have a vertex snapped to the channel end.
- If a 1d_xs cross-section line occurs elsewhere along an open 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. Alternatively, an X connector can be used if end cross-sections are required for both channels. See Section 5.8.3.
- 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 S and G channels, end cross-sections are ignored.

5.10.7 Interpolated Cross-Section Protocols

Cross-sections may be 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 to avoid interpolating sections from side channels
    - 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 from upstream to 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.
- 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 no longer supported fixed field formats. This includes CS or HW 1d_xs formats (see

- Table 5-14).
- 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. Interpolate Cross-Sections can also be used to switch this feature ON or OFF.

5.11 Nodes

Nodes are created in a several ways:

- Digitised points in a 1d_nwk layer;
- From a pit in a 1d_nwk layer (see Section 5.12.2);
- From a manhole in a 1d_mh layer (see Section 5.12.5), or
- If none of the above, automatically created by TUFLOW at channel ends.

Note: The use of the term "node" in this manual refers to both manually digitised nodes and nodes that are automatically created at the ends of channels where no digitised nodes exist.

If a node has not been digitised at the end of a channel a new node is automatically created. The ID of automatic 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 Create Nodes. If automatic creation of Nodes is turned off, a manually defined node must be specified at the ends of each channel in the model.

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.

Table 5-15 below lists the available node types.

Table 5-15 1D Node and Pit Types (1d_nwk Point Objects)

Channel/Node	Type	Description
Primary Channel (Line) or Node (Point) Types		
Node	**Node** (or leave blank)	No special node type.
Circular Culvert	**C**	Circular (in the vertical) pit channel inlet.
Depth-Discharge Pit or Channel	**Q**	A pit channel whose flow is defined by a depth-discharge curve from a database of curves (see Section 0).
Rectangular Culvert	**R**	A rectangular (in the vertical) pit channel inlet.
Weir	**W**	A weir (in the vertical) pit channel inlet.

Storage at a node is defined using either a storage table of elevation versus surface area or using the widths and lengths of the connected 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.

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

1 User defined NA tables take precedence (see Section 5.11.3). The overwrite principle applies, so that if a NA table has been previously defined, the latter NA table prevails.

2 Nodes without any NA tables assigned as for Step 1 above, and have one or more connected channels that have the UCS attribute left blank or set to "Y" or "T", have their NA table automatically calculated from the channel widths (see Section 5.11.1).

3 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. 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.

4 If there are any nodes remaining without a NA table, an ERROR occurs.

Manually digitised nodes may be automatically linked to 2D domains without the need to digitised objects within a 2d_bc layer. Refer to the discussion within Section 5.12.4.

Table 5-16 lists the attributes required for manually created nodes.

Table 5-16 Nodes: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. Digitised nodes can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	Leaving this attribute blank or specifying "Node" will define the point feature as a Node type. "Node" is recommended for easy identification.	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS	Not used.	Char(1)
5	Len_or_ANA	Adds the value specified as additional nodal area (surface area in m^2). 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 set to this value with an elevation range based on the US_Invert and DS_Invert values. 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 effect of Minimum Channel Storage Length. Minimum NA is applied after the multiplication.	Float
6	n_nF_Cd	Not used.	Float
7	US_Invert	If no NA table exists and Len_or_ANA is greater than zero, used to set the upper elevation of the NA table. Note that in this case if US_Invert is less than DS_Invert, US_Invert is set to the DS_Invert plus 5 metres. If Conn_1D_2D is set to "SXL", US_Invert is used as the amount by which to lower the 2D cell.	Float
8	DS_Invert	If no NA table exists and Len_or_ANA is greater than zero, used to set the bottom elevation of the NA table.	Float

No.	Default GIS Attribute Name	Description	Type
		Also used to set the upstream and downstream inverts of connected open channels, culverts/pipes and other channel types – as an example see discussion for US_Invert in Table 5-2. Also see Section 5.11.4. If set to -99999, not used.	
9	Form_Loss	The Form_Loss value is applied as an energy loss to all outgoing culverts (ie. culverts that are digitised as leaving the pit/node), therefore, the Form_Loss value should be based on the outgoing pipe velocity. 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 5.7.6.	Float
10	pBlockage	Not used.	Float
11	Inlet_Type	Not used.	Char(256)
12	Conn_1D_2D	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 node occurs. This negates the need to create SX objects in a 2d_bc layer (see Flags attribute in Table 7-5). Notes: • "SXL" can be specified to connect the 1D node to the 2D domain and lower the 2D cell by the amount of the US_Invert attribute. • By default, if more than one 2D cell is automatically connected, nodes are assumed to be connected using the Sag (S) approach (see Section 5.12.4). To override the default approach specify "SXG" for an on-grade connection.	Char(4)
13	Conn_No	If "SX" is specified for Conn_1D_2D, can be used to control the number of 2D cells connected as follows. See Section 5.12.4 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. **Note that if a culvert with more than one barrel is connected to the node, the number of barrels is used to set the width, which controls the number of automatically selected SX cells.**	Integer
14	Width_or_Dia	If "SX" is specified for Conn_1D_2D, can be used to control the number of 2D cells connected. For example, if a width of 3.6m is entered on a 2m grid, two boundary cells are automatically selected.	Float

No.	Default GIS Attribute Name	Description	Type
15	Height_or_WF	Not used.	Float
16	Number_of	Not used.	Integer
17	HConF_or_WC	Not used.	Float
18	WConF_or_WEx	Not used.	Float
19	EntryC_or_WSa	Not used.	Float
20	ExitC_or_WSb	Not used.	Float

Table 5-17 1D Model Network (1d_nd) Attribute Descriptions for Nodes

No.	Default GIS Attribute Name	Description	Type
Read GIS Node **Command**			
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. Digitised nodes can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	Not used for Nodes although it can be recommended to type in "Node" for easy identification.	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Logical
4	Bed_Level	If no NA table exists and Len_or_ANA is greater than zero, used to set the bottom elevation of the NA table. Also used to set the upstream and downstream inverts of connected open channels, culverts/pipes and other channel types – as an example see discussion for US_Invert in Table 5-2. Also see Section 5.11.4. If set to -99999, not used.	Float
5	ANA	Adds the value specified as additional nodal area (surface area in m^2). If no nodal area data exists for the node TUFLOW automatically creates an NA table of constant surface area using this value. 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	Float

No.	Default GIS Attribute Name	Description	Type
		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 effect of Minimum Channel Storage Length. Minimum NA is applied after the multiplication.	
6	Conn_1D_2D	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 node occurs. This negates the need to create SX objects in a 2d_bc layer (see Flags attribute in Table 7-5). Notes: • "SXL" can be specified to connect the 1D node to the 2D domain and lower the 2D cell by the amount of the US_Invert attribute. • By default, if more than one 2D cell is automatically connected, nodes are assumed to be connected using the Sag (S) approach (see Section 5.12.4). To override the default approach specify "SXG" for an on-grade connection.	Char(4)
7	Conn_Width	If "SX" is specified for Conn_1D_2D, can be used to control the number of 2D cells connected as follows. See Section 5.12.4 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. **Note that if a culvert with more than one barrel is connected to the node, the number of barrels is used to set the width, which controls the number of automatically selected SX cells.**	Float
8	R1	Reserved for future use	Float
9	R2	Reserved for future use	Float
10	R3	Reserved for future use	Float

5.11.1 Storage Calculated from Channel Widths

The channel storage is, by default, automatically assigned to the nodes at the channel's ends. The channel storage approach is invoked using the 1d_nwk UCS attributeTable 5-11. If the attribute is left blank or set to "Y" (yes) or "T" (true), the channel storage is assigned to the two nodes at the channel's ends, with the storage split equally between the nodes. For each node the surface area at different elevations is calculated as the product of the channel width by half the channel length. The channel slope is taken into account when distributing the storage.

This approach does not require any specification of a NA table and is therefore the easiest to implement. 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 flow paths of a river. It may not be a suited to, for example, floodplain areas where the storage may differ significantly from that calculated using the widths of the floodplain channels.

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

Nodal storage is not automatically calculated based on the channel width and length if the 1d_nwk UCS attribute is set to "N" (no) or "F" (false). This would require manual specification of an NA table (see Section 5.11.3).

5.11.2 Storage above Structure Obverts

For models where the storage contributed by culverts and bridges is significant (eg. an urban pipe model), use Storage above Structure Obvert to minimise the storage contributed by these channels above their obvert (note that some storage is necessary to prevent a divide by zero in the equations).

5.11.3 Storage Nodes (User Defined NA Tables)

Storage at a node can be manually defined using an elevation versus surface area table (NA table – NA stands for **N**odal surface **A**rea). This provides the opportunity to accurately define the storage of the floodplain including any backwater areas that do not act as flow paths.

The NA table must be in a comma or space delimited text file linked to a 1d_na layer using the command Read GIS Table Links. Similar to the other table link GIS layers (1d_xs and 1d_bg), 1d_na GIS layers and .csv files are often stored in a separate folder underneath the model folder (same level as the mi or GIS folder):

- 1d_na for nodal surface area tables in a model\na folder

Unlike the other table link GIS layers (1d_xs and 1d_bg) the 1d_na layer must be digitised as a point and not a line. Table 5-18 presents the attributes of a 1d_na GIS layer.

Minimum NA is useful for stabilising 1D nodes that have small surface areas, particularly at shallow depths or when wetting and drying. The Minimum NA value is not applied to automatically created NA tables 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. See also Minimum NA Pit.

Minimum Channel Storage Length can be useful to add additional storage for stability reasons to nodes at the ends of very short channels.

Table 5-18 1D Nodal Area Table Link (1d_na) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Table Links Command			
1	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(50)
2	Type	**"NA":** Nodal surface area versus height table. The first column is elevation and the second surface area in m^2.	Char(2)
3	Flags	No optional flags available	Char(8)
4	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(20)
5	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(20)
6	Column_3	Not used.	Char(20)
7	Column_4	Not used.	Char(20)
8	Column_5	Not used.	Char(20)
9	Column_6	Not used.	Char(20)
10	Z_Increment	Not used.	Float
11	Z_Maximum	Not used.	Float
12	Skew (in degrees)	Not used.	Float

5.11.4 Using Nodes to Define Channel Inverts

1D nodes can be used to set the inverts of connected channels. Channel inverts are sourced from either, end cross-sections (see Section 5.10.6) or nodes if an upstream or downstream channel invert of -99999 is specified within the 1d_nwk file.

For this option the nodes "DS_Invert" attribute in the 1D_nwk layer or "Bed_Level" attribute in the 1d_nd format can be used to set the upstream and downstream inverts of any connected channels. If both nodes and cross-sections are specified the cross-section is given higher priority (refer to Section 5.10.5). This feature is useful during pipe network modelling, where cross-sections are not required (the shape is defined by the culvert type). The downstream inverts (of incoming channels) and upstream invert (of outgoing channels) will be set to the node elevation. In order to model different incoming and outgoing inverts (for manhole drop losses) these need to be specified on the channels.

5.11.5 Automatically Connecting Nodes to 2D domains

1D nodes may be automatically connected to 2D domains using the 1d_nwk Conn_1D_2D attribute (see Table 5-16). The approach is similar to the connection of 1D pits to a 2D domain and further discussion may be found in Section 5.12.4.

5.12 Pipe Networks

Pipe networks can be modelled using 1d_nwk channels to represent pipes, 1d_nwk nodes to represent pits and 1d_mh nodes to represent manholes.

In TUFLOW terminology, pits are used to convey water between above ground and the underground drainage layer and manholes are used to simulate energy losses at junctions. Pit (Section 5.12.2) and manhole (Section 5.12.5) features can exist in the same location.

5.12.1 Pipes

Culverts within a pipe network are created by digitising lines within one or more 1d_nwk GIS layer and assigning a type C, R or I (refer to Section 5.7.1, Table 5-1 and Table 5-3). The culverts are considered linked when the channel ends are snapped to each other.

There are a range of methods for determining the losses at these junction this is discussed in Section 5.12.5.

5.12.2 Pits and Pit Channels

Pits are created by digitising points within one or more 1d_nwk GIS layers and setting the "Type" attribute to C, Q, R or W (refer to Table 5-15). 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. Two nodes are automatically created: one at ground level for connection to the 2D domain and one at the pipe junction. Both pits and nodes can be automatically connected to 2D domains - see Section 5.12.4.

The term "pit channel" refers to the small channel automatically inserted at a pit node.

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_1D_2D 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 12.5) and the pit channel will remain unconnected.

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 its location to its connected node.

Minimum NA Pit sets the minimum NA (Nodal Area) of the upstream (ground) nodes for all pit channels. This command was introduced 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.

Table 5-19 describes the 1d_nwk attributes required for pits.

Table 5-19 Pits: 1D Model Network (1d_nwk) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Network Command			
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. Digitised pits can have their ID left blank and TUFLOW will assign an ID.	Char(12)
2	Type	Used to specify a Pit Channel as one of C, Q, R or W as per Table 5-15.	Char(4)
3	Ignore	If a "T", "t", "Y" or "y" is specified, the object will be ignored (T for True and Y for Yes). Any other entry, including a blank field, will treat the object as active.	Char(1)
4	UCS (Use Channel Storage at nodes).	Not used.	Char(1)
5	Len_or_ANA	Adds the value specified as additional nodal area (surface area in m^2). 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. 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 effect of Minimum Channel Storage Length. Minimum NA is applied after the multiplication.	Float
6	n_nF_Cd	Not used.	Float
7	US_Invert	Used to specify the ground elevation of the pit. This is used to set the upstream and downstream elevation of the pit channel. 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_1D_2D attribute below). If Conn_1D_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.	Float

No.	Default GIS Attribute Name	Description	Type
8	DS_Invert	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.	Float
9	Form_Loss	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 5.7.6.	Float
10	pBlockage	Not used. This field is reserved for future builds for blocking pit channels.	Float
11	Inlet_Type	For Q pit channels, the name of a pit inlet type in the Pit Inlet Database (see Section 0). Otherwise not used.	Char(256)
12	Conn_1D_2D	Used to specify a "SX" or "SXZ" flag that automatically creates a 2D SX cell and 2D/1D connection where the 1D pit occurs. This negates the need to create SX objects in a 2d_bc layer. The following options/changes are available: • "SX" can also be used to automatically connect 1D nodes as well as pits to the 2D domain. • "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 5.12.4). To override these default approaches specify either "SXG" or "SXS". Pit Default 2D Connection can be used to set a global default value for Conn_1D_2D, and up to four characters is now accepted (previously only three were accepted). "NO" can also be used to not connect the node or pit to a 2D domain.	Char(4)
13	Conn_No	If "SX" is specified for Conn_1D_2D, can be used to control the number of 2D cells connected. See Section 5.12.4 for a discussion on how the 2D cells are selected.	Integer

No.	Default GIS Attribute Name	Description	Type
		• 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.	
14	Width_or_Dia	**C, R, W Pit Type:** Sets the diameter of a C pit channel or the width of an R or W pit channel. **Q Pit Type:** 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).	Float
15	Height_or_WF	Sets the height of an R pit channel. Not used for other pit types.	Float
16	Number_of	**C, R, W Pit Type:** Sets the number of pits (of same dimension) within the one pit channel. **Q Pit Type:** 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 tripled. If set to zero (0), the MapInfo default, a value of 1 is used (ie. a single inlet pit is assumed).	Integer
17	HConF_or_WC	**R Pit Type:** Sets the height contraction coefficient, otherwise not used for other pit types.	Float
18	WConF_or_WEx	**C, R Pit Type:** Sets the width contraction coefficient, otherwise not used for other pit types.	Float
19	EntryC_or_WSa	**C, R Pit Type:** Sets the entry loss coefficient, otherwise not used for other pit types.	Float
20	ExitC_or_WSb	**C, R Pit Type:** Sets the exit loss coefficient, otherwise not used for other pit types.	Float

5.12.3 Pit Inlet and Depth/Stage vs Discharge Databases

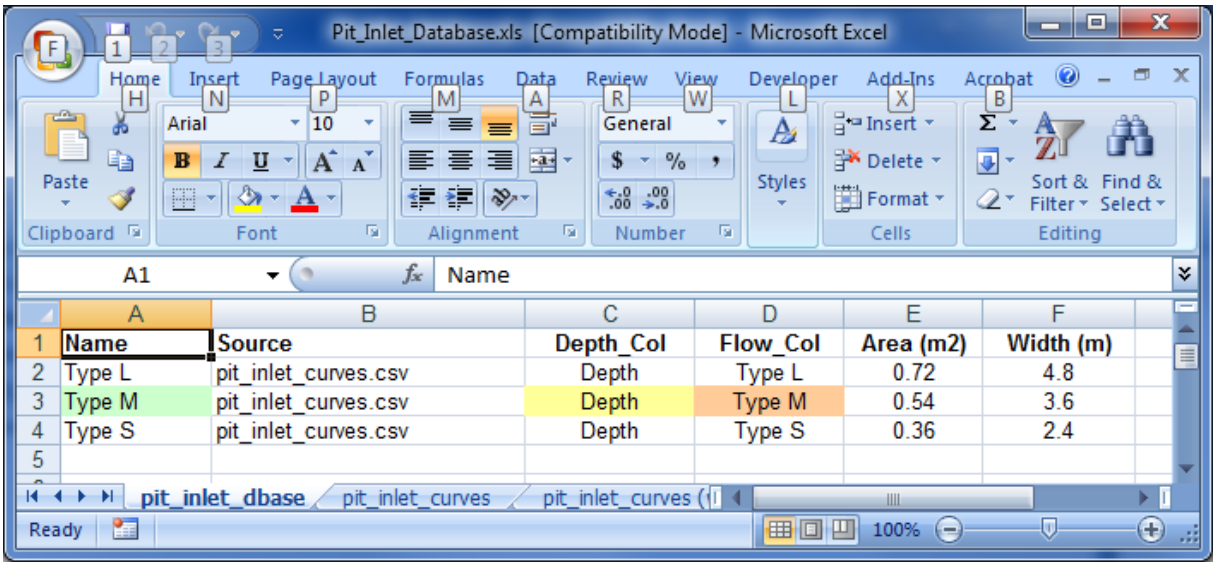
Pit channels can be specified as a Q Type, where flow through the pit is controlled by a depth-discharge curve defined within the Pit Inlet Database. The database is similar to the BC Database and is described below. The alternative naming convention Depth Discharge Database performs the exact same function and can be used instead of Pit Inlet Database. Only a single Depth Discharge or Pit Inlet Database should be used.

The 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 within the same Excel spreadsheet. The TUFLOW Microsoft Excel macro can be used to export these csv files from the parent spreadsheet. It is available to download at no cost from www.tuflow.com. The format of the Pit Inlet Database is described in Table 5-20.

Table 5-20 Pit Inlet Database Format

Column No.	Description
1	Contains the pit inlet type as referenced by the 1d_nwk Inlet_Type attribute (see Table 5-19).
2	Contains the name of the source .csv file that contains the depth-discharge curve.
3	The heading label of the column containing depth within the source .csv file.
4	The heading label of the column containing flow within the source .csv file.
5	The pit inlet's nominated full flow area in m^2. 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).
6	The pit inlet's nominated flow width in m. The width is used for the automatic selection of 2D cells when connecting to a 2D domain (see Section 5.12.4) and for extending the depth discharge curve.

The image below shows an example of a Pit Inlet Database .csv file. For the example, 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.



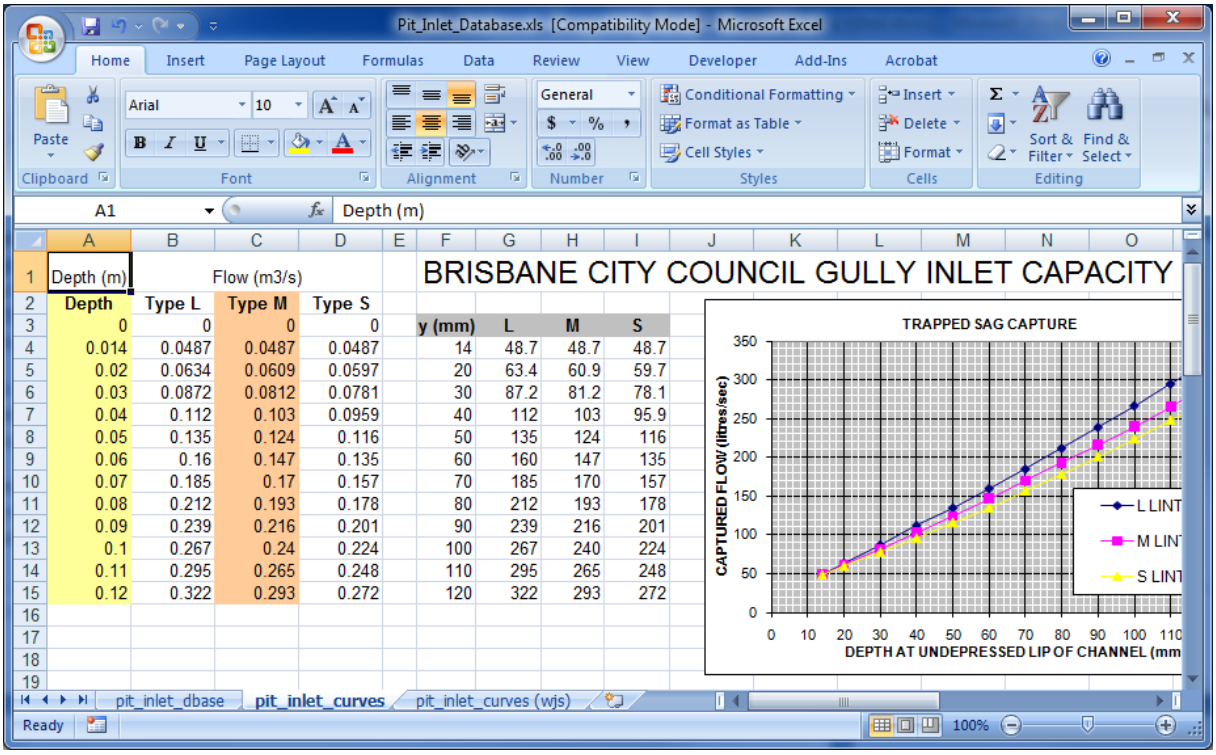


Figure 5-8 Example of Pit Inlet Database

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 or feet and ft^3/s if the model is running in US Customary Units (see Units).
- 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 from the pipe network onto the 2D domain.

- 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.

Table 5-21 presents the different Q pit flow regimes. These are output to the .eof file next to the velocity and flow values, and to the _TSF and _TSL output layers (see Sections 12.7 and 13.2.3).

Table 5-21 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.

5.12.3.1 Road Crossfall Options

The command Pit Default Road Crossfall increases the depth at Q pits based on the crossfall slope of the road cross-section. This approach aims to account for changes in ground elevation that occur at a finer scale than the 2D cell resolution. TUFLOW calculates the water depth that is used by the Pit Inlet Database using an adjacent side triangle depth approach, instead of the actual cell flow depth. This is shown in Figure 5-9. The resulting triangle has the same area as the vertical flow area in the 2D cell the pit is connected to (ie. the triangle's area is the depth in the 2D cell times the width of the cell).

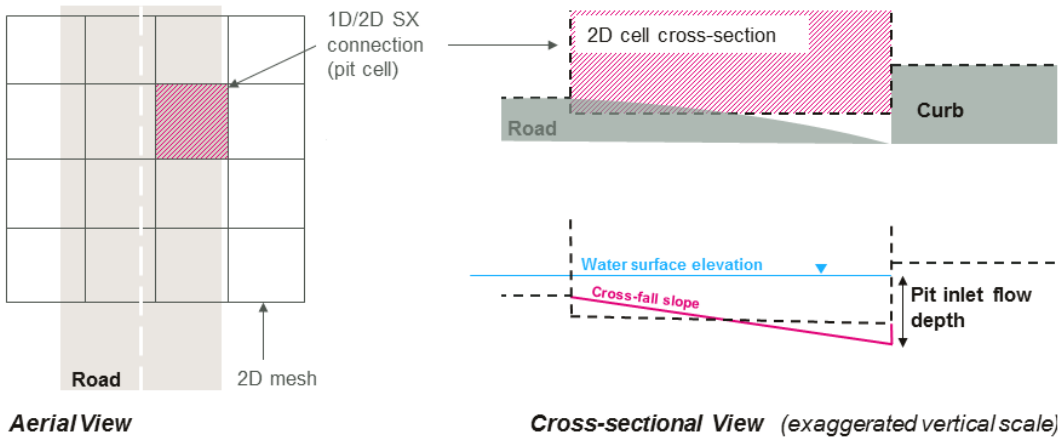


Figure 5-9 Road Crossfall Option

Use of this command can improve the ability to capture water from 2D cells into pits feeding the pipe network. Especially for larger cell sizes, the shallow depth that can occur in the cell can be unrepresentative of the depth at the entrance to the pit, therefore restricting the amount of flow entering the pit. The larger depth using an equivalent triangle will push more water into the pit providing a more realistic representation of the pit's capture.

5.12.4 Connecting Pits and Nodes to 2D Domains

Both pits and nodes can be automatically connected to 2D domains using the 1d_nwk Conn_1D_2D attribute (see Table 5-19). If Conn_1D_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. WARNING 2037 is given if there is overlapping 2D domains and more than one 2D cell is active.

The command Pit Default 2D Connection can be used to set the global default value for the 1d_nwk Conn_1D_2D attribute.

By default TUFLOW will connect the 1D element to one or more 2D cells at pit connections. 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 connection cells is limited to 10 per pit inlet.

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. The upper limit is 10 connected cells.
- 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, etc… 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, etc… 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_1D_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 an S flag for the Conn_1D_2D attribute (ie. “SXS”). 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_1D_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.

5.12.5 Manholes

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 5-3 for more information).

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 (this is the default approach).
- "FX" for the Fixed loss approach.

The manhole loss approach can be selected globally using the command, Manhole Default Loss Approach

The manholes types, loss methods and dimensions are output in the _mhc_check file. For more information on the manhole check file, please refer to the TUFLOW wiki.

5.12.5.1 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.

Three different approaches to the sizing of automatic manholes and the application of losses are available through the .ecf command Manhole Approach. The current default is Method C. This is the preferred approach unless backward compatibility is required for legacy reasons. Further details on

Methods A and B may be found by referring to the release notes of previous TUFLOW builds or by contacting support@tuflow.com.

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.

5.12.5.2 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 5-22.

A manhole in a 1d_mh layer will override any automatically created manhole. 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 5.7.1.

Table 5-22 1D Manhole (1d_mh) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Manhole Command			
1	ID	Unique identifier up to 12 characters in length (see ID for 1d_nwk layers). 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 5.11.	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)

No.	Default GIS Attribute Name	Description	Type
3	Loss_Approach	"NO" for no manhole loss approach (the standard entry/exit losses of the connected culverts applies). "EN" for the Engelund 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".	Char(1)
5	Invert_Level	The bottom or bed elevation of the manhole. This attribute 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. 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 Engelund 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 Engelund 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

5.12.5.3 Digitising Culverts Connected to Manholes

During the simulation, a culvert is determined to be an inlet or outlet culvert based on the **flow direction**, not the digitised direction of the 1d_nwk object. Therefore, if reverse flow occurs this is taken into account in determining the appropriate loss coefficients. 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 (ie. from upstream to downstream).

5.12.5.4 Engelund Manhole Loss Approach

The Engelund energy loss approach at manholes is based on the work of F.A. Engelund, and used by the MOUSE software (DHI, MOUSE PIPE FLOW – Reference Manual, p94, 2009). The adopted approach has been further enhanced through advice from, and discussions with, staff from Technical Services Branch of the Gold Coast City Council, Queensland, Australia. This is the default manhole loss approach.

The Engelund approach provides an automatic method for determining the energy loss coefficients 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 calculated values 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. Its time varying value 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_{Θ} is based on the angle of 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. Refer to Section 5.8.3 for further information.
- 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 calculated value is reported as 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.

- K_m is the manhole exit coefficient. 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 Engelund 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

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

Q_p = Flow in outlet culvert

W_m = Flow width in manhole = 1d_mh width 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_nwk Form_Loss attribute)

K_f = Fixed loss (1d_mh K_Fixed attribute)

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

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

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

$$A_m^{'} = \frac{W_m y_m Q_{om}}{Q_{im}}$$

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

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

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

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

$$K_{outlet\ pipe} = \min\left(K_\theta + K_{drop}, K_{Bend_Max}\right) + K_b + K_f \qquad \text{(applied as loss using outlet culvert velocity)}$$

5.12.5.5 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.

The energy loss is applied as an inlet loss to the outlet culvert if K_fixed is 0.5 or less. 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 Engelund loss approach as an additional (calibration) energy loss if required.

5.12.5.6 Discussion on Approaches to Modelling Pipe Junction Losses

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

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

2. **Least Conservative** (flatter hydraulic grade lines along pipes with lower flood levels overall): No manholes (Manholes at All Culvert Junctions set to "OFF" and no 1d_mh layers specified) and adjustment of structure losses (the default). In this case there will be minor losses at pipe junctions, because the Structure Losses by default will adjust the entrance and exit losses at pipe connections down according to the approach/departure velocities (see equations in Section 5.6.6). As the velocities are usually similar from one pipe to the next the entrance and exit losses are reduced downwards, often close to zero (as would be expected for a junction of two pipes with no manhole, no change in direction and no change in invert level).

3. **Middle Ground:** Automatically create and size manholes and apply the Engelund approach (the default setting). The automatic creation of manholes may be slightly conservative as a manhole (including junctions) is created at every closed node (a closed node occurs where only culverts and pits are connected). If there are many short pipes in the network an excess of manholes may result thereby causing greater energy losses. However, if single very long culvert channels are used to represent lengths of same sized pipes, any manholes along the long pipes will not be modelled and losses will be underestimated. In this case it is generally good practice

to split the long culvert channels into several channels (this will also produce an improved hydraulic grade line along the pipe).

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

5.13 Boundaries and 1D / 2D Links

1D and 2D domains use the same approach to define boundary conditions. They both access the same boundary condition database, although separate 1D and 2D databases can be used if desired. They also use the same commands within the ecf. and .tcf control files. For this reason, all model boundaries are discussed in a separate section of this manual in Chapter 7.

The linking of 1D and 2D domains is discussed separately in Chapter 8.

5.14 Presenting 1D Domains in 2D Output

Output from 1D domain(s), whether they be ESTRY, Flood Modeller, XP-SWMM or 12D, can be combined with the 2D map output. This allows easier viewing of results and the ability to animate the 1D results in combination with the 2D results. Refer to Section 9.5 for further information.

6 2D Model Domains

Chapter Contents

6 2D Model Domains 6-1

6.1 Introduction 6-3

6.2 Schematisation 6-4

6.3 Solution Scheme 6-6

6.3.1 2D Upstream Controlled Flow (Weirs and Supercritical Flow) 6-7

6.4 Boundaries and 1D and 2D/2D Links 6-9

6.5 2D Domain Extent and Resolution 6-10

6.6 Layering Datasets 6-11

6.7 Active / Inactive Areas 6-14

6.8 Elevations 6-16

6.8.1 Direct Reading of DEM Grids 6-16

6.8.2 Zpt Layers (2d_zpt) 6-17

6.8.3 3D Breakline Layers (2d_zln) 6-19

6.8.4 3D TIN Layers (2d_ztin) 6-19

6.8.5 Z Shape Layers (2d_zsh) 6-25

6.8.6 Variable Z Shape Layer (2d_vzsh) 6-34

6.8.7 Using Multiple Layers and Points Layers 6-39

6.8.7.1 Point Only Layers 6-39

6.9 Land Use (Materials) 6-41

6.9.1 Bed Resistance 6-41

6.9.2 Log Law Depth Varying Bed Resistance 6-43

6.9.3 Materials File 6-45

6.9.3.1 .tmf Format 6-45

6.9.3.2 .csv Format (Manning's n vs Depth Curves) 6-48

6.9.4 Rainfall Losses 6-53

6.10 Soil Infiltration 6-54

6.10.1 Green-Ampt 6-54

6.10.2 Horton 6-57

6.10.3 Initial Loss/Continuing Loss (ILCL) 6-57

6.10.4 Soils File (.tsoilf) 6-58

6.10.5 Groundwater 6-61

6.11 Cell Modification 6-62

6.11.1 Storage Reduction (2d_srf) 6-62

6.11.2 Cell Width Factor (CWF) 6-62

6.11.3 Form Loss Coefficient (FLC) 6-62

6.12 2D Hydraulic Structures 6-64

6.12.1 Introduction 6-64

6.12.2 2D Flow Constrictions (2d_fcsh and 2d_fc Layers) 6-68

6.12.2.1 Applying FC Attributes 6-74

6.12.2.2 Layered Flow Constrictions (2d_lfcsh Layers) 6-77

6.13 Modelling Urban Areas 6-83

6.13.1 Buildings 6-83

6.13.2 Roads 6-84

6.13.3 Fences and Walls 6-85

6.1 Introduction

This chapter of the Manual discusses features specifically related to the 2D model domain. 1D domain features are discussed separately in Chapter 5 and 1D/2D linking is discussed in Chapter 8.

6.2 Schematisation

The model topography in a 2D domain 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 6-1:

- "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
- Determines the bed slope when testing for the upstream controlled flow regime (see Section 6.3.1).

The ZU and ZV points:

- Control how water is conveyed from one cell to another;
- Represent where the momentum equation terms are centred and where upstream controlled flow regimes are applied;
- Deactivate if the cell has dried (based on the ZC point) and cannot flow; and
- Wet and dry independently of the cell wetting or drying (see Cell Wet/Dry Depth). This allows for 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: the Map Output Format == SMS HIGH RES option is available for outputting elevations and hydraulic calculations at the cell centres, mid-sides and corners.)

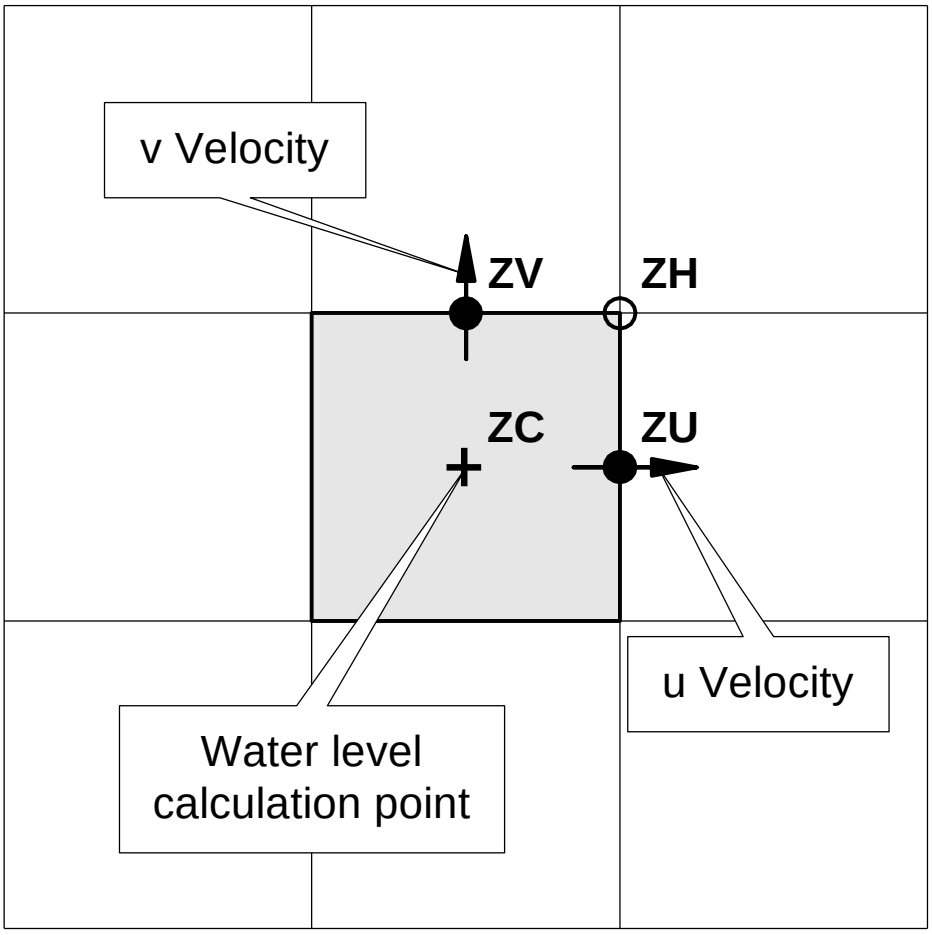


Figure 6-1 Location of Zpts and Computation Points

6.3 Solution Scheme

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:

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

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

(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 3.6).
- External forces such as bed friction, rotation of the earth, wind, wave radiation stresses, and barometric pressure.

An Alternating Direction Implicit (ADI) finite difference method is used for the computational procedure. It was originally 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.

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 dam break.

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.

6.3.1 2D Upstream Controlled Flow (Weirs and Supercritical Flow)

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

If Supercritical is set to ON (the default) 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 number 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. 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 can 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 adjusted globally using the .tcf command Global Weir Factor. It can also be varied spatially over the grid by setting a weir factor of zero where there is to be no automatic weir calculations using Read GIS WrF. The weir factor also allows calibration or adjustment where the broad-crested weir equation is applied. The weir factor is not the broad-crested weir coefficient. 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. Refer to Syme (2001b) for further information.

Note that the global value and the spatially varying value are multiplied together (ie. one does not replace the other).

The .tcf command HX Force Weir Equation can be used to force the weir flow equation to be applied across all active HX cell sides when the flow is upstream controlled. Note that the default approach uses either weir flow or super-critical flow when the flow is upstream controlled, depending on whether the ground surface gradient from HX cell centre to cell side is adverse (weir flow) or not adverse (super-critical flow). When the flow is downstream controlled, regardless of the ground surface slope, the full 2D equations are applied including allowance for momentum across the HX 1D/2D link.

6.4 Boundaries and 1D and 2D/2D Links

2D boundary conditions are discussed alongside 1D boundary conditions in Chapter 7 of this manual. The linking of 1D and 2D domains is discussed separately in Chapter 8.

6.5 2D Domain Extent and Resolution

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 due to some post-processing software only operating within this range.

Several options are available for setting the grid location and orientation. 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 2d_loc 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.
- Using a DEM to set the size and location of a 2D domain (see Read GRID Location). This option is useful where the model extent is the same as the DEM.

It is not essential at any point to specify dimensions that are an exact multiple of Cell Size.

Table 6-1 Location (2d_loc) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Location Command			
1	Comment	Optional field for entering comments. Not used.	Char(250)

After establishing the model domain origin, orientation and extent, the .tgc command Cell Size is used to define the model resolution.

Within this computational domain, 2D cells can be set to be active or inactive. From the 2011-09-AA version of TUFLOW, the redundant (inactive) areas in the model domain are removed from the computation. However, when allocating memory for processing of the arrays this is still accounted for.

To determine the redundant area, search the TUFLOW log file (.tlf) for “redundant” and the output will look like the text string below. If the redundant value is large, revising the model domain will reduce the memory usage.

```
Isolating redundant perimeter sections of 2D domain Domain_001...

...Reduced computational grid by 8%.  Now extends from [5,7] to [196,169].
```

6.6 Layering Datasets

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:

7. All elevations in the model start with an un-initialised value of 99999. TUFLOW will output an error if any elevations of 99999 (or higher) occur after the processing of the elevations.
8. A default elevation is specified first in the .tgc file using Set Zpt. A flood-free elevation is usually specified.
9. The elevations read directly from a DEM using Read Grid Zpts, or sampled from a DEM using Read RowCol Zpts, is specified next. These elevations will override those previously specified wherever they exist. Alternatively, elevations can be interpolated from a SMS, 12D or LandXML TIN (see Read TIN Zpts) or TUFLOW can be used to create a TIN (see Create TIN Zpts).
10. If there are areas where data is either missing or erroneous, these can easily be corrected via interpolation using Read GIS Z Shape regions or polygons. This is achieved by digitising a polygon around the missing or erroneous data. TUFLOW interpolates elevations across the region based on the existing Zpts around the perimeter of the polygon. An example is shown in Section 6.8.5. Missing or erroneous data often occurs with aerial surveys where the sampling is in areas of thick vegetation, water or where post processing has poorly filtered the removal of elevated objects (eg. buildings).
11. Sometimes there is a need to remove cut and fill works from the topography. For example, model calibration often requires the removal of, some levees, road embankments and developments that may not have existed at the time of the historic event to back date the topography within the model. This is easily done using Read GIS Z Shape by simply digitising polygons around the various features.
12. 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:

- 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.
- 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 into TUFLOW using Read GIS Zpts.

13. 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,

when a water level reached somewhere within the model, or based on the water level difference between two locations.

A 2D domain's Zpts are built up using one or more of the commands shown in Table 6-2. This concept of layering datasets may also be applied to other GIS layers including (but not limited to) 2d_mat, 1d_nwk and 2d_code layers.

Table 6-2 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. 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 GIS 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 package where each Zpt is assigned an elevation from a DTM. The updated layer is then exported and read by TUFLOW using 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). This command is specific to the model origin, alignment and cell size. If any of these change the RowCol layer will need to be re-generated. For this reason, directly inspecting from DEM (Read Grid Zpts) or TIN (Read TIN Zpts) is the preferred approach.
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. Read GIS Z Shape offers greater functionality and maybe preferable to using Read GIS Zpts or 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. 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 7-5) to adjust the 2D cell elevations along HX lines.

Command	Description
Read Grid Zpts	Directly interrogates an ESRI ASCII grid (.asc) or binary grid (.flt) to define Zpt elevations. This command is similar to Read TIN Zpts but works on a grid rather than a TIN.
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, 12D format or Civil 3D to set the Zpt values within the TIN.
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. Sets the ZC (cell centre elevation) to the lower of the ZU and ZV (cell sides).

6.7 Active / Inactive Areas

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 6-3. The default code value is one (1) for active 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. Read GIS BC in the .tbc file also automatically sets the Boundary Cell code of 2 along external boundaries.

A typical approach is to set the cells to be all inactive with the (Set Code == 0) and then read a GIS layer defining the active cells with Read GIS Code == <layer>. The GIS layer would have the attribute set to 1 for active. For example:

```
Set Code == 0  ! Set all cells to inactive (ie. Code 0)

Read GIS Code == ..\model\mi\2d_code_M01_002.MIF
```

Note when using the Read GIS Code command, code values are extracted from objects in a 2d_code layer (see Table 6-3). When using the Read GIS Code BC, code values are extracted from objects in a 2d_bc layer (see Table 7-5) that have a Type attribute of "CD" and a code value taken from the 2d_bc f attribute. Confusing the two GIS layer types and commands will result in a WARNING 2320 message being issued.

For models containing multiple 2D domains, a useful option for setting the cell codes is the INVERT flag (see Read GIS Code or Read GIS Code BC), which allows the same code layer/polygons to be used for both 2D domains.

Other useful commands include Read GRID Code, which allows for the assigning of cell codes from an ASCII grid and Set Code and Clean Zpt, which assigns cells as active or inactive based on whether an elevation has been assigned to the cell.

TUFLOW automatically strips any redundant rows/columns around the active area of the model to reduce simulation times. This is described in Section 6.5.

Table 6-3 Cell Codes

Type	Code	Description
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). There must be a water cell on one side and a null or land cell on the other at an external boundary. **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 7.4.1 and Read GIS BC).
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.

Table 6-4 2D Code (2d_code) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Code Command			
1	Code	The code value (see Table 6-3) to be assigned to cells falling on or within the object.	Integer

6.8 Elevations

6.8.1 Direct Reading of DEM Grids

The use of the .tgc command Read Grid Zpts allows TUFLOW to directly interrogate (point inspect) a DEM to set the Zpt elevations. This command is similar to Read TIN Zpts but works on a grid rather than a TIN.

Grid formats currently supported include the ESRI ASCII Grid format (.asc) and binary grids (.flt). These formats are also as used by TUFLOW_to_GIS to export grids (See Section 15.2.1).

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

Instructions for exporting an ASCII format grid using a number of GIS packages is provided on the TUFLOW wiki as per the links below:

- MapInfo with Vertical Mapper
- ArcMap
- QGIS

Note that for ArcMap and QGIS using the binary .flt version is faster to read and write and can be used instead of the ASCII format. Instructions for exporting to this format can be found on the wiki for ArcMap and the TUFLOW utility asc_to_asc.exe (to be renamed grd_to_grd.exe) can readily convert one, a selection or all grid files in a folder between various formats as discussed here.

TUFLOW will accept any file extension at present, as there doesn't seem to be a standard. If the file extension is .flt the binary float version is assumed, TUFLOW will assume the file is in the ASCII format for any other extensions. ERROR 2333 or ERROR 2334 will be issued by TUFLOW if the file is not in the ASCII format.

By using this command, your TUFLOW model will very likely become cell size independent. To change cell size only involves changing the .tgc command Cell Size, and modifying your timestep.

Like other .tgc commands, the command Read Grid Zpts can be specified multiple times. An option to specify ADD, MIN or MAX in the same way as for other similar commands is also available.

Clip regions can be specified as a second argument in the command Read Grid Zpts (and also Read TIN Zpts) by reading in a GIS layer containing one or more polygons to clip the area of Zpts to be inspected. For example, the command below will only assign elevations to Zpts that lie inside polygons within the 2d_clip_DEM layer:

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

The attributes of the clip layer are not used, and only polygons are processed. As such, any of the TUFLOW empty files can be used as the template for this layer. Polygons can have holes in them if required.

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

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

6.8.2 Zpt Layers (2d_zpt)

2d_zpt layers are used to modify Zpt elevations over an area. When the layer contains point objects, these are located at the ZC, ZU, ZV and ZH locations within a 2D cell, with each point being assigned an elevation. The layer is usually initially generated by TUFLOW using Write GIS Zpts, which creates the .mif/.mid or .shp/.dbf files. Elevations are then assigned to the fourth (Elevation) attribute from a DEM using 3D surface modelling software (for example, Point Inspection Analysis tool in Vertical Mapper or the Extract Values to Points feature in Spatial Analyst).

Unlike nearly all other layers, TUFLOW only reads the .mid or .dbf file for the 2d_zpt layer. The .mid/.dbf file contains the attribute data in comma delimited format. Read RowCol Zpts reads the attribute data and locates the elevation within the 2D domain using the n, m and Type attributes. The attributes of the 2d_zpt layer are listed in Table 6-5.

Read RowCol Zpts can be used as many times as needed. 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.

The Read RowCol Zpts command has two disadvantages; the GIS layer is cell size dependent and the point features are processed by TUFLOW based on their computational grid location (m,n), not the real-world coordinate location. This means, the 2d_zpt layer may need to be updated if the model cell size, origin or extent are changed. For this reason, the preferred approach is to use Read Grid Zpts (see Section 6.8.1) or Read TIN Zpts to define the base Zpt elevations. A combination of features discussed in Section 6.8 may be used to provide further definition to key features of the topography.

2d_zpt layers may also be referenced using the .tgc command Read GIS Zpts. In this case, the format of the 2d_zpt layer is different and as shown in Table 6-6. Any Zpt (ZC, ZU, ZV and ZH) falling within/on an object is assigned the object's value. The object may be a region (polygon), line or point noting that lines may also be digitised within a 2d_zln layer and referenced using the command Read GIS Z Line (see Section 6.8.3).

The ADD option adds the value of the "Elevation" attribute 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.

The Read GIS Zpts format is still supported, however, the Read GIS Z Shape offers all the functionality of the Read GIS Zpts and a lot more, it is therefore the preferred approach. The Read GIS Z Shape layer offers greater traceability in terms of check files and also allows min / max option to be specified for each object rather than for the whole layer.

Table 6-5 2D Z-point (2d_zpt) Attribute Descriptions (Read RowCol Zpts Command)

No.	Default GIS Attribute Name	Description	Type
Read RowCol Zpts Command			
1	n	2D grid row in which the cell is located.	Integer
2	m	2D grid column in which the cell is located.	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 6-1.	Char(1)
4	Elevation	Elevation of the point. It is good practice to change the name of this attribute from "Elevation" to the name of the DEM for data traceability.	Float

Table 6-6 2D Z-point (2d_zpt) Attribute Descriptions (Read GIS Zpts Command)

No.	Default GIS Attribute Name	Description	Type
Read GIS Zpts Commands			
1	Elevation	Elevation of the point or polygon in a 2d_zpt GIS layer, or a polyline within a 2d_zln GIS layer. When the ADD option is used, the value of the "Elevation" attribute is added to the value of the existing Zpts.	Float

6.8.3 3D Breakline Layers (2d_zln)

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. 2d_zln GIS layers use the same format as 2d_zpt layers as shown in Table 6-6. They are referenced using the .tgc command Read GIS Z Line.

The Read GIS Z Line default is to model a "thin" line which modify the ZH, ZU and ZV Zpt elevations only. If the THICK option occurs, interpolated Z values are applied to whole cells (ie. at the cell centres (ZC), all cell sides and cell corners). Other optional flags such as MAX, MIN, RIDGE or GULLY are also available.

Users should also be aware of the Read GIS Z HX Line for assigning Zpt elevations along HX lines.

The use of both the 2d_zpt and 2d_zln layers were previously the most powerful methods for easily manipulating the 2D geometry. They continue to be supported, however, users should familiarise themselves with the full range of capabilities available with the use of 2d_ztin (Section 6.8.4), 2d_zsh (Section 6.8.5) and 2d_vzsh (Section 6.8.6) layers as described in the following sections. One of the advantages of continuing to use both the 2d_zpt and 2d_zln layers is that they only require one attribute and are therefore simpler to use.

6.8.4 3D TIN Layers (2d_ztin)

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 Grid Zpts, Read RowCol Zpts or Read TIN Zpts to read the data into TUFLOW.

The protocols applied to the Create TIN Zpts command 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:
   (i) All vertices (nodes) of the line are converted to points.
   (ii) 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 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.

(iii) 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:

   (i) 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).

   (ii) 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.

   (iii) 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.

A second argument to specify a GIS layer containing one or more polygons to clip the area of Zpts to be inspected can be used with the Read Tin Zpts command. Refer to Section 6.8.1 for more information.

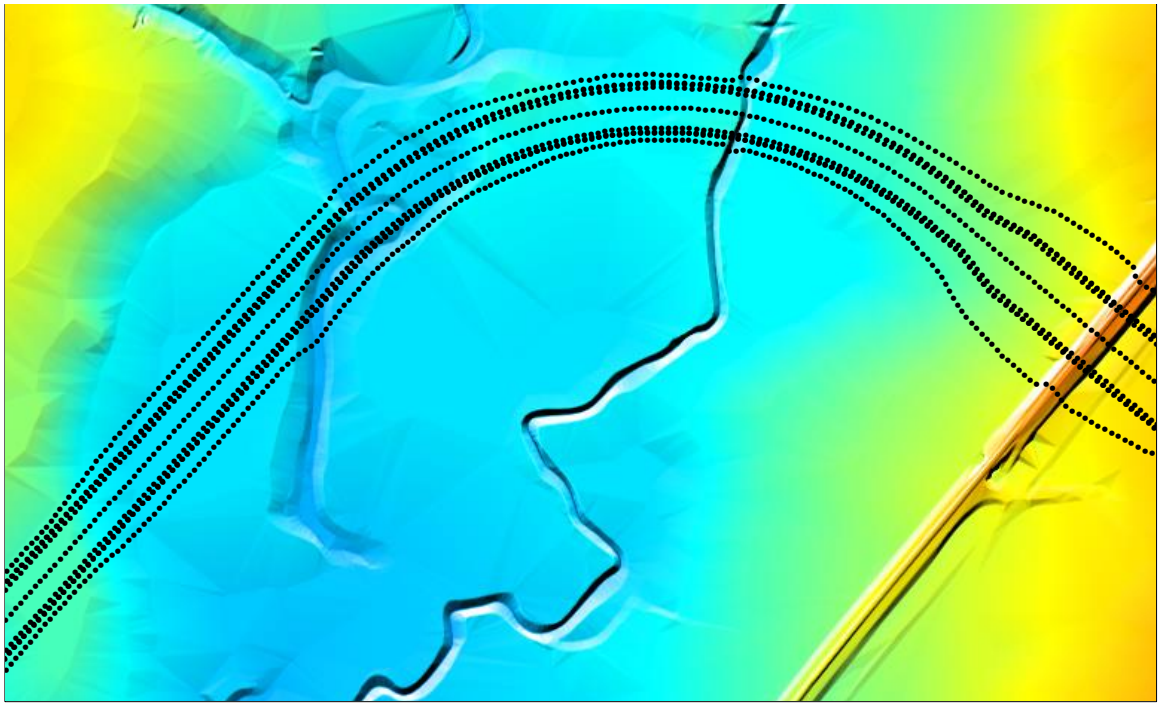
Table 6-7 2D Tin (2d_ztin) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read TIN Zpts Command			
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 created 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

Example 1: Haul Road Example of Using Create TIN Zpts

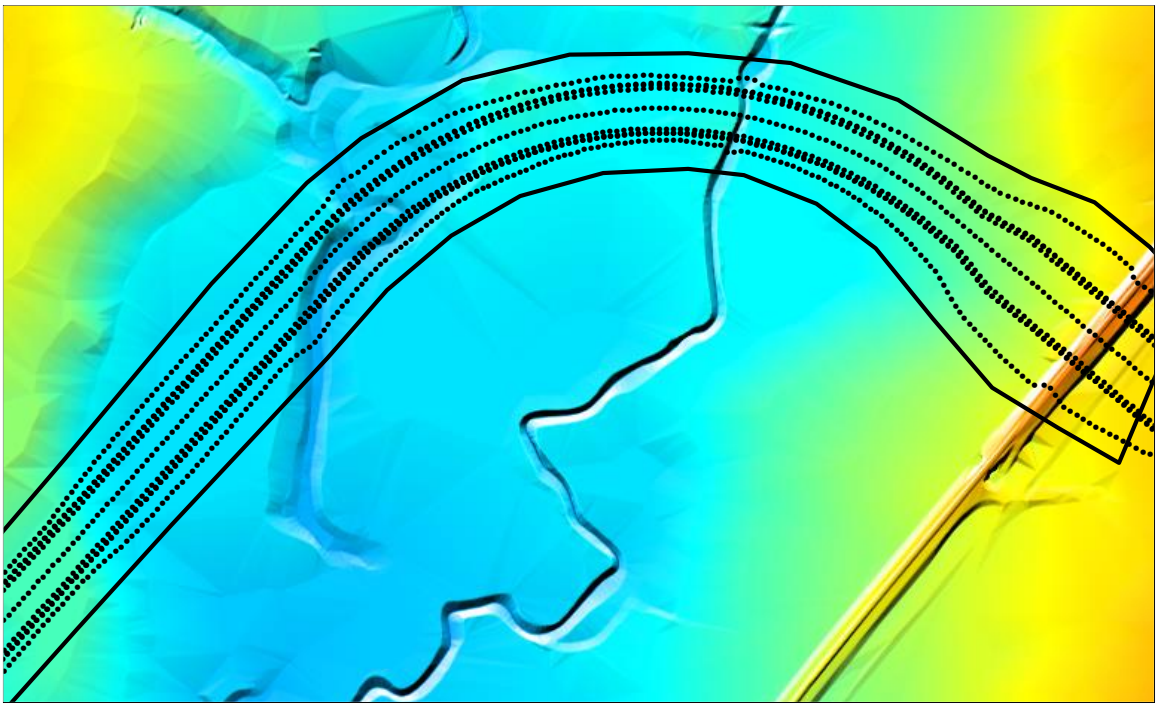
The images and text below illustrate an example of using Create TIN Zpts to easily incorporate a proposed haul road into a TUFLOW model. Read GIS Z Shape can also be used in a similar manner. The data for this example was provided courtesy of Hatch[2].

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.

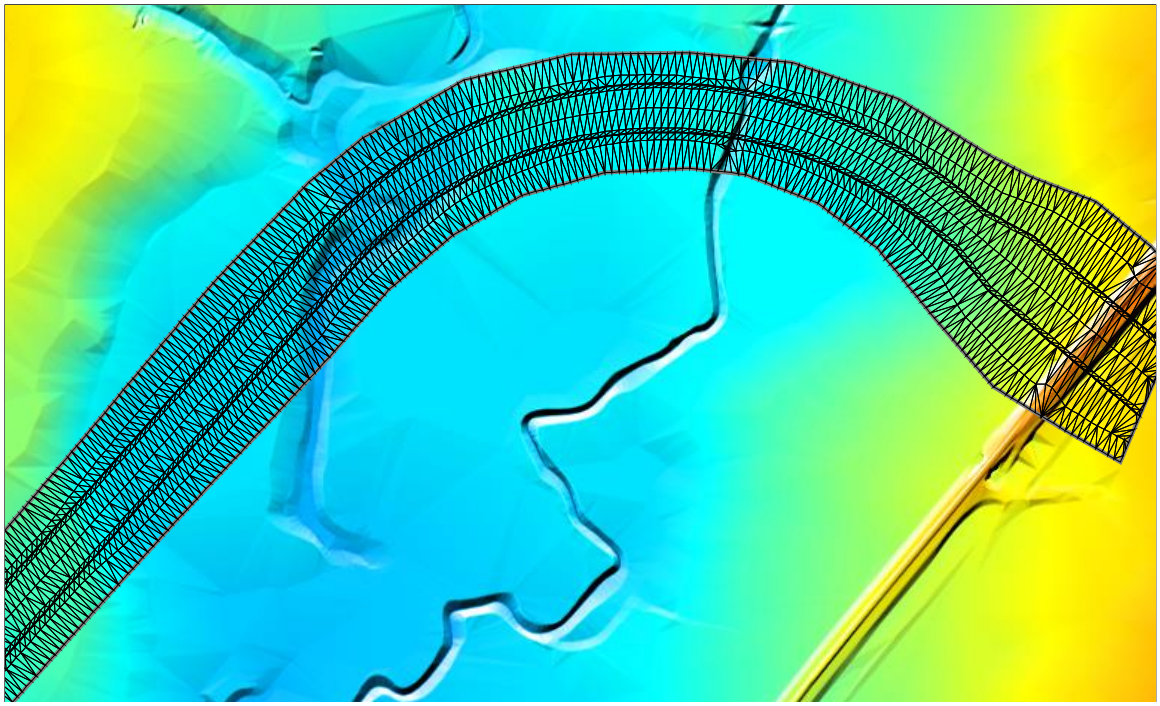


The points were copied into a MapInfo layer named 2d_ztin_haul_road_pts.tab, with the only attribute being the Elevation attribute that contains the Z values. A new layer was created and named 2d_ztin_haul_road.tab, and the polygon shown as the black solid line in the image below was digitised. Note 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.

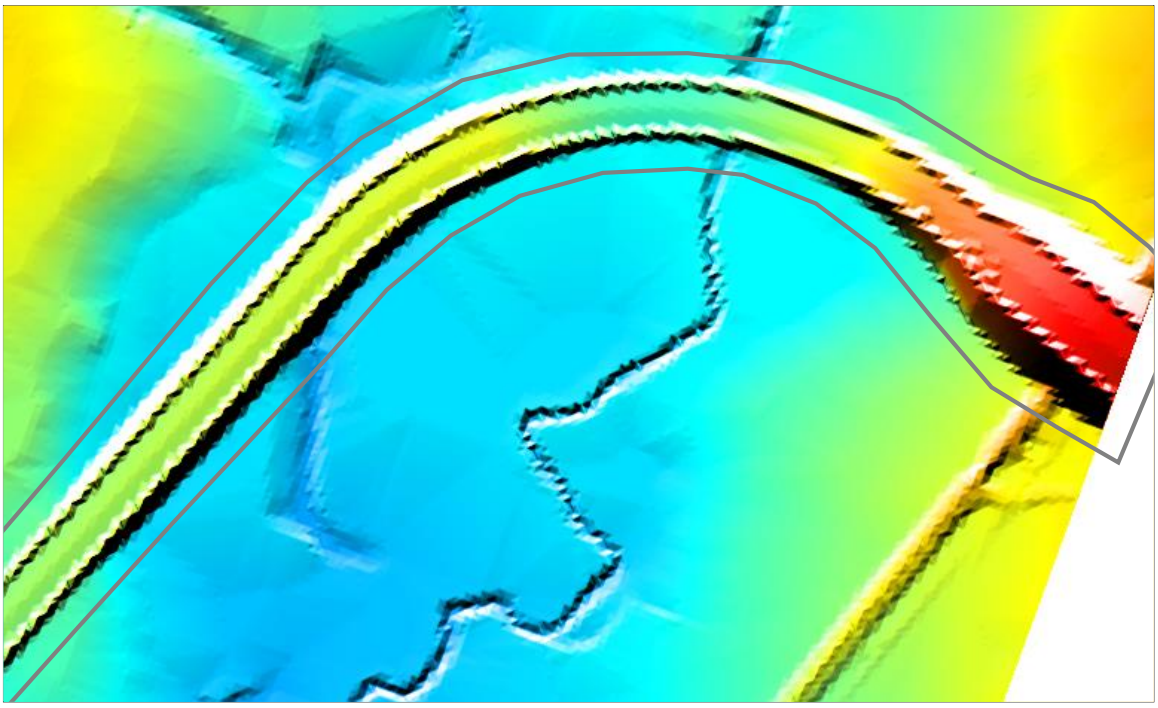
[2] www.hatch.com



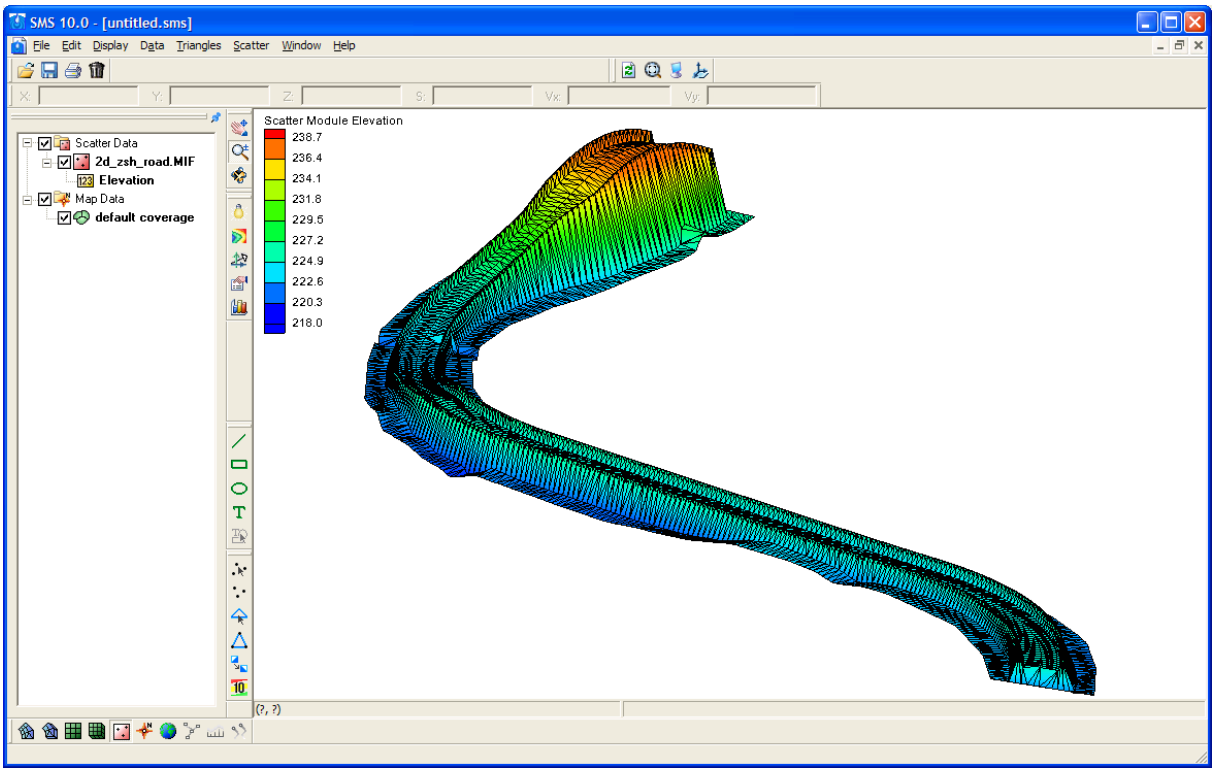
Create TIN Zpts == 2d_ztin_haul_road.mif was used to have TUFLOW create a TIN within the polygon as shown in the image below, and override the DEM Zpt values.



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.



Example 2: Highway Embankment Removal Example

The figures below present 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. Removal of the highway only required the digitising of 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.

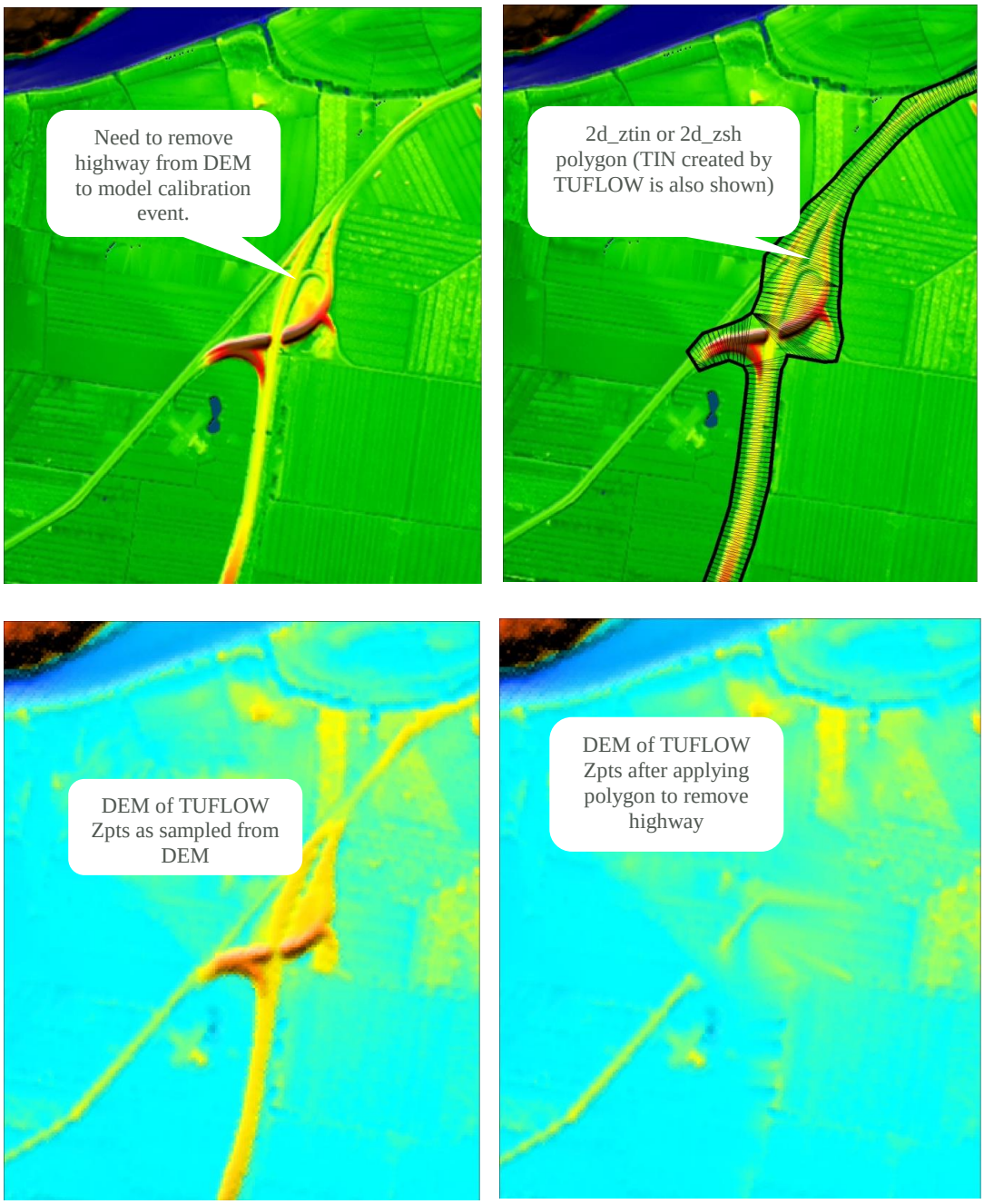


Figure 6-2 Example of Using a 2d_ztin or 2d_zsh Layer to Remove a Highway Embankment

6.8.5 Z Shape Layers (2d_zsh)

Read GIS Z Shape offers a wide range of options for manipulating and modifying the Zpt values. These include 3D breaklines and TINs. Table 6-8 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:

   (i). **Polygons:** All polygons are triangulated according to the process described for Create TIN Zpts in Section 6.8.4. The only difference to note is that if a line is to be used for the TIN generation, the Shape_Options attribute for the line must include the keyword "TIN".

   (ii). **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.

   (iii). **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_Options attribute setting as follows:

   (i) All ADD lines are applied first.

   (ii) Followed by 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.

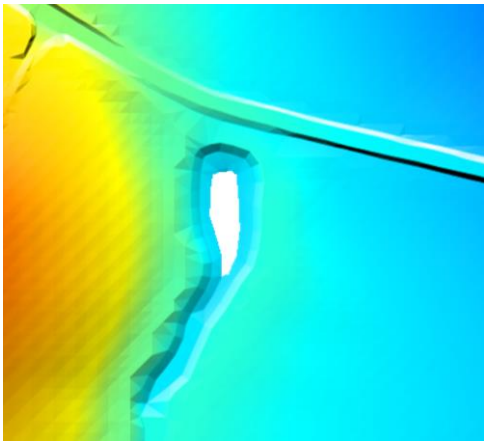
2. The priority can of course be further controlled by using different layers and controlling the order which layers are listed and subsequently processed in the .tgc file.

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.

Some examples of using Read GIS Z Shape are given below.

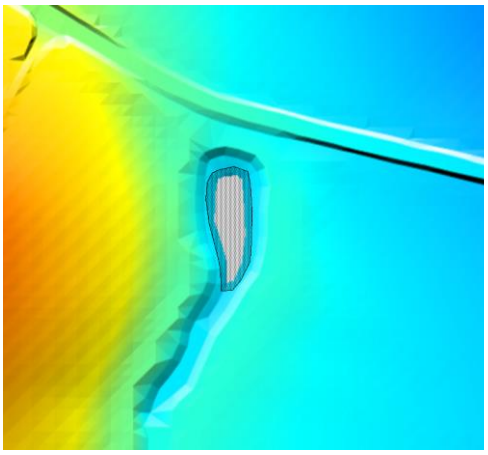
Example 1: Triangulating Elevations over a Null Area

The image below shows an example of a DEM that is missing data over a small area within the 2D domain. Gaps in coverage typically occur over water bodies and occasionally between the tiles of ALS or LiDAR data received from a third party provider.

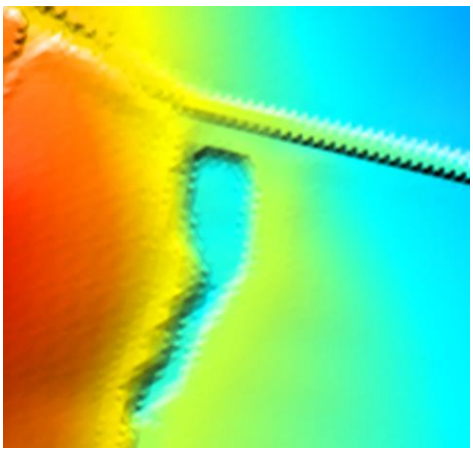


The .tgc command Set Zpt may be used to quickly and easily assign elevations to Zpts falling within these areas, however the limitations of the command mean the same elevation will be assigned to all null Zpts across the entire 2D domain. This may not be suitable in situations where there are multiple gaps in coverage or where the gap is located on steep terrain.

2d_zsh polygons may be used to triangulate Zpt values based on the Zpt elevations of the polygon perimeter.



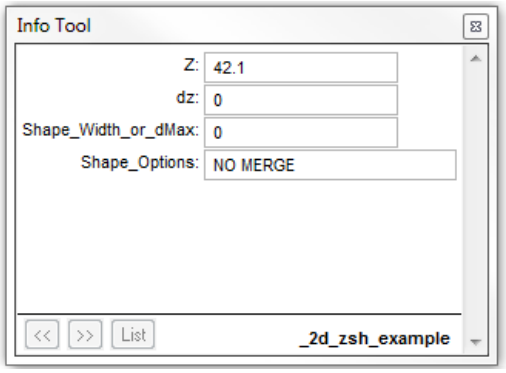
Import in an empty 2d_zsh GIS layer, and digitise a polygon around the gap in coverage as shown. Ensure there is a reasonable buffer around the null area. The attributes of the polygon may be left blank. Alternatively, a value may be entered in the "Shape_Width_or_dMax" attribute to control the maximum distance between intermediate points inserted around the polygon's perimeter to interpolate elevations. When left blank, this distance is half the 2D cell's size.



This image shows the resulting _DEM_Z.flt check file. The _zsh_zpt_check layer can be used to view the final Zpt elevation assigned

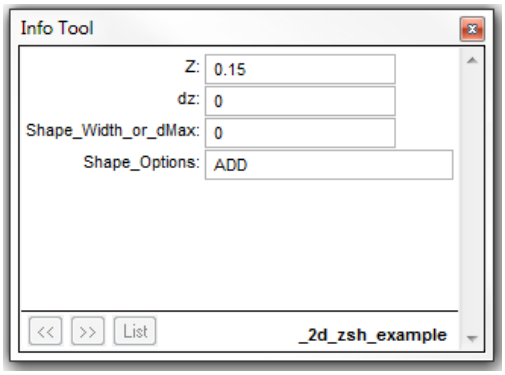
Example 2: Use of the NO MERGE and ADD Shape Options

The NO MERGE option can be used to assign a single elevation to all Zpts falling within the 2d_zsh polygon. This may be useful to set the elevation of a polygon to a known finished floor level of a proposed development. Digitise a polygon within an empty 2d_zsh GIS layer, and populate the "Z" attribute of each object with the desired elevation. Set the "Shape_Options" attribute to NO MERGE. The example below will assign an elevation of 42.1mAHD to all Zpts located within the 2d_zsh polygon.



Note that if the NO MERGE option is omitted and no points are snapped to the perimeter of the polygon, the Z attribute will be ignored and the Zpt elevations will be triangulated based on the Zpt elevations of the polygon perimeter.

Alternatively, to raise the polygon by a fixed value (ie. to represent the slab height of a building) enter this value in the Z attribute and set the "Shape_Options" to ADD.

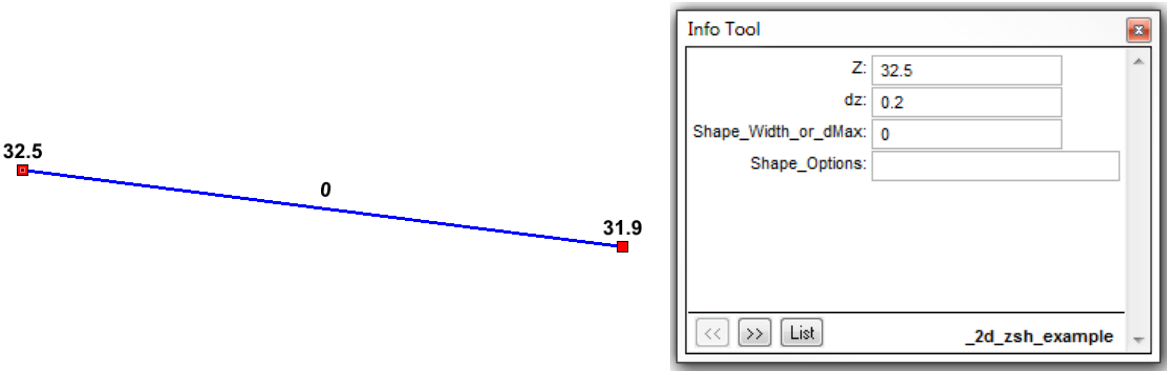


TUFLOW will add the value entered in the Z attribute to the existing Zpt elevations within the polygon. The entry within the figure to the left will raise Zpt elevations by 0.15m. The use of a negative value will lower the Zpt elevations by the value of the "Z" attribute. The _zsh_zpt_check layer can be used to view the elevation points (Zpts) that have been modified.

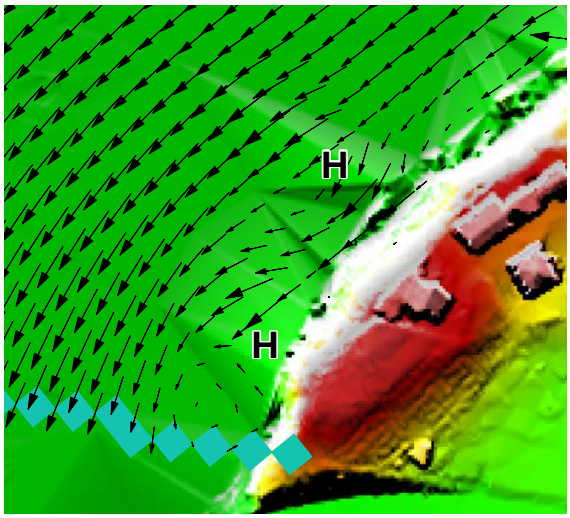
Example 3: Raising an Embankment

The ADD option may also be used when the 2d_zsh object has been digitised as a line. Populate the Z attribute with the amount the embankment is to be raised by. Populate the "Shape_Options" attribute with ADD as shown in the second figure of Example 2 above. This will raise the existing Zpt elevations by the value of the Z attribute. By default, TUFLOW will assume a thin line, and only alter the ZH, ZU and ZV Zpt elevations of a cell. The "Shape_Width_or_dMax" attribute may be optionally specified to represent a THICK or a WIDE line (refer to Section 6.8.5 and Table 6-8).

Alternatively, if a 3D breakline has been digitised, the dZ attribute on the snapped points may be used to raise or lower the embankment. The dZ attribute increases or decreases the elevation of the point's Z attribute by the amount of dZ. In the example below, a 3D breakline has been created by snapping points to either end of the polyline. The elevations along the line are determined by a linear interpolation of the Z attribute of the points. Entering a positive dZ value at each point will raise the elevations at the points by the amount of dZ at each point (0.2m for one of the points in the figure below). The elevations along the line are then interpolated based on these revised values. The _zsh_zpt_check layer can be used to view the final Zpt elevation assigned.

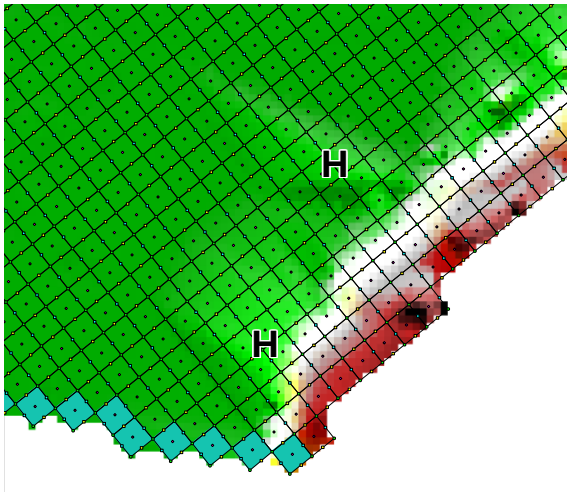


Example 4: Removing ridges from a poorly triangulated DEM.

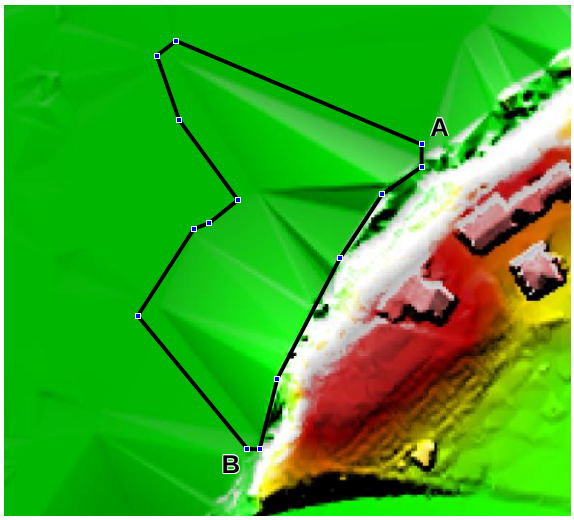


The image shows two false ridges indicated by the H letters. These were caused by a poor triangulation by the TIN software used to create the DEM. These ridges caused unrealistic flow patterns as shown by the velocity vectors.

The blue cells at the bottom are the downstream boundary cells.

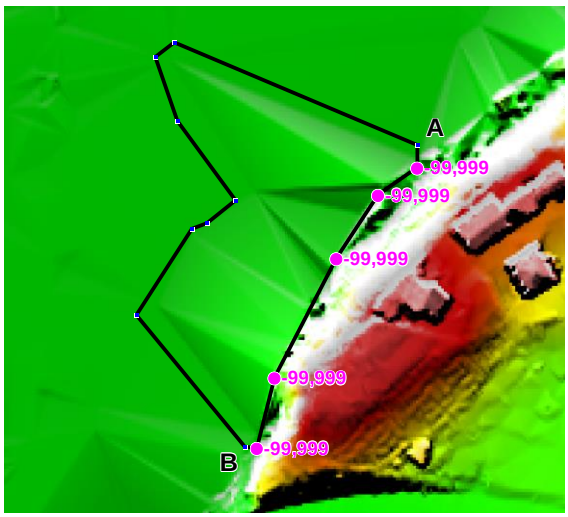


This image is of a DEM produced from the 2d_zpt_check.mif layer (which is shown as the points). This is how TUFLOW sees the DEM data with the false ridges clearly shown.

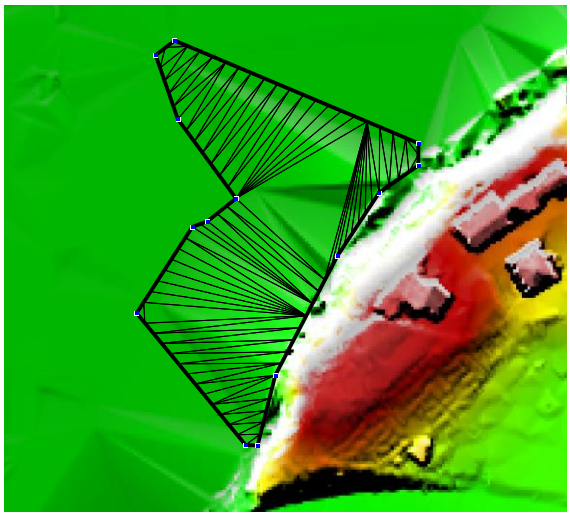


To remove the ridges, import an empty 2d_zsh layer, and digitise a polygon around the ridges as shown. By default (ie. using the default attribute values), the elevations assigned around the perimeter of the polygon are interpolated from the current Zpt values.

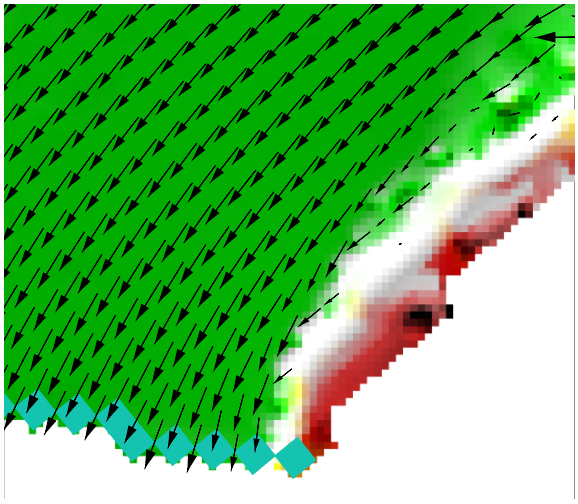
One problem with this approach is that the elevations along the right hand side (ie. along the edge of the floodplain between Locations A and B) are interpolated from the high Zpts along this boundary.



To solve this problem, digitise points either into the 2s_zsh layer or into a 2d_zsh…_pts layer 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 GIS layer.



This image shows a DEM created from the 2d_zpt_check GIS 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 6-8 2D Z-Shape (2d_zsh) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Z Shape Command			
1	Z	**Point:** Elevation 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. If the ADD option is specified, the value entered is used to increase (positive 'Z' values) or decrease (negative 'Z' values) the elevation of the Zpt values by the amount specified (ie. a value of 0.5 will raise existing Zpt values by 0.5m). Otherwise ignored. **Polygon:** If the NO MERGE option is specified, used to set the elevation of the polygon perimeter if there are no points snapped to the perimeter. If the ADD option is specified, the value entered is used to increase (positive 'Z' values) or decrease (negative 'Z' values) the elevation of the Zpt values by the amount specified. Otherwise ignored.	Float

No.	Default GIS Attribute Name	Description	Type
2	dZ	**Point:** Change in elevation of the 'Z' attribute of the z-shape object 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. For example, if the Z attribute has been set to 5mAHD, a dZ value of 0.1m will raise the point's Z elevation to 5.1mAHD. **Line:** Not used. Recommend setting to zero. **Polygon:** Not used. Recommend setting to zero.	Float
3	Shape_Width_or_dMax	**Point:** Not used. **Line (no TIN):** Provided "TIN" is not specified for the Shape_Options attribute, this attribute specifies the width (thickness) of the breakline in metres. If equal to zero, only the elevations along the cell sides and corners 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_Options 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 or ADD options are specified.	Float
4	Shape_Options	**Point:** Not used. **Line or Polygon:** ADD: Add the shape's 'Z' attribute value 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 a value of zero for the Z and dZ attributes for the region). MAX, RIDGE or RAISE: Only changes a Zpt elevation if the Z Shape elevation at the Zpt is higher.	Char(20)

No.	Default GIS Attribute Name	Description	Type
		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 of 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.	

6.8.6 Variable Z Shape Layer (2d_vzsh)

TUFLOW 2D model topography can be varied over time to simulate 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 using a number of methods:

- At a specified time;
- When the water level reaches a specified height at a specified (trigger) location; or
- When the water level difference between two triggers exceeds a specified amount.

Table 6-9 presents the attributes of a 2d_vzsh layer and an example is discussed and illustrated in the next section.

This feature 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 simpler to define and manage.

Variable Z Shapes can be restored once or repeatedly. Examples would be a breach of a flood defence wall or levee that is reinstated 6 hours later, or a sand bar of a creek entrance that repeatedly opens and closes. To use the restore feature, two additional attributes Restore_Interval and Restore_Period are required as described below in Table 6-9 (note: these attributes may need to be manually added to the GIS layer). If these attributes do not exist (ie. there is less than 10 attributes), or Restore_Period is zero or negative, there will be no restoration of the Z shape. **Note: If additional attributes have been added to a 2d_vzsh layer for the user's own reference, ensure that the Restore_Interval and Restore_Period attributes are located in columns 9 and 10 otherwise the users additional attributes will be interpreted as restore functions.**

Table 6-9 2D Variable Z-Shape (2d_vzsh) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Variable Z Shape Command			
1	Z	Same as for Table 6-8.	Float
2	dZ	Same as for Table 6-8. 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_Options has not been specified.	Float

No.	Default GIS Attribute Name	Description	Type
3	Shape_Width_or_dMax	Same as for Table 6-8.	Float
4	Shape_Options	**Point:** TRIGGER or TIGGER 1D: Indicates the point is not an elevation point, but a trigger location. The trigger must be given a name using the Trigger_1 attribute. TRIGGER 1D is required if a 1D node water level is used to trigger a 2D variable Z-Shape. The trigger point must be snapped to the 1D node or channel end in the 2d_vzsh layer to achieve this. **Line or Polygon:** Same as for Table 6-8. 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. REPEAT: Specify this option for the variable Z shape to repeatedly function indefinitely based on the below trigger and restore attributes.	Char(20)
5	Trigger_1	**Point:** If Shape_Options is set to TRIGGER or TRIGGER 1D, 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)
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.	Float

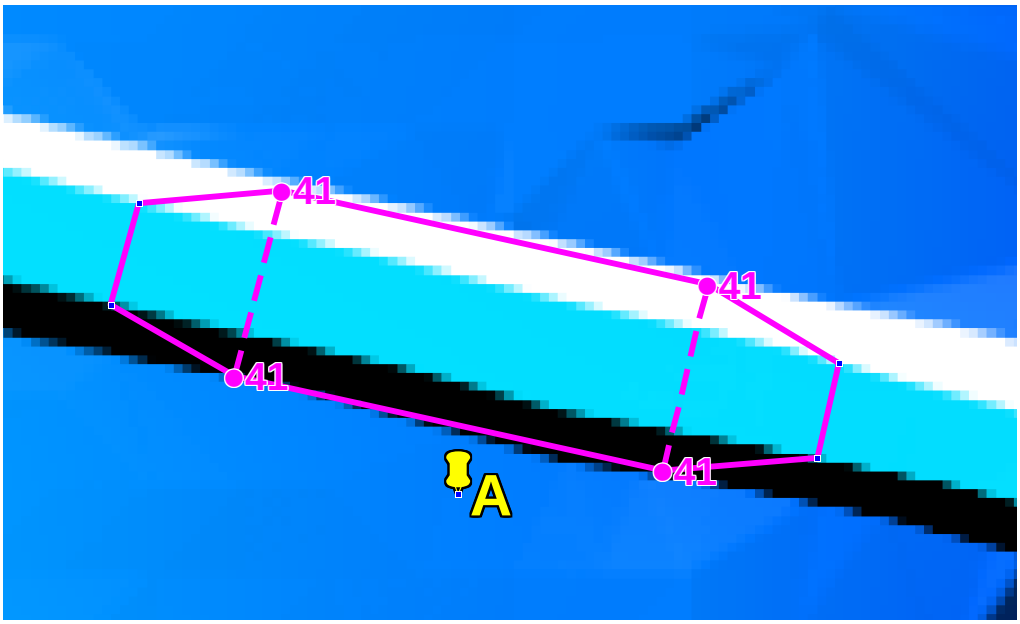
No.	Default GIS Attribute Name	Description	Type
		If Trigger_1 is specified and Trigger_2 is left blank, the water level in metres at Trigger_1 that needs to be reached to trigger the failure. If both Trigger_1 and Trigger_2 are specified, the water level difference in metres between Trigger_1 and Trigger_2 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.	
8	Period	**Point:** Not used. **Line or Polygon:** Time in hours over which the variation in Zpt elevations occurs.	Float
9	Restore_Interval	The time in hours between when the variable Z shape has finished altering the geometry and when to start restoring the Zpts back to their original values.	Float
10	Restore_Period	Time in hours over which the variation in Zpt elevations occurs to restore the Zpts back to their original values.	Float

Example 1: Variable Z Shape Example: Breaching of an Embankment

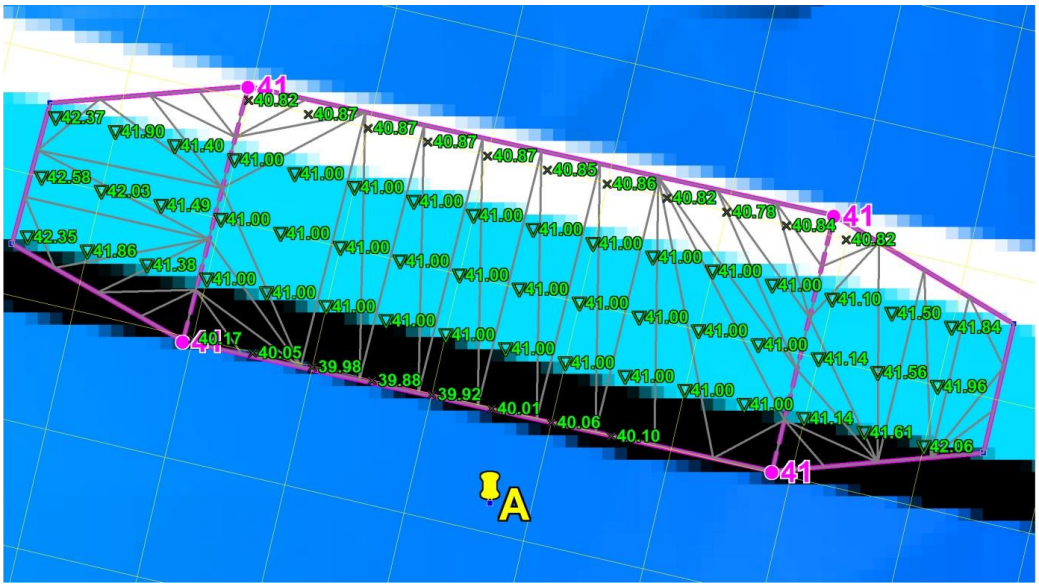
The image below shows an example of a 2d_vzsh layer. The solid magenta line is the polygon, the magenta dashed lines are polylines used to enforce TIN breaklines, and the four magenta points all have an elevation of 41.0m. The Shape_Options attribute for the polygon was set to MIN (this means that the 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 the Shape_Options attribute; and A for Trigger_1 attribute. This sets the point as a trigger point and the "A" is the name of the trigger point. 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.

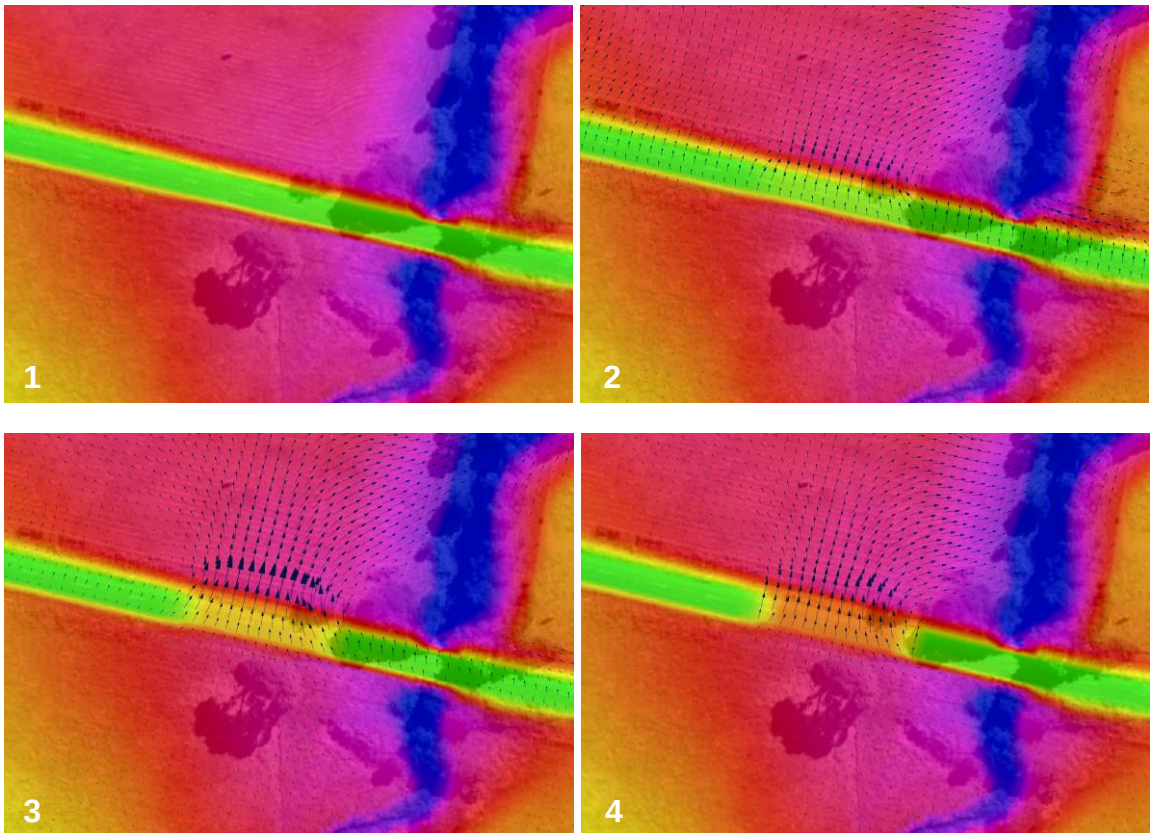


The final eroded Zpt values are based on the TIN created by TUFLOW (the grey triangles in the image below). The central section will be horizontal at 41m sloping up either end to elevations based on the road level. The 2d_vzsh_zpt_check.mif layer is useful to view the Zpts affected by the variable Z 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_Options). 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 modelled breach which occurs 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

view the changes in ground level over time). The images below are from SMS using the Map Output Format == SMS HIGH RES option.



6.8.7 Using Multiple Layers and Points Layers

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 better manage the variety of data these commands sometimes require.

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 a requirement of the shapefile format that the different geometries (points, lines and regions) are in separate shapefiles. The TUFLOW empty template files include the following filename suffixes to differentiate which files are suitable for point, line or region features.

- _P for point features (eg. 2d_zsh_M03_002_P.shp)
- _L for line features (eg. 2d_zsh_M03_002_L.shp)
- _R for region or polygon features (eg. 2d_zsh_M03_002_R.shp)

This is optional for MapInfo users, the different geometries can occur in the same MapInfo file or can be separated if preferred.

A maximum of nine (9) layers per command line is allowed, and each layer is separated by a vertical bar ("|"). For example, to read a Z Shape layer which has both line and points, the command may be:

```
Read GIS Z Shape == gis\2d_zsh_M03_002_L.shp | gis\2d_zsh_M03_002_P.shp
```

6.8.7.1 Point 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 simplifies the datasets making them easier to manage and interrogate.

A layer is treated as a separate points layer if:

1. 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 6-10. The exceptions are for Read GIS FC Shape, which requires the first two attributes. This option requires that the points layer be defined within the command line syntax. For example:

```
Read GIS Z Shape == gis\2d_zsh_M03_002_L.mif | gis\2d_zsh_M03_002_P.mif
```

2. The points file uses the same filename as the associated line or region file with the addition of "_pts" as a suffix to its filename (for example 2d_zsh_M03_002_pts.mif will be automatically associated with the line file 2d_zsh_M03_002.mif). This option is supported for 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).

```
Read GIS Z Shape == gis\2d_zsh_M03_002_L.mif
```

The data processing logic for points layers is outlined below:

1. The specified layer, 2d_zsh_M03_002_L.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:

(i). 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:

```
Read GIS Z Shape == gis\2d_zsh_M03_002_L.mif | gis\2d_zsh_M03_002_P.mif
```

(ii). 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_M03_002_L_pts.mif.

(iii). The first approach (i) above prevails over the second (ii) if both apply.

(iv). If neither (i) or (ii) apply, TUFLOW assumes there is no separate points layer.

Multiple points layers can be specified. The points layer can be referenced in any location except for the first layer within the command line entry. For example, the below syntax will produce an error due to the points file being the first entry.

Incorrect:

```
Read GIS Z Shape == gis\2d_zsh_M03_002_P.mif | gis\2d_zsh_M03_002_L.mif
```

Correct:

```
Read GIS Z Shape == gis\2d_zsh_M03_002_L.mif | gis\2d_zsh_M03_002_P.mif
```

Table 6-10 2D Z (2d_z_) Attribute Descriptions

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

6.9 Land Use (Materials)

6.9.1 Bed Resistance

The bed resistance values for 2D domains are created by using GIS layers containing polygons of different bed resistance. Bed resistance formulation is set Manning's n, Manning's M (1/n) or Chezy using the Bed Resistance Values command in the .tcf file. Chezy can be specified as a direct value or by bed ripple heights. The default and recommended bed resistance formulation is Manning's n. Manning's n values can also be varied with depth, the VxD product or varied with depth using the Log Law formula.

The recommended approach is to use materials to define how the bed roughness varies over the model. 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, depth or VxD varying Manning's n. The material layer can also be used to set land-use hazard categories (see Section 6.9.3.2) and also rainfall losses if using direct rainfall (see Section 6.9).

Material/roughness values are used by TUFLOW during conveyance calculations at the cell mid-sides (refer to Section 6.2 and 6.3). Three alternatives are offered to assign material/roughness values at cell-mid-sides (ZU and ZV):

- The material/roughness value can be averaged from the neighbouring cell centre (ZC) n or M values (the latter being the default prior to Build 2007-07-AA). To use these options see Bed Resistance Cell Sides.

- The bed resistance values can be assigned directly to the cell mid-sides by interrogating the material polygons or raster grid. This is the default option. It 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).

The most common approach is to digitise one or more 2d_mat materials layers (see Table 6-11) and assign Manning's n values to the materials using Read Materials File. This approach allows the easy adjustment of Manning's n values, for example during model calibration or sensitivity testing.

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 data layering of commands in the .tgc file (see Section 4.7).

- Use Set Mat to set the most common material to all cells in a 2D domain.
- Use Read GIS Mat or Read GRID Mat to allocate the remaining material values.

The Read GIS Mat and Read GRID Mat commands may be used as many times as required to further modify the materials in parts of a 2D domain. Each subsequent dataset will overwrite the preceding assigned material value, as described in Section 6.6.

The default material value is zero. As a material value of zero is not allowed, every cell and cell-side must now be assigned a material value using Set Mat, Read GIS Mat and/or Read GRID 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 the previous default of one. The assigned material ID values must be within the range 1 to 32,767.

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.

Table 6-11 2D Material (2d_mat) Attribute Description

No.	Default GIS Attribute Name	Description	Type
Read GIS Mat Command			
1	Material	The material ID value referenced within a Materials File (see Section 6.9).	Integer

6.9.2 Log Law Depth Varying Bed Resistance

At very shallow depths the Manning's n value and/or equation may not be a reliable estimate of bed resistance. The Log Law or "Law of the Wall" approach offers a theoretically based derivation of resistance based on a bed shear analysis. This relationship along with benchmarking against flume test results was used by Boyte (2014) to derive the following equation that varies Manning's n with depth based on the roughness height of the surface. A limiting Manning's n value based on the n value that would normally be applied is also specified to transition to conventional n values at greater depths.

$$n = \max\left[\frac{\kappa y^{\frac{1}{6}}}{\sqrt{g \log\left(\frac{y}{2.71828\, z_0}\right)}}, n_{limit}\right]$$

$$z_0 = \frac{k_s}{30} + \frac{0.11\nu}{U_f}$$

k_s is the roughness height in m
κ is typically in the range 0.38 to 0.42 (recommend 0.4)
y is depth
ν is the kinematic viscosity and is set to 10^{-6} m^2/s
U_f is the friction velocity defined as $\sqrt{Sgy}$ where y approximates A/P and S is the water surface slope
n_{limit} is the limiting n value, ie. the Manning's n value applicable to greater depths

Figure 6-3 illustrates how the equivalent Manning's n varies with depth using the log law for a roughness height of 1mm (0.001m) that would be applicable to a small pebble bed. The different series are the variations in the slope, *S*, where 0.001 is 0.1% slope, 0.02 is 2% slope and 0.1 is a 10% slope. As can be seen there is a significant variation in Manning's n below 2cm (0.02m) and a trend to a n value of around 0.018, with only a minor variation due to slope.

In terms of applying a limiting n value, if, for example, n_{limit} was set to 0.02, then the Manning's n value would not fall below 0.02.

Figure 6-4 shows a comparison using the Log Law versus a constant Manning's n value as developed whilst undertaking the thesis by Boyte, 2014 that investigates the use of the Law of the Wall for direct rainfall modelling using TUFLOW. The flow depths in this example range from 4 to 20cm and the roughness height, k_s, was 3cm.

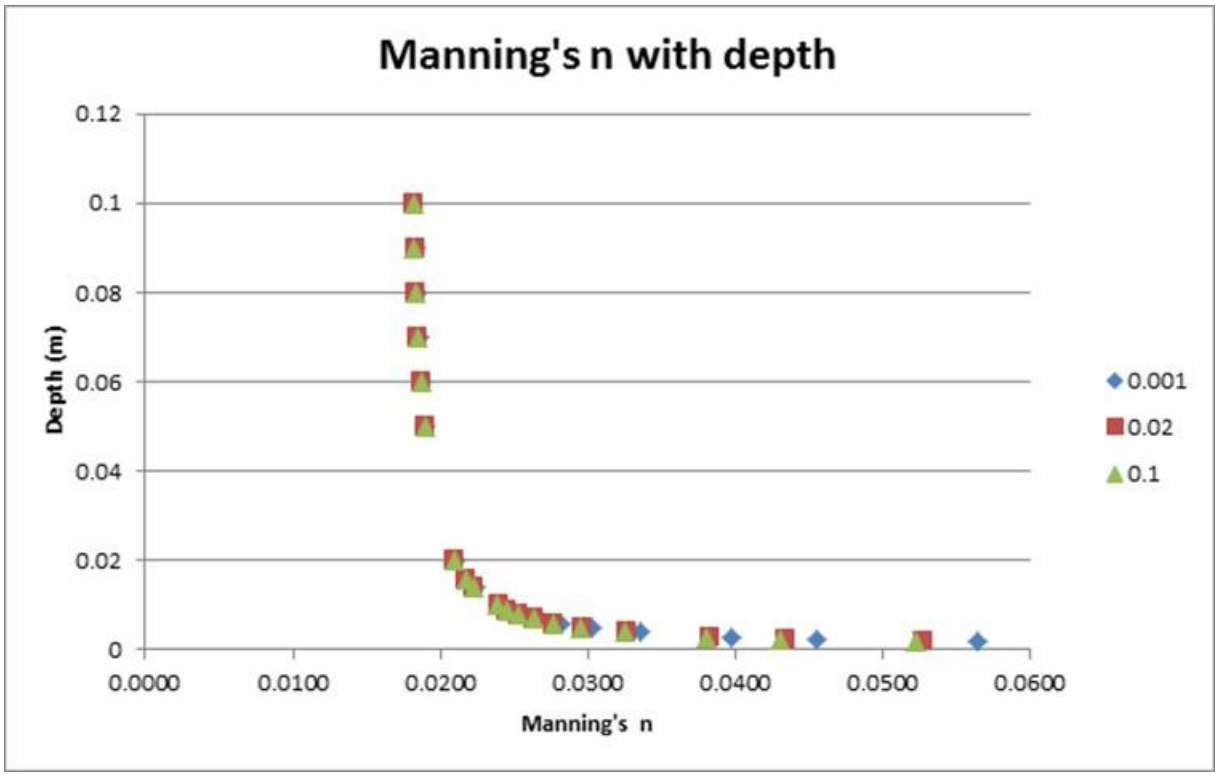


Figure 6-3 Example of Log Law Variation of Manning's n with Depth

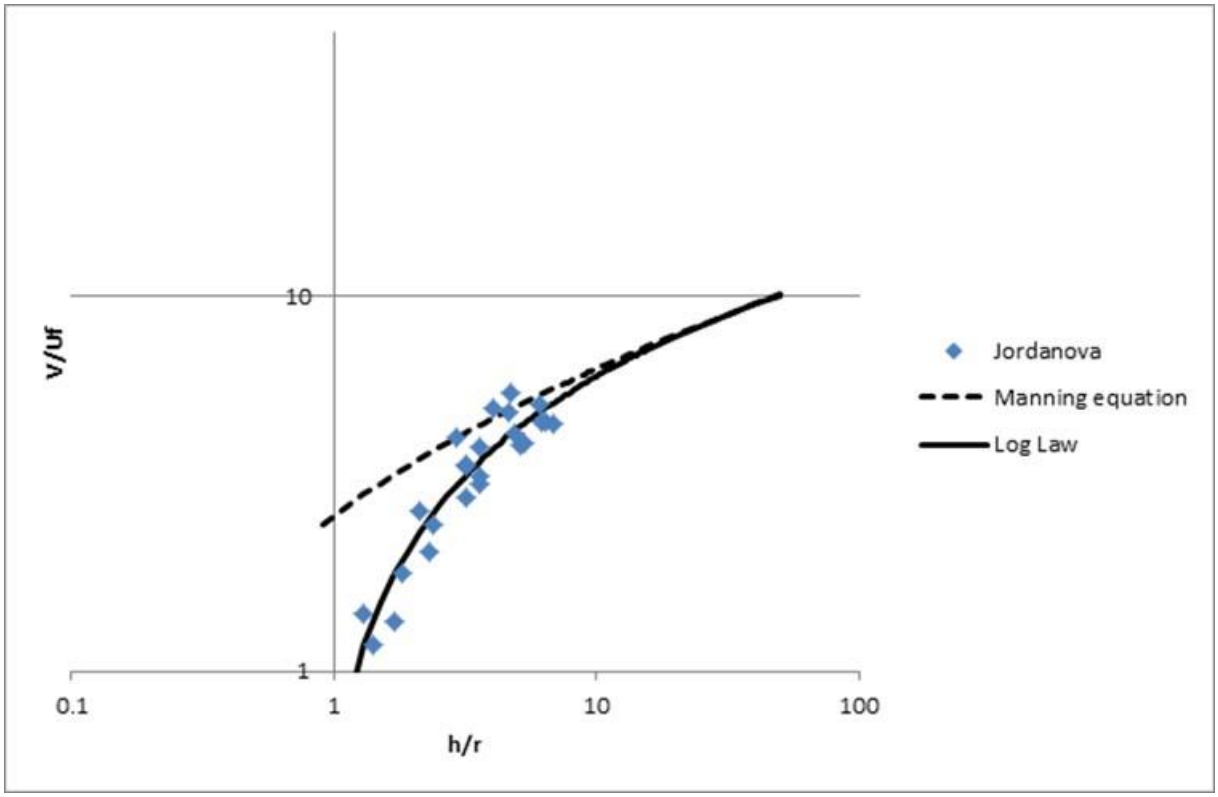


Figure 6-4 Example of Log Law versus Constant Manning's n with Depth

6.9.3 Materials File

Materials file(s) contain information on a material's roughness and optionally rainfall losses if using direct rainfall. The file is referenced within the .tcf file using Read Materials File and can be in one of two formats (.tmf and .csv) as described below. The .csv format offers greater flexibility for future options, and supports curves of Manning's n versus depth.

If a second argument, is provided, this value is used to factor all Manning's n values. For example, to increase all Manning's n values by 10%, enter Read Materials File == My_Materials.tmf | 1.1.

More than one materials file may be specified by repeat occurrences of the command Read Materials File however, most models will use only a single materials file. Any combination of .tmf and .csv files can be used and up to 1,000 materials are allowed in total.

Note: The .tmf format does not offer all materials functionality, whereas the .csv format does. For example, the log law bed resistance option is only available via the .csv format.

6.9.3.1 .tmf Format

The .tmf format is a text file containing Manning's n and other information for different materials (eg. land-uses). The file can contain comments using the "#" and/or "!" delimiters at any location. The format of the materials .tmf file is described in Table 6-12. The first two columns are mandatory and must be specified. All other columns are optional.

Note: The .tmf format does not offer all materials functionality, whereas the .csv format in the following section does.

Table 6-12 Materials .tmf File Format

No.	Description
1	Mat (Material ID) number, which must be an integer.
2	Manning's n value. **Note, if the four values in columns 5 to 8 are specified, the Manning's n value in this column is ignored and not used.**
3	Sets the initial loss in mm if using direct rainfall boundary (Read GIS RF or Global Rainfall BC). Refer to Section 6.9.4.
4	Sets the continuing loss rate in mm/hr if using direct rainfall boundary (Read GIS RF or Global Rainfall BC). Refer to Section 6.9.4.
5	y_1 – The depth below which the Manning's n value n_1 (column 6) is applied.
6	n_1 –The Manning's n value applied below depth y_1 (column 5).
7	y_2 – The depth above which the Manning's n value n_2 (column 8) is applied.
8	n_2 –The Manning's n value applied above depth y_2 (column 7). 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. When specifying values for columns 5 to 8, initial and

	continuing loss values must be specified in columns 3 and 4 as described above (use zero values if not using direct rainfall).
9	Reserved for a future release and should be set to -1.
10	Defines the Storage Reduction Factor (SRF) value. The default is an SRF of 0. Enter 0 to ensure there is no change in 2D cell storage for the material type. See Section 6.11.1 for more information.
11	Defines the Fraction Impervious of the overlying material type. The value entered should be a number from 0.0 to 1.0 where 0.0 is fully pervious and 1.0 is fully impervious. The default is a value of 0.0, assuming that the overlying material is 100% pervious. This feature is used to influence the amount of water that is infiltrated into the ground with the soil infiltration feature. Refer to Section 6.10 for more information. **Note: This does not apply to rainfall losses if applying direct rainfall (see Section 7.4.3.2).**

The .tmf file format is shown in the examples below. See Set Mat, Read GIS Mat, Read Grid Mat and Read RowCol Mat for assigning the material IDs to the 2D domains. These material values may also be used to define bed resistance values across 1D XZ cross-sections (see Section 5.10.1.1.2).

```
! 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
```

To reduce the storage of cells, enter a SRF value in the tenth column as shown in line 7 for Material 15 below. The example shown reduces the storage (surface area) of all cells with Material ID of 15 by 20%. Note that a value of 0 has been entered for the ninth column as this field has been reserved for a future release. If not used, specify 0 to ensure no change in storage for the material.

```
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, 0, 0, 0, 0, -1, 0.2  ! roads with width reduced by 20%
```

To specify a fraction impervious, enter a value between 0.0 and 1.0 in the eleventh column as shown in line 4 for Material 12 below. The example shown partially restricts the rate of infiltration by applying a fraction impervious of 0.1 or 10%.

```
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, 0, 0, 0, 0, -1, 0, 0.1 ! parks and gardens (10% impervious)
13, 0.15, 10, 2  ! sugar cane
14, 0.12, 30, 2.5  ! natural forest
15, 0.02, 2, 0, 0, 0, 0, 0, 0, 0.2  ! roads with width reduced by 20%
```

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

The.csv format is intended to be generated from an .xls file database of materials and associated data in a similar manner to BC databases (with the option of expeditiously using the Excel TUFLOW Tools.xlm macros to export to the .csv format - the Excel TUFLOW Tools.xlm can be downloaded from here).

The format of the materials .csv file is described in Table 6-13.

Note: The .csv format offers access to all features, whereas the .tmf format does not.

Table 6-13 Materials .csv File Format

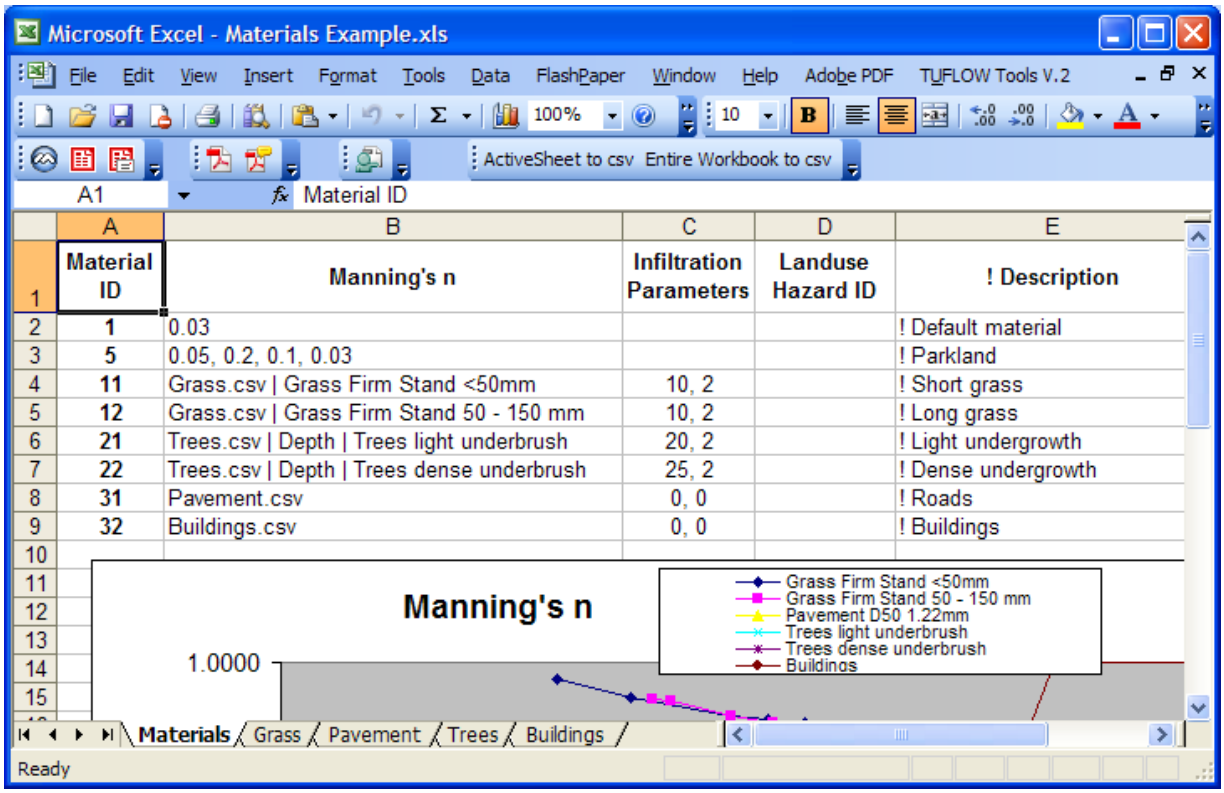
No.	Description
1	Mat (Material ID) number, which must be an integer.
2	Contains information on the bed resistance values (usually Manning's n). The options available are: • Single n value (the n value is assumed constant at all water depths). • Two pairs of depth and n values (see the y_1, n_1, y_2, n_2 option described in Table 6-12 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. • If the first four characters are "VxD:" the Manning's n values for the material will be varied according the velocity times depth product, rather than depth. The VxD versus n values are provided in the same manner as for depths described above. Note that the Manning's n value never increases during the simulation, it can only remain the same or decrease. This feature was provided to allow the user to reduce Manning's n with increasing VxD to represent the "stripping" of river banks in high VxD areas, and then keeping the reduced Manning's n as the flood recedes, hence why the n value cannot increase once decreased during the simulation. VxD was used in preference for simply V to avoid reducing n values at very shallow depths (but high velocities) when the flood first inundates the 2D cells. • If the first four characters are "Log:" the bed resistance as defined by the theoretically derived log law for shallow flows is applied. Three numbers are required after the "Log:" in space or comma delimited format. These are: Ks (roughness height), Kappa (0.3 to 0.4) and the limiting Manning's n value. See Section 6.9.2 for more information.
3	Sets the rainfall loss 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 column values in the .tmf format. See Table 6-12.
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 based on velocity, depth and 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.

No.	Description
5	Defines the Storage Reduction Factor (SRF) value. If no fifth column entry exists, no SRF is applied. The default is an SRF of 0, ie. no change in storage. See Section 6.11.1 for more information.
6	Defines the Fraction Impervious of the overlying material type. The value entered should be a number from 0.0 to 1.0 where 0.0 is fully pervious and 1.0 is fully impervious. The default is a value of 0.0, assuming that the overlying material is 100% pervious. This feature is used to influence the amount of water that is infiltrated into the ground with the soil infiltration feature. Refer to Section 6.10 for more information. **Note: This does not apply to rainfall losses if applying direct rainfall (see Section 7.4.3.2).**

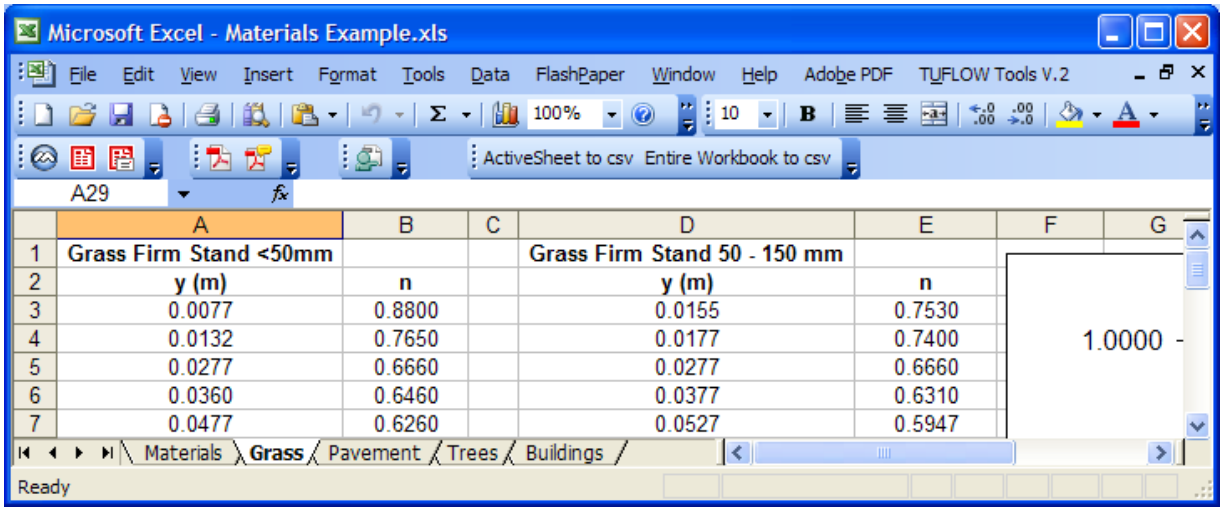
To give a description of the material, this must be done after all inputs for that material, and must be preceded by a "!" or "#". Future releases may read more columns, hence the necessity to use a "!" or "#" to preserve backward compatibility.

In the example shown in Figure 6-5 below:

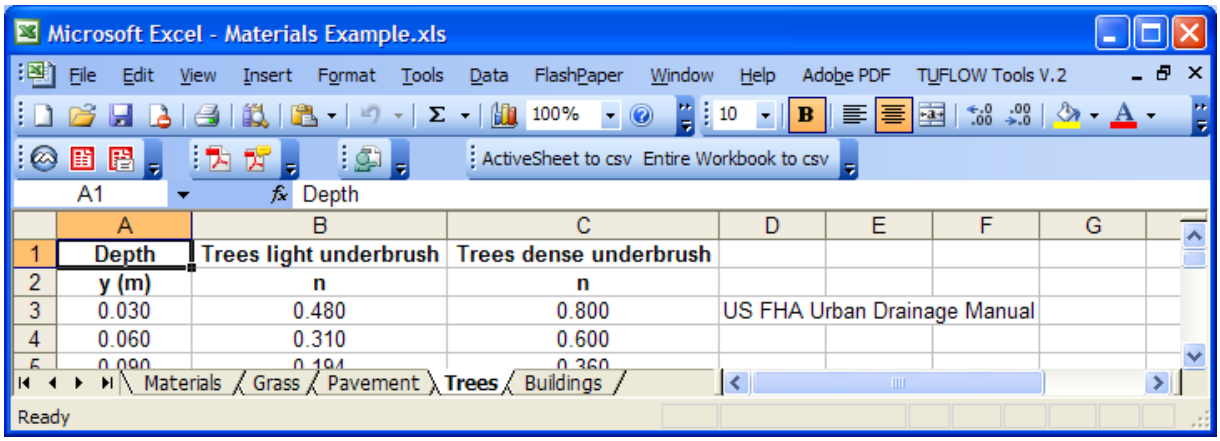
- Material 1 would have a constant n value of 0.03 and no infiltration parameters.
- Material 5 would vary n with depth using the four y_1, n_1, y_2, n_2 values (as per .tmf format approach) and no infiltration parameters.
- Materials 11 and 12 will source n vs y curves from a file called Grass.csv. 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.



Material ID	Manning's n	Infiltration Parameters	Landuse Hazard ID	! Description
1	0.03			! Default material
5	0.05, 0.2, 0.1, 0.03			! Parkland
11	Grass.csv | Grass Firm Stand <50mm	10, 2		! Short grass
12	Grass.csv | Grass Firm Stand 50 - 150 mm	10, 2		! Long grass
21	Trees.csv | Depth | Trees light underbrush	20, 2		! Light undergrowth
22	Trees.csv | Depth | Trees dense underbrush	25, 2		! Dense undergrowth
31	Pavement.csv	0, 0		! Roads
32	Buildings.csv	0, 0		! Buildings



Grass Firm Stand <50mm			Grass Firm Stand 50 - 150 mm	
y (m)	n		y (m)	n
0.0077	0.8800		0.0155	0.7530
0.0132	0.7650		0.0177	0.7400
0.0277	0.6660		0.0277	0.6660
0.0360	0.6460		0.0377	0.6310
0.0477	0.6260		0.0527	0.5947



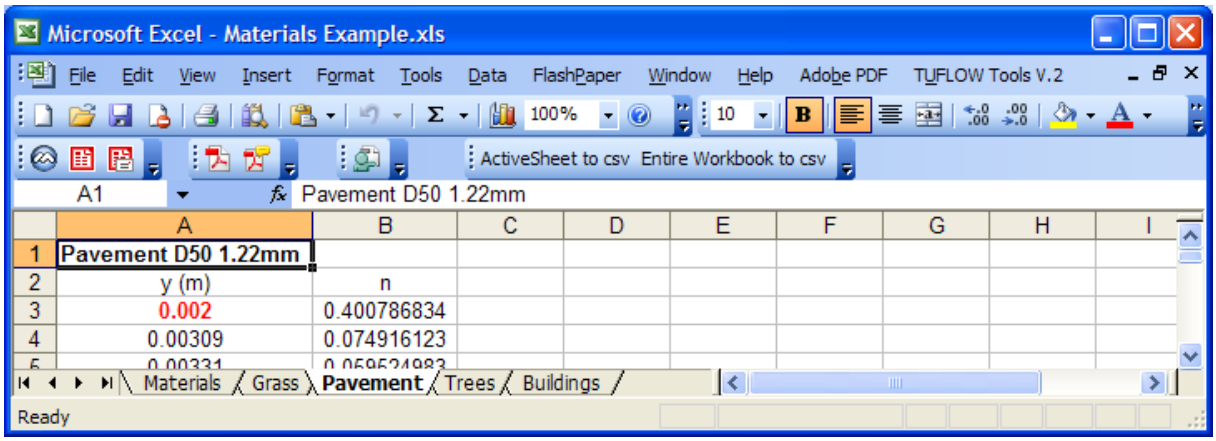


Figure 6-5 Example of Materials .csv File Format

6.9.4 Rainfall Losses

Rainfall losses applied through the Materials Definition file (.tmf or .csv format) remove the loss depth from the rainfall before it is applied as a boundary on the 2D cells. Rainfall losses are ideal for modelling situations in which water is prevented from reaching the ground, such as through the interception by trees.

Note that the rainfall losses are different to the ILCL infiltration losses that can be applied using the .tsoilf file (refer to 0). The ILCL soil infiltration losses will infiltrate ponded water into the ground. It is possible to use both methods in the same simulation – for example, rainfall that doesn't reach the ground would be modelled as a material IL/CL (applied as a loss to the rainfall) and infiltration into the ground as IL/CL via soil types. The log file (see Section 12.6) will report the material and soil properties separately as shown in the example below:

Example Material Properties:

```
#4 - Material 4:
Fixed Manning's n = 0.030
IL = 1.0mm, CL = 0.0mm/h
Landuse Hazard ID not set.
SRF (Storage Reduction Factor) = 0.
Fraction Impervious = 0.
```

Example Soil Properties

```
#1 - Soil 1 [based on pre-defined soil type SAND]:
Suction = 49.5 mm
HydCond = 117.8 mm/hr
Porosity = 0.417
Initial Moisture = 0.2
Soil Capacity = 0.217
```

Specifying the "fraction impervious" on the material allows the materials and the soils to be independent (ie. the same soil can be present under both road and forest). **This fraction impervious only applies to the infiltration into the soil and not to the rainfall losses.**

6.10 Soil Infiltration

Three methods are available to infiltrate ponded water into the ground:

- Green-Ampt;
- Horton; and
- Initial Loss/Continuing Loss (ILCL).

All methods monitor the amount of water infiltrated and if the ground becomes saturated, infiltration ceases. The amount of water that can be infiltrated depends on:

- The loss approach and soil loss parameters;
- The depth to groundwater or an impervious layer;
- The soil's porosity and initial moisture; and
- The faction impervious value of the overlying material layer.

Only wet 2D cells can infiltrate water into the ground. If using soil infiltration the default ground water depth is infinite, see Section 6.10.5 for more details on ground water level / depth.

The amount of water that enters the soil is also dependent on the fraction impervious value of the overlying material layer. The default is that the overlying material is 100% pervious (ie. 0% impervious). However, if, for example, a concrete parking lot overlies a sandy soil, the imperviousness of the parking lot can be specified as 100% to totally, or 90% to partially restrict the rate of infiltration. This is described in the materials file in Section 6.9.

6.10.1 Green-Ampt

The Green-Ampt approach varies the rate of infiltration over time based on the soil's hydraulic conductivity, suction, porosity and initial moisture content. The method assumes that as water begins to infiltrate into the soil, a line develops differentiating between the "dry" soil (with moisture content θ_i) and the "wet" soil (with moisture content equal to the porosity of the soil η). As the infiltrated water continues to move through the soil profile in a vertical direction, the soil moisture changes instantly from the initial content to a saturated state. This concept is presented in Figure 6-6.

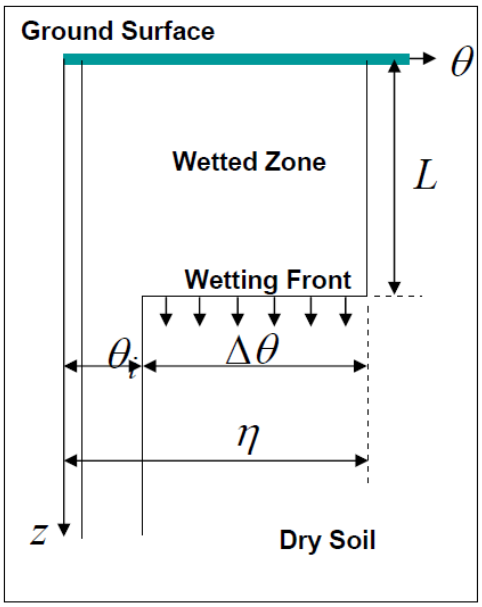


Figure 6-6 Green-Ampt Model Concept[3]

The basic form of the Green-Ampt equation is expressed as follows:

$$f(t) = K\left(1 + \frac{\Delta\theta(\varphi + h_0)}{F(t)}\right)$$

Where:

t is time
K is the saturated hydraulic conductivity
$\Delta\theta$ is defined as the soil capacity (the difference between the saturated and initial moisture content)
φ is the soil suction head
h_0 is the depth of ponded water
F(t) is the cumulative infiltration calculated from:

$$F(t) - \Delta\theta(\varphi + h_0)\ln\left(1 + \frac{F(t)}{\Delta\theta(\varphi + h_0)}\right) = Kt$$

United States Department of Agriculture (USDA) soil types have been hardwired into TUFLOW and are presented in Table 6-14 along with the soil parameters. Alternatively, it is possible to define a customised soil type by specifying user defined values as shown in Table 6-15.

[3] Figure courtesy of University of Texas http://www.crwr.utexas.edu/gis/gishydro99/class/perales/project.htm

Table 6-14 USDA Soil types for the Green-Ampt Infiltration Method[4]

USDA Soil Type	Suction		Hydraulic Conductivity		Porosity
	mm	inches	mm/hr	in/hr	Fraction
Clay	316.3	12.453	0.3	0.012	0.385
Silty Clay	292.2	11.504	0.5	0.020	0.423
Sandy Clay	239.0	9.409	0.6	0.024	0.321
Clay Loam	208.8	8.220	1.0	0.039	0.309
Silty Clay Loam	273.0	10.748	1.0	0.039	0.432
Sandy Clay Loam	218.5	8.602	1.5	0.059	0.330
Silt Loam	166.8	6.567	3.4	0.134	0.486
Loam	88.9	3.500	7.6	0.299	0.434
Sandy Loam	110.1	4.335	10.9	0.429	0.412
Loamy Sand	61.3	2.413	29.9	1.177	0.401
Sand	49.5	1.949	117.8	4.638	0.417

[4] Rawls, W, J, Brakesiek & Miller, N, 1983, 'Green-Ampt infiltration parameters from soils data', Journal of Hydraulic Engineering, vol 109, 62-71.

6.10.2 Horton

The Horton approach to infiltration uses the following equation:

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

Where:

f_0 is the initial infiltration rate in mm/hr or inches/hr
f_c is the final (indefinite) infiltration rate
t is time in hours (period of time that the cell is wet)
k is the Horton decay rate.

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

Figure 6-7 below shows an example of how the infiltration rate varies over time for f_0 equal to 3, f_c equal to 1 and k equal to 0.1.

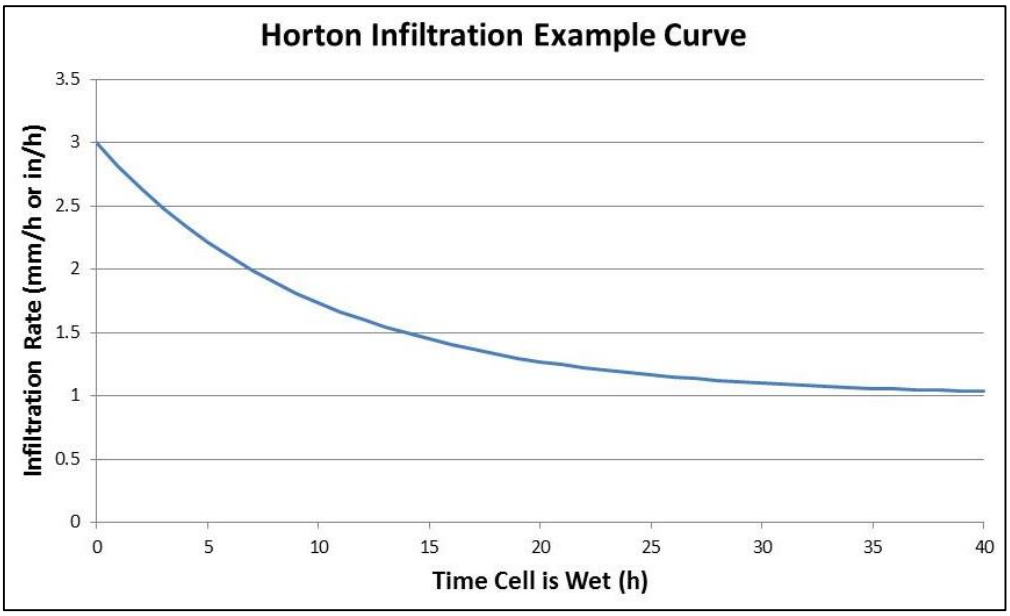


Figure 6-7 Example of Horton Infiltration Rate over Time

6.10.3 Initial Loss/Continuing Loss (ILCL)

The Initial Loss/Continuing Loss method is a more simplistic approach compared to the Green-Ampt and Horton infiltration methods. The ILCL method infiltrates water based on an initial amount then at a constant rate.

Note that the IL/CL infiltration is totally separate to the IL/CL materials values used to generate excess rainfall for direct rainfall simulations (refer to Section 6.9.4 for further information).

6.10.4 Soils File (.tsoilf)

Soils infiltration is applied to the model by defining a soils (.tsoilf) file which is read into the .tcf using the command Read Soils File. Table 6-15 presents the parameters of the .tsoilf file and Figure 6-8 shows an example of a completed file. It is similar to the materials file where you assign an integer ID to each soil, define the infiltration method (Options are "NONE", "GA", "HO" and "ILCL") followed by the soil parameters as the remaining values. The Porosity (saturated moisture content), Initial Moisture (fraction of the soil that is initially wet) and Max Ponding Depth are all optional with default values of 1.0, 0.0 and 0.0 respectively.

Table 6-15 .tsoilf Parameters

Column No.	Infiltration Method				
	No Infiltration	Green-Ampt		Horton	Initial/Continuing Losses
1	Soil ID	Soil ID	Soil ID	Soil ID	Soil ID
2	NONE	GA	GA	HO	ILCL
3		USDA Soil Type (see Table 6-14)	Suction (mm or in)	Initial Loss (mm or inches)	Initial Loss (mm or inches)
4		Initial Moisture (Fraction)	Hydraulic Conductivity (mm/h or in/h)	Initial Loss Rate (f_0) (mm/h or in/h)	Continuing Loss (mm/h or in/h)
5		Max Ponding Depth (m)	Porosity (Fraction)	Final Loss Rate (f_c) (mm/h or in/h)	Porosity (Fraction)
6			Initial Moisture (Fraction)	Exponential Decay Rate (k) (h^{-1})	Initial Moisture (Fraction)
7			Max Ponding Depth (m or ft)	Porosity (Fraction)	
8				Initial Moisture (Fraction)	

Note for the Green-Ampt method, the initial moisture and porosity values in Table 6-15 above are fractions. The soil capacity is defined as the difference between the saturated moisture content (porosity) and the initial moisture content, hence the initial moisture should not exceed the porosity otherwise the soil capacity is set to zero and no infiltration will occur for that soil type. A WARNING 2508 is issued if this occurs.

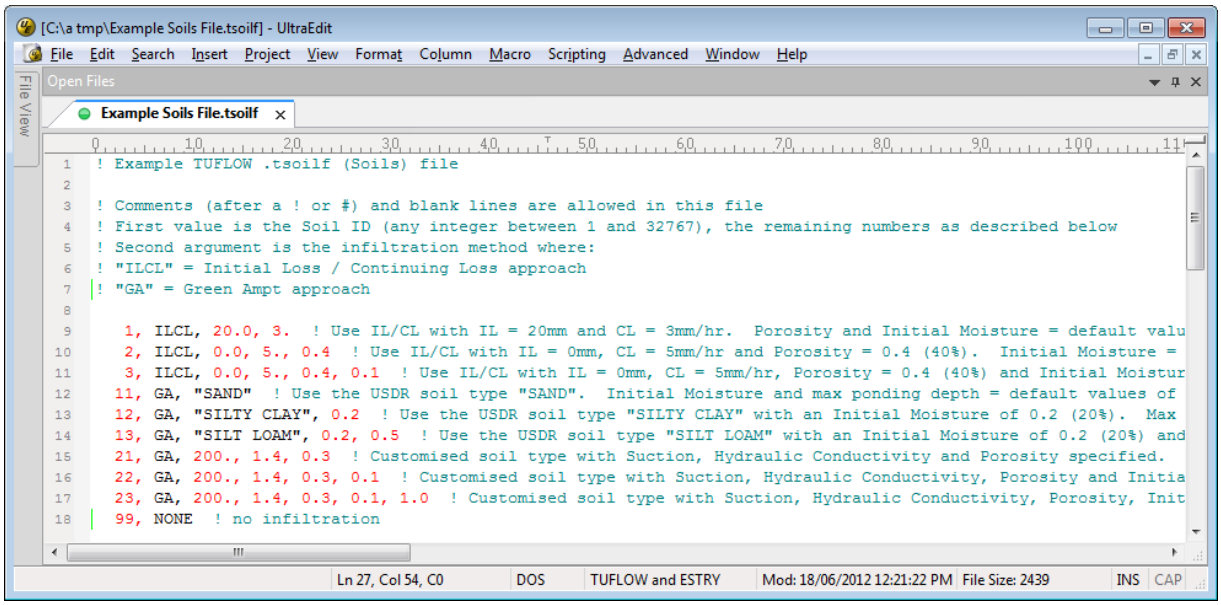


Figure 6-8 Example Soils .tsoilf File Format

One or more soils need to be specified globally and/or via GIS layers/raster grids to activate the infiltration feature. The first attribute of the GIS layer/s must be the Soil ID referenced within the .tsoilf file, in the same way that a 2d_mat layer references a Material ID stored within the materials .tmf or .csv file.

Table 6-16 2D Soil (2d_soil) GIS Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Soil Command			
1	Soil	The soil ID value referenced within a Soils File (see Section 6.10.4).	Integer

Each soil type can have a different infiltration method including a no infiltration option, and have different infiltration parameters as shown in Table 6-15. Other parameters that can optionally be set are:

- The imperviousness of the surface (see Fraction Impervious parameter within the materials definition in Table 6-12 or Table 6-13); and
- Groundwater or impervious level beneath the ground surface (refer to Section 6.10.5).

Optional .tgc commands that can be used are Set Soil, Read GIS Soil and Read GRID Soil. Note that 2D infiltration is activated by the occurrence of one of these commands. If none of the commands exists for a 2D domain, soil infiltration does not occur for that domain.

If soils are specified, the soil ID for each cell are written to the 2d_grd_check layer. Two Map Output Data Types are available to visualise soil infiltration:

- "CI" for Cumulative Infiltration over time in mm or inches.
- "IR" for Infiltration Rate in mm/hr or inches/hr.

The Soil ID at each cell is written to the 2d_grd_check layer. The .tlf file contains the parameters for each Soil ID.

6.10.5 Groundwater

It is also possible to define a groundwater depth (GWD) or a groundwater level (GWL). Any infiltrated water causes the groundwater level to rise. The rate of rise is influenced by the soil porosity and initial moisture, for the Green-Ampt method, these attributes are required, they can optionally be specified for ILCL and Horton infiltration as described above. If the groundwater level rises to the ground surface, no additional infiltration will occur. The default setting is for an infinitely deep groundwater level. Only the vertical filling of the groundwater table can occur (ie. there is no horizontal movement of groundwater, ground water does not fall over time).

The depth to ground water or ground water level may be specified globally and/or via GIS layers/ASCII grids to activate the feature using one or a combination of the following commands:

Set GWD or Set GWL
Read GIS GWD or Read GIS GWL
Read GRID GWD or Read GRID GWL

If both a ground water level and ground water depth are specified the higher level is applied. For example if a GWL of 1.0m is applied to a cell with elevation of 2.5m and a GWD of 1.0 is applied the starting ground water level will be 1.5m (2.5m elevation – 1.0m GWD).

The groundwater feature may only be used in conjunction with at least one of the soils infiltration methods described in Section 6.10. **Consequently, the .tgc commands used to define the groundwater depth / level must be read in following (below) at least one of the commands used to define the soil type:** Set Soil, Read GIS Soil or Read GRID Soil.

The depth to groundwater (in metres or feet) from the ground surface can be visualised over time by adding "**dGW**" to the Map Output Data Types.

Table 6-17 2D Groundwater (2d_gw) GIS Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS GWD or Read GIS GWL Command			
1	Groundwater	The ground water level in metres above datum (if using the command Read GIS GWL) or depth below ground surface in metres (if using the command Read GIS GWD) of the groundwater table at the start of the model simulation.	Float

6.11 Cell Modification

6.11.1 Storage Reduction (2d_srf)

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

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

1 Using the Set SRF and Read GIS SRF .tgc commands. The 2d_srf layer has only one attribute, a float or real value nominally called SRF.

2 Assigned to materials .tmf /.csv files as the 10th and 5th column values respectively (refer to Table 6-12 and Table 6-13).

3 Using the .tgc command Read GRID SRF to assign SRF values from an ASCII grid.

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

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

If a large reduction in cell storage is applied for stability reasons it may be necessary to also reduce the cell conveyance using a higher roughness or Cell Flow Width (see Section 6.11.2). For example if a 10m cell size model has a SRF value of 0.99, without any SRF the cell area is 100m^2 (10m x 10m), with a SRF of 0.99 this would be reduced to 1m^2. If no reduction in cell flow width or hydraulic roughness is applied, a significant volume of flow may enter the cell between timesteps, this would cause the water level in the cell to jump a large amount, potentially leading to oscillations in water levels.

6.11.2 Cell Width Factor (CWF)

The .tgc commands Set CWF, Read GIS CWF and Read Grid CWF can be used to adjust the 2D cell flow widths (in the same manner as for 2D flow constrictions (refer to Section 6.12.2). The CWF is a factor, for example 0.1 will limit the flow width to 10%. The changed flow width applies to all depths. When reading from a polygon object in GIS format, any cell sides that fall within the polygon will have the width factor applied. These factors can be reviewed within the uvpt_check file.

6.11.3 Form Loss Coefficient (FLC)

The .tgc commands Set FLC, Read GIS FLC and Read Grid FLC can be used to apply an additional energy loss at the 2D cell side (in the same manner as for 2D flow constrictions). The form or energy loss can be applied as fixed values or on a form loss per unit length basis. The advantage to applying the FLC on a per unit length basis is that it makes these inputs independent of the 2D cell size when using regions (polygons). As such, if the 2D cell size is changed the same energy loss will be applied

to both models over the area of the region. This per unit length this approach conforms with the existing FLC input for region objects in FC (flow constriction) and Layered FC GIS layers as mentioned below.

Section 6.12 provides a detailed discussion about the application of additional form losses.

One important clarification for form loss values applied in this manner is that the form loss value for a polygon layer is applied to each cell side. For the flow constriction and layered flow constriction layers, this value is a form loss per unit length in the direction of flow as described in 6.12.2.

6.12 2D Hydraulic Structures

6.12.1 Introduction

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 flow constrictions.

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 with a range of types available, as described in Section 5.7.3.

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 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. 2D schemes still however require the introduction of additional form losses since they do not model losses in the vertical or fine-scale horizontal effects (such as around a bridge pier). See Syme (2001b) 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, Bradley, 1978).**
- **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 9.3.3). Upstream and downstream water levels may also be specified using PO points, or extracted 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 methods.
- Using publications such as Hydraulics of Bridge Waterways (Bradley, 1978), 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 subtracting this term from the calculated head difference $(h_1 - h_2)$.

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

- 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 6-18 lists the recommended approaches for modelling 1D and 2D structures using TUFLOW. This is discussed further in the following paragraphs.

Table 6-18 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

1D structures are discussed in 5-17. 1D modelling is the preferred approach where the total structure width is less than of one or two 2D cells. Entry and/or exit are defined for each structure. Testing has shown that these losses may need to be reduced where the structure width is significant compared with the cell size (Syme 2001b).

1D structures can be linked to the 2D domain using either an SX or HX connection, as outlined in Section 8.2. The influence of these connection types on the modelled flow behaviour is shown in Figure 6-9.

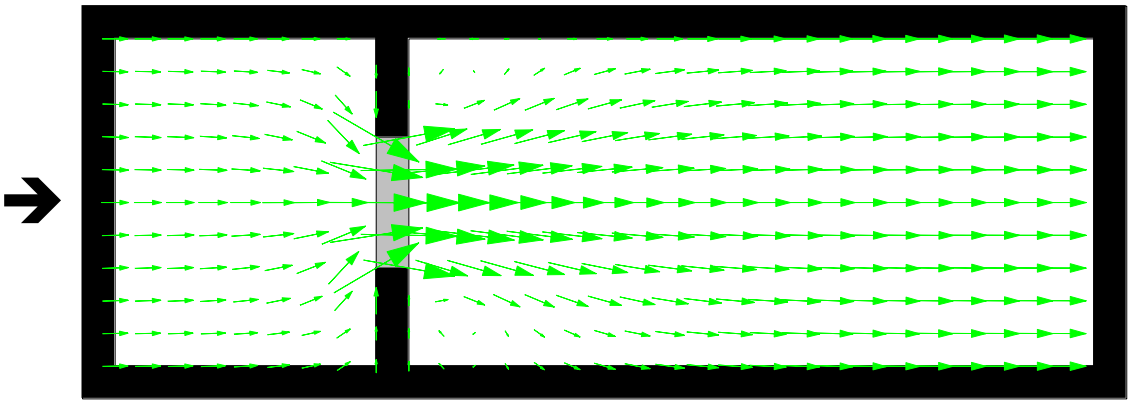
SX Link: Momentum is not transferred into or out of the 1D element to/from the 2D domain. "Suppressed" flow patterns in the 2D domain occur at the structure outlet when using 1D elements, especially if the structure width is significant compared with the cell size. The water tends to spread, rather than jet out as there is no inertia across the link. The effect of this is illustrated in Figure 6-9, 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.

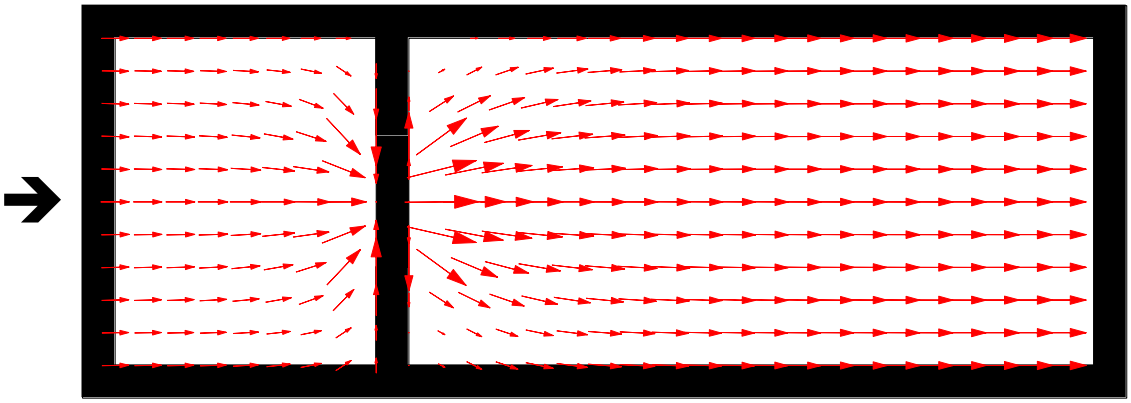
HX Link: Momentum is not transferred into or out of the 1D element to/from the 2D domain, however the velocity field across the HX link is assumed to be undisturbed. Provided the HX link is appropriately located (ie. perpendicular to the flow field) this produces the effect of preserving momentum as illustrated by the dark blue arrows (bottom image in Figure 6-9). Use of HX links at a structure may require a smaller 1D timestep than that required by a SX link.

2D Approach

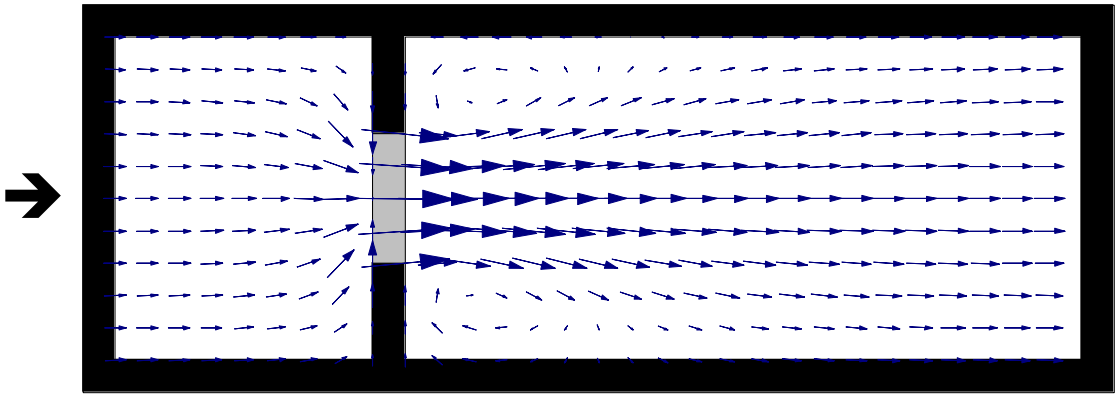
2D representation of structures is preferred where the total structure width is greater than one or two 2D cells. The flow area must be adequately represented by the 2D Zpts and any adjustments to cell widths (see Section 6.12.2). The head drop across the structure during different flow regimes should be validated against other methods and/or literature. Some additional form losses are normally required to achieve correct head drop (see Syme, 2001b). Momentum is transferred through the structure as shown in the top image (green arrows) in Figure 6-9, 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 6-9.



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 6-9 Different Flow Patterns from 2D FCs and 1D/2D Links when Modelling a Submerged Culvert

6.12.2 2D Flow Constrictions (2d_fcsh and 2d_fc Layers)

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

- Placing a lid (obvert or soffit) on the cell side.
- Changing the flow width of the cell side.
- Adding additional form (energy) losses.
- Including side wall friction ("BC" FC_Type only).

Flow constrictions can be digitised within GIS layers and read into the .tgc file (not the .tcf file) using the command Read GIS FC Shape. 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 can be 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 6-19 describes the different 2d_fcsh layer attributes as used by Read GIS FC Shape. If Write Check Files is set in the .tcf then the _fcsh_uvpt_check file will be created to allow you to cross-check that the changes to the cell-sides are as intended. The _fcsh_uvpt_check file contains information on the cell sides modified by the Read GIS FC Shape command. Details on the properties at ALL cell sides can be found in the _uvpt_check file.

Point objects associated with the 2d_fcsh layer can be placed in a separate layer. In this case, only the first two attributes are required. This is discussed in Section 6.8.7.1

The original Read GIS FC continues to be supported and its attributes are documented in

Table 6-20. If any 2d_fc inputs are included in the model and Write Check Files is set, then the _fc_check file will be created.

Table 6-19 Flow Constriction Shape (2d_fcsh) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS FC Shape Command			
1	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
2	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
3	Shape_Width_or_dMax	**Point:** Not used. **Line:** See same attribute in Table 6-8 for 2d_zsh layers. **Polygon:** Not used.	Float
4	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)
5	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 option is specified in Bed Resistance Values.) • "BD" for Bridge Deck • "FD" for Floating Deck	Char(2)
6	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

No.	Default GIS Attribute Name	Description	Type
7	FLC_or_FLCpm_below_Obv	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.). This parameter should be used as a calibration parameter. **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.** 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
8	FLC_or_FLCpm_above_Obv	Form loss coefficient to be applied above the FC obvert. See FLC_below_Obvert attribute above for more information.	Float
9	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	Float

No.	Default GIS Attribute Name	Description	Type
		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.*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. This parameter can be used as a calibration parameter to fine-tune the energy losses across a bridge or floating structure. Ignored for blank FC_Type.	
10	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 6-20 Flow Constriction (2d_fc) Attribute Descriptions

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

No.	Default GIS Attribute Name	Description	Type
Read GIS FC Command			
1	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 is only available if Manning's n option is specified in Bed Resistance Values.) • "BD" for Bridge Deck • "FD" for Floating bridge Deck	Char(2)
2	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).	Float
3	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). Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.	Float
4	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.	Float
5	v_width_factor	Width constriction factor in the Y-direction. See description above for u_width_factor.	Float
6	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.	Float

No.	Default GIS Attribute Name	Description	Type
		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. $\Delta h = \zeta_a \frac{V^2}{2g}$	
7	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. Ignored for "Blank" type FC.	Float
8	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.	Float
9	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 • "L" – block left hand side wall • "T" – block top side wall • "B" – block bottom side wall	Char(10)

No.	Default GIS Attribute Name	Description	Type
		Note: the quotes should be omitted.	
10	Invert_2	leave blank (not used as yet)	Char(10)
11	Obvert_2	leave blank (not used as yet)	Char(10)
12	Comment	Optional field for entering comments. Not used.	Char(250)

6.12.2.1 Applying FC Attributes

The following should be noted 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 (Bradley, 1978):

- The TUFLOW 2D solution automatically predicts the majority of "macro" losses due to the expansion and contraction of water through a constriction, or around 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 or the command Read GRID 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 (in the direction of flow); 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 and a 2D FCSH to a bridge structure is shown in Figure 6-10 and Figure 6-11. **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 9.3.3).

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.

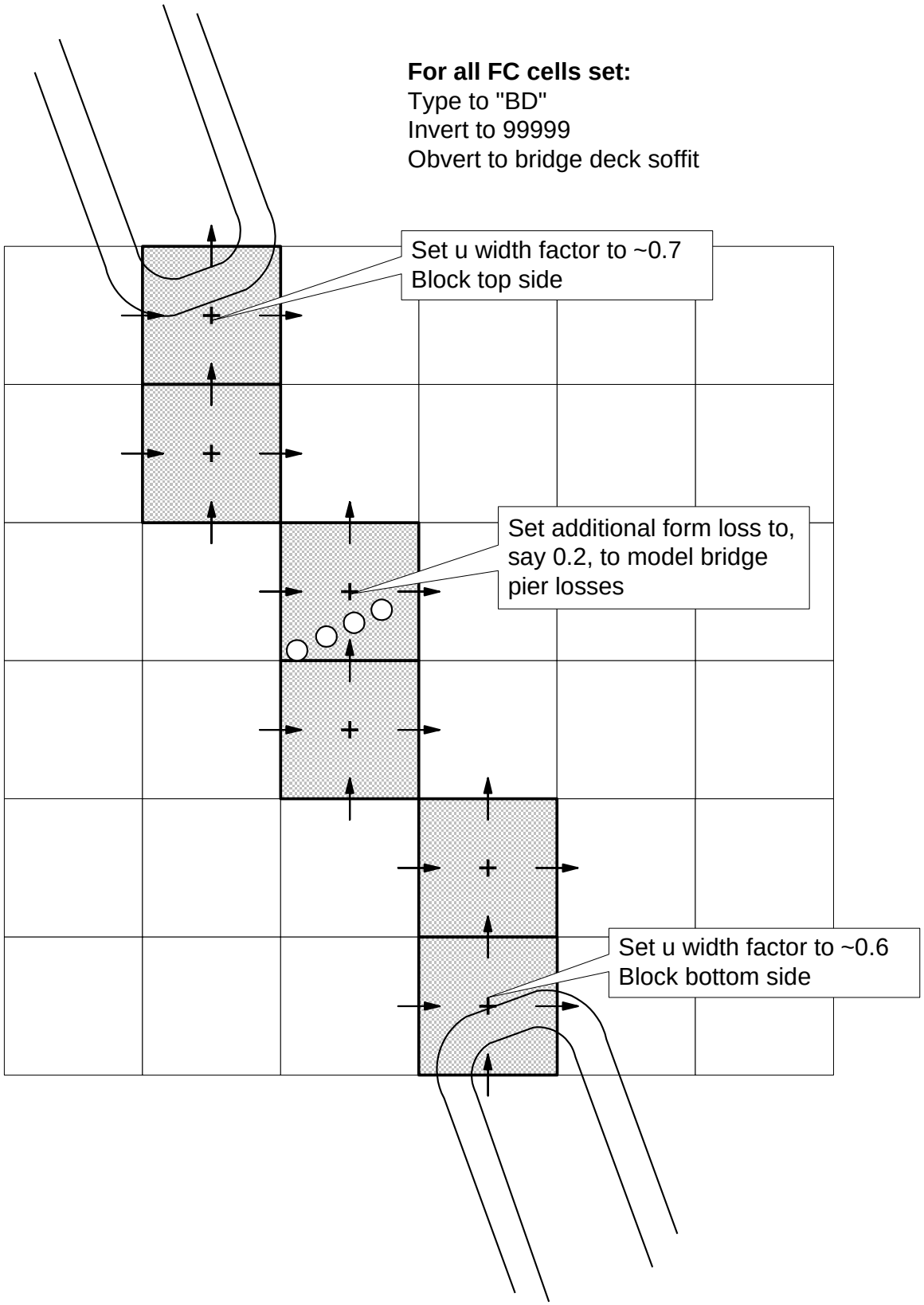


Figure 6-10 Setting FC Parameters for a Bridge Structure

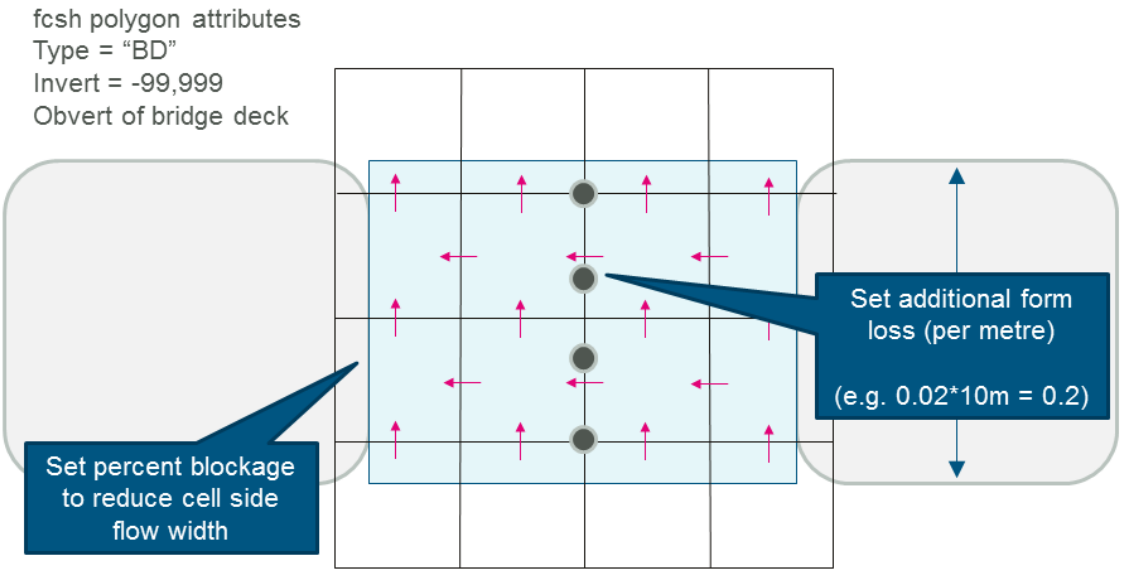


Figure 6-11 Setting FCSH Parameters for a Bridge Structure

6.12.2.2 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 the flow under and over a bridge deck, or a pipeline crossing a waterway.

Four flow constriction layers (not GIS layers) are represented. The lower three layers each have their own attributes. Each layer is assigned its own percentage blockage and form loss coefficient. The top (fourth) layer assumes the flow is unimpeded, representative of 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.

For example, the layers of a bridge structure could be defined as follows.

- **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 is 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. 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.

As of the 2016-03 release two options are available to specify the method in which form losses are applied. To specify the method on a structure by structure basis, populate the Shape_Options attribute (refer to Table 6-21) with either PORTION (the default) to proportion losses to the depth of water, or CUMULATE to accumulate the losses as the depth of water increases.

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

The different approaches will produce different results, therefore either Defaults == PRE 2016 or Layered FLC Default Approach == CUMULATE may need to be set for legacy models.

Alternatively, the .tcf command Layered FLC Default Approach may be specified to define the method to be applied to all structures in the model, noting that the previously described method overwrites this setting for that particular structure. The default approach prior to the TUFLOW 2016-03 version was CUMULATE.

The differences in how the losses are applied between the two methods is explained in the following examples. If the form loss method has been set to PORTION, the losses are applied pro-rata according to the depth of water in each layer using the equation below. Note that Layer 4 (eg. above the bridge deck rails) is always assumed to contribute a zero FLC. If a layer is not flooded the depth for that layer, y_n, is set to zero.

$$\zeta_{total} = \frac{(y_1\zeta_1 + y_2\zeta_2 + y_3\zeta_3)}{y_{total}}$$

$$y_{total} = y_1 + y_2 + y_3 + y_4$$

$$\zeta_n = \text{Layer n FLC}$$

$$y_n = \text{Layer n water depth (set to zero if dry and cannot exceed depth of layer)}$$

$$\zeta_{\text{total}} = \text{Applied overall FLC}$$

If the form loss method has been set to CUMULATE, the losses are accumulated as the water level rises through the layers according to the following equation. This approach was replaced as the default setting for the 2016-03 release due to producing inconsistent results where the bridge is substantially overtopped (drowned out) with a large percentage of the flow occurring through Layer 4, and the overall energy loss reducing with increasing water depth once the structure is submerged.

$$\zeta_{total} = \zeta_1 + \zeta_2 \frac{y_2}{D_2} + \zeta_3 \frac{y_3}{D_3}$$

ζ_n = Layer n FLC
D_n = Depth of layer n
y_n = Layer n water depth (set to zero if dry and cannot exceed depth of layer, D)
ζ_{total} = Applied overall FLC

Bridges are defined as either a line or polygon GIS features using the .tgc command Read GIS Layered FC Shape. The file attribute description is provided in Table 6-21.

Points can be used to vary the invert of the bed, and thickness of each layer in 3D. In this case, four attributes are required as outlined in

Table 6-22. This can for example be used to model an arched bridge or sloping deck. The point objects can be placed in a separate layer to the line or polygon GIS features.

Table 6-21 Layered Flow Constriction Shape (2d_lfcsh) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Layered FC Shape Command			
1	Invert	Performs same function as described for the Z attribute in Table 6-8 and is applied to the Invert elevation values. **Point or Line:** Invert of constriction (m above datum). To leave the Zpt levels unchanged (ie. use the existing Zpt elevations), enter a value of 99999. **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 and the Zpt elevations within the polygon will be adjusted by triangulated around the perimeter at every half cell size.	Float
2	dZ	Performs same function as described for dZ in Table 6-8 and is applied to the Invert elevation values.	Float
3	Shape_Width_ or_dMax	Same as for the same attribute in Table 6-8.	Float
4	Shape_Options	**Point:** Not used. **Line or Polygon:**	Char(20)

No.	Default GIS Attribute Name	Description	Type
		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. CUMULATE: Accumulates losses through each of the layers in the 2d_lfcsh as the depth of water increases. This will overwrite the global setting specified with the .tcf command Layered FLC Default Approach. PORTION: Proportions the losses through each of the layers in the 2d_lfcsh based on the depth of water. This will overwrite the global setting specified with the .tcf command Layered FLC Default Approach. **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.	
5	L1_Obvert	The obvert (soffit) of Layer 1.	Float
6	L1_pBlockage	The percentage blockage of Layer 1. For example, enter 5 for a blockage of 5%.	Float
7	L1_FLC	Layer 1 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.). This parameter should be used as a calibration parameter. **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:**	Float

No.	Default GIS Attribute Name	Description	Type
		• **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.** The form loss coefficient is applied as an energy loss based on the dynamic head equation below where ζ_a is the FLC value. $\Delta h = \zeta_a \frac{V^2}{2g}$	
8	L2_Depth	The depth in metres of Layer 2.	Float
9	L2_pBlockage	The percentage blockage of Layer 2. For example, enter 100 for a solid obstruction such as a bridge deck or pipe.	Float
10	L2_FLC	Layer 2 form loss coefficient. See notes for L1_FLC	Float
11	L3_Depth	The depth in metres of Layer 3.	Float
12	L3_pBlockage	The percentage blockage of Layer 3.	Float
13	L3_FLC	Layer 3 form loss coefficient. See notes for L1_FLC	Float
14	Notes	Optional field for entering comments. Not used.	Char(40)

Table 6-22 Layered Flow Constriction Shape Point (2d_lfcsh…_pts) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
1	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
2	L1_Obvert	The obvert (soffit) of Layer 1.	Float
3	L2_Depth	The depth in metres of Layer 2.	Float
4	L3_Depth	The depth in metres of Layer 3.	Float

6.13 Modelling Urban Areas

The modelling of urbanised floodplains presents many more challenges to hydraulic modellers than those of rural floodplains. It requires careful consideration of the representation of buildings, walls and fences to accurately replicate the overland flow routes. With the advancement of modern computing hardware and software, 2D solutions are increasingly being used over 1D solutions to represent urban areas.

This section of the manual discusses the various methods available in TUFLOW to represent the features within an urban area. It does not aim to identify the most appropriate method, however, discusses the pros and cons to provide the modeller with guidance on selecting the most suitable method for their study.

A paper by Syme (2008) on modelling approaches for buildings and fences can be found in the publications section of the TUFLOW Library.

6.13.1 Buildings

A number of methods can be used for representing buildings in a TUFLOW 2D model. These are:

- Utilising a higher bed resistance (roughness);
- Raising or removing buildings from the 2D cells;
- Reducing the cell storage;
- Restricting the flow using a cell width reduction or additional form loss; or
- Applying a breakline along the upstream faces of buildings.

Utilising a high bed resistance parameter for buildings is a commonly used method to encourage the development of preferential flow paths around buildings during urban flooding scenarios. This method is commonly used and preferred over others as it accounts for the storage that the building provides once it becomes inundated (Syme, 2008). Depth varying roughness is recommended if using this approach in combination with direct rainfall modelling. A low roughness value is recommended at shallow depths, representing the rapid run-off response associated with rainfall on building roofs. High bed roughness is recommended for deeper flows when the building structure impedes overland flow. The application of depth varying roughness is described in Section 6.9.

Buildings may also be “removed” from the 2D domain by categorising the buildings as “Land” cells (see Section 6.7) or raising the Zpt elevations above the predicted flood level. This method may be appropriate when the building/s have been designed to not flood. In this case, care should be taken to ensure the chosen cell size is able to appropriately represent the area of the building and therefore the storage ‘removed’ from the floodplain. Representing buildings in this fashion will not take into account any below ground storage, such as that provided by underground car parks or basement properties. If using a direct rainfall approach (rain on grid), this method is most likely not suitable. Any land (inactive)

cells will have no rainfall applied, meaning the flow volume will be incorrect, artificially high cells (eg. 10m above ground levels) may cause stability issues as flows exit the building.

Flood studies often require the simulation of extreme flood events, where a great number of buildings are expected to become inundated. It may not be appropriate in these cases to "remove" the building entirely from the 2D domain. A more suitable approach may be to raise the Zpt elevations of the building polygons to match the finished floor level, thereby specifying the elevation at which the building is able to flood. This may allow for direct rainfall simulations using a modification of the cell elevations.

The use of a breakline around the edges of the building can be used to prevent the building becoming a flow path, whilst not removing the storage volume. For example three sides of the building could be blocked, this will still allow water to enter the building but not flow through it.

Flow constrictions (see Section 6.12.2) or Cell Width Factor (see Section 6.11.1) may be used to impede the flow of water as it passes through the building. The sides of 2D cells may be partially blocked to represent obstructions such as internal or external walls. Additionally a form loss can be applied using flow constrictions (see Section 6.12.2) or form loss coefficients (see Section 6.11.3).

A storage reduction factor can also be used to reduce the area in the cell available for water using the SRF options (see Section 6.11.1). This can be used in conjunction with the other methods, for example, a building may have the storage reduced by 20% by applying a SRF of 0.2, this could be combined with a Cell Width Factor of 0.8 (80% reduction in flow area through the building).

In the absence of guidelines in the literature it is recommend that the approaches for modelling buildings by sensitivity tested or calibrated if possible.

6.13.2 Roads

Roads typically represent the main flow routes through urban areas. The chosen cell size of the 2D domain is a key factor in ensuring an appropriate number of cells have been used to represent the width of the roads as previously illustrated in Figure 3-1. Choosing too coarse a resolution may result in the roads not being accurately represented and minor flow routes not being represented at all. An unnecessarily fine resolution may result in long simulation times and stability problems if the Courant condition is exceeded. Ideally, it is recommended that the main flow routes be represented with a width of approximately 3-4 cells across.

For overland flood studies where a high proportion of sheet flow is predicted, differentiation between the road and pavement elevations may influence the predicted areas of inundation. A DTM based on LiDAR or ALS data, may not necessarily pick up this detail, therefore requiring manual modification of the topography using breaklines.

A new feature in the 2016-03 release is the ability to apply the Log Law rule for very shallow flow through automatically varying Manning's n. This is of particularly interest for modelling flow over smooth surfaces such as roads and concrete (see Section 6.9.2).

6.13.3 Fences and Walls

Fences and walls can cause significant blockages during a flood event, influencing the direction that flood waters take. Walls act to deflect the path of flood waters and if overtopped, may act as weirs. Fences may partially impede the flow of water and can be prone to becoming blocked with debris. There is also the potential for both walls and fences to collapse during a flood event.

Fences and walls are typically included within a TUFLOW model by digitising a series of breaklines. The 2d_zsh GIS layer permits the width of the breakline to be specified (see Section 6.8.5). A "thin" line modifying only the ZU and ZV (cell side) elevations may be preferred to represent fences, in situations where the width of the cell size is notably greater than the width of the obstruction. A "thin" line does not modify the ZC (cell centre) elevations hence has no impact on the cell storage. A "thick" line on the other hand, modifies the ZU, ZV and ZC Zpt elevations and may be more appropriate to represent wider obstructions such as railway embankments.

The 2d_vzsh GIS layer (see Section 6.8.6) may be used to represent an embankment that collapses during the simulation. The breach may be triggered at a specified time, by the water level at a defined location or the water level difference between two locations exceeding a specified amount.

7 Boundaries and Initial Conditions

Chapter Contents

7 Boundaries and Initial Conditions 7-1

7.1 Introduction 7-2

7.2 Recommended BC Arrangements 7-3

7.3 1D Boundaries (1d_bc Layers) 7-5

7.4 2D Boundaries (2d_bc, 2d_sa and 2d_rf Layers) 7-9

7.4.1 2d_bc Layers 7-16

7.4.2 2d_sa Layers 7-20

7.4.2.1 Approaches to distributing flow 7-20

7.4.2.2 Streamlines 7-20

7.4.2.3 RF Option 7-22

7.4.2.4 Trigger Option 7-22

7.4.2.5 Flow Feature 7-23

7.4.3 Rainfall 7-24

7.4.3.1 Rainfall Overview 7-24

7.4.3.2 2d_rf Layers 7-24

7.4.3.3 Gridded Rainfall 7-25

7.4.3.4 Rainfall Control File (.trfc file) 7-27

7.5 Boundary Condition (BC) Database 7-28

7.5.1 BC Database Example 7-31

7.5.2 Using the BC Event Name Command 7-33

7.6 Initial Conditions 7-35

7.6.1 Initial Water Levels (IWL) 7-35

7.6.1.1 1D Domains 7-35

7.6.1.2 2D Domains 7-36

7.6.1.3 Automatic Initial Water Level (Set IWL == AUTO) 7-37

7.6.2 Restart Files 7-38

7.1 Introduction

This chapter of the manual discusses the available boundary conditions and the boundary condition database. 1D and 2D boundaries are discussed together in this chapter as both access the same database (see Section 7.5), although separate databases can be set up if desired.

The options for applying initial conditions to the 1D and 2D domains of the model are also discussed.

7.2 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 HT (water level-time) or HS (sinusoidal water level) 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 (water level-flow or stage-discharge) 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.

2D water level (HQ, HS and HT) and flow (QT) boundaries should be digitised approximately perpendicular to the flow direction. The boundary can be digitised at any orientation to the 2D grid (ie. not parallel to the grid), and can be defined using a polyline with multiple vertices (ie. it does not have to be a straight line). The reason these boundaries should be digitised perpendicular to flow is that the boundary type forces a uniform water level along the boundary cells. Spurious circulations can occur along the boundary if it is digitised along (as opposed to across) a waterway. This is often noticed by a changing dV value in the output window (refer to Section 12.2) or by viewing the flow directions in the vicinity of the boundary.

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

Table 7-1 provides an overview of the boundaries types typically used for different locations.

Table 7-1 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.	Specify rainfall losses (Section 6.9.4) or soils infiltration (Section 6.10). Alternatively, RF can be used by specifying negative values.

7.3 1D Boundaries (1d_bc Layers)

Boundary conditions for 1D domains are defined using one or more 1d_bc GIS layers. The different types of boundaries and links are described in Table 7-2. Note, links to 2D domains are automatically created from any links specified in the 2d_bc layer (see Section 5.13).

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 7-3.

In addition to points, 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 an H boundary (this includes nodes connected to the 2D domain via a 2D SX). If no suitable nodes are found an ERROR occurs.

Note: It is not possible to assign both a flow type boundary and water level type boundary to the same 1D node. For example a HT and QT boundary on the same node.

Table 7-2 1D Boundary Condition and Link Types

Type	Description	Comments
Water Level Boundaries		
HS	Sinusoidal (Tidal) Water Level (m)	Not currently supported for 1D nodes. Note, this boundary was originally supported in a fixed field format in prior releases. It will be supported in the same manner as 2D HS boundaries in a future release. In the meantime recommend using a HT boundary with the S flag (ie. HTS) to fit a sinusoidal curve through a HT time-series.
HQ	Water Level (**H**ead) 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 (**H**ead) versus **T**ime (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.
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.
Combinations		Any number of water level boundaries can be assigned to the same node. The sum of the water levels is assigned as the boundary. 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. **If you have a 2D SX boundary connected to a node and also have a HT and/or HS boundary at the same node, the 2D SX boundary prevails and no warning is given.**
Flow Boundaries		
QC	Constant Flow (m^3/s)	The original fixed field QC boundary is no longer supported. To setup a constant flow boundary, specify a QT boundary, leave the Source column blank, and enter in a constant value for the Column 2 value in the boundary condition database file (see Table 7-8).
QH	Flow vs Water Level (**H**ead) (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. A region (polygon) can be used to equally distribute the flow hydrograph to all nodes that fall within the region. Any nodes that are an H boundary or are connected to a 2D SX cell are not included. The limit on the number of 1D QT regions assigned to a 1D node is ten (10).

Type	Description	Comments
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 7-3 1D Boundary Conditions (1d_bc) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS BC Command			
1	Type	The type of BC using one of the two letter flags described in Table 7-2.	Char(2)
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. 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.	Char(6)
3	Name	The name of the BC in the BC Database (see Section 7.5). An error will be generated if no name is specified.	Char(50)
4	Description	Optional field for entering comments. Not used.	Char(250)

7.4 2D Boundaries (2d_bc, 2d_sa and 2d_rf Layers)

2D boundary conditions and linkages between the 1D and 2D domains are defined using a combination of one or more 2d_bc, 2d_sa or 2d_rf GIS layers. The different types of boundaries and links are described in Table 7-4.

The GIS layer for a 2d_bc format file may contain points, lines, polylines and regions. The "cross-hair" approach shown in Figure 7-1 is the default method for selecting boundary or link cells. Cells are only selected if the boundary or link line intersects the cell cross-hairs that extend from the cell's mid-sides. As such, if a boundary or link line starts inside a cell, and does not intersect with the cross-hairs, that cell will not be selected.

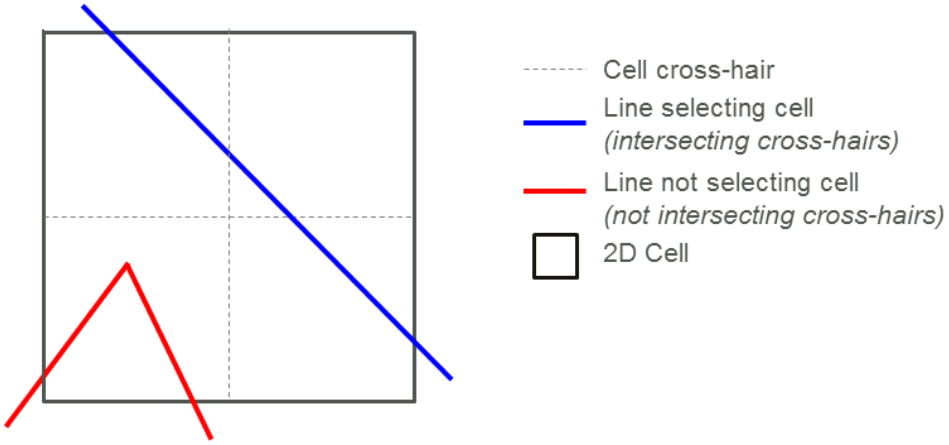


Figure 7-1 Cell Cross-hair Selection Approach

The crosshair approach does not apply to:

- 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).

Conditions have been built into TUFLOW to ensure 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 7-5 (for example, HX 1D/2D interface boundaries and the 2D QT boundaries).

Error checks have been incorporated into TUFLOW during the input pre-processing of 2D boundary conditions, provided Line Cell Selection == Method D (the default) is set. TUFLOW stops with an error if a cell is:

- Assigned a HT or HS that is already a Q, S or HX cell;
- Assigned a 2D or HX that is already any other boundary;
- Assigned a Q or V that is already a H or S; and
- Assigned a ST, RF or SX that is already an H or Q.

Temporally and spatially varied 2D water level boundary can be defined by combining some of the boundary types listed in Table 7-4. This feature is particularly useful for coastal models where the tidal boundary varies in height and phase along the boundary. Also see Blank BC Type.

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.

Table 7-4 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 similar. See Section 8.3.1.
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.
HQ	Water Level (**H**ead) versus Flow (**Q**)	Assigns a water level to the cell(s) based on a water level versus flow (stage-discharge) curve. Alternatively, TUFLOW can automatically generate the HQ curve if a slope is specified using the b attribute (see Table 7-5). The variation in Manning's n with depth feature is taken into account if automatically calculating the stage-discharge curve.
HT	Water Level (**H**ead) versus **T**ime	Assigns a water level to the cell(s) based on a water level versus time curve.
HX	Water Level (**H**ead) from an e**X**ternal Source (ie. a 1D model)	For linking 1D and 2D domains. One or two 1D nodes provide a water level to the 2D every half timestep. TUFLOW automatically creates 1D QX boundaries at the 1D node(s) (see Table 7-2), which also receives flow from the 2D domain

Type	Description	Comments
		every half timestep. 2D HX boundaries are linked to 1D nodes using CN connections (see below and Figure 8-4).
Treatment		Cell(s) can be wet or dry. It is not a requirement that at least one cell is wet. HT lines can be oblique to the X-Y axes. 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.
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 2D2D 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)	The original fixed field QC boundary is no longer supported. To setup a constant flow boundary, specify a QT boundary, leave the Source column blank, and enter in a constant value for the Column 2 value in the boundary condition database file (see Table 7-8).
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 super-elevation or other similar effects occur.
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 QT option, 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.**
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)	Applies a rainfall hyetograph. 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 rainfall losses are applied on a material-by-material basis (see Section 6.9.4). **The double precision version of TUFLOW is recommended for direct rainfall models (see Section 11.4).** RF boundaries have their own command, Read GIS RF and Read RowCol RF, and own GIS layer (see Section 7.4.3.2 and Table 7-7). If a negative rainfall is specified 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 Section 7.4.2 and Table 7-6).

Type	Description	Comments
		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 11.4). When using Read GIS SA RF a rainfall hyetograph is 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. Initial and continuing losses are entered as attributes to the 2d_sa layer (see Table 7-6). 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. The SA Minimum Depth command sets the minimum depth a wet cell must have to apply an SA inflow. This command can resolve a problem that has occurred where large SA inflows onto very shallow, high roughness areas can appear to gradually flow up hill. SA Proportion to Depth proportions the SA inflow according to the depth of water. This feature also enhances SA inflows by applying an inflow in proportion to the depths of water of the wet cells contained within the SA polygon. Where the SA hydrographs have been derived by a hydrologic model that has already included routing effects, this feature will tend to place more inflows in the deeper areas (ie. the creeks, rivers and downstream areas of the SA region), and hence reduce any routing duplication effects. SA Proportion to Depth set to ON is the default setting. Use Defaults == Pre 2012 to maintain backwards compatibility for previous releases. Read GIS Streams can be used to apply the flow along streamlines as opposed to the lowest cells.
SD **SH**	Flow versus Depth or Head (m^3/s)	Extracts the flow directly from the cells based on the depth (SD) or water level (SH) of the cell. Used for modelling pumps or other water extraction. Flow values must not be negative. SD or SH boundaries can be connected to another 2D cell or a 1D node, to model 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 condition database, the Column 1 data would be depth (SD) or water level (SH) 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.
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

Type	Description	Comments
		specifies 2 m^3/s and the ST is applied over four cells, then the total flow applied to the model would be 8 m^3/s. If the total flow required is 2 m^3/s, then an f attribute of 0.25 could be applied so that only 0.5 m^3/s is applied to each cell. Alternatively the Read GIS SA ALL could be used to distribute a flow equally over a number of cells.
SX	Source of flow from a 1D model.	For linking 1D and 2D domains. 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. 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, SD, SD, 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 direction or digitised length of the line is not important. 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. 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. The same node can be connected to each end of a HX line if the water level is not to vary along the line.**
SC	Connection of 2D SD and SH boundaries	Used for connecting 2D SD and SH boundaries to another 2D cell or 1D node (eg. modelling the pumping of water from one location to another). An example 2D pump model is documented in Module 14 of the TUFLOW tutorial.

Type	Description	Comments
Wind Stresses		
WT	Modelling Wind Stresses as a force on the water column	Models time varying wind stresses. 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 7-8). The three columns are: • Time (h) • Wind (m/s) • Direction (degrees relative to East, ie. East = 0º, North = 90º, 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 6.8.6 on Variable Z Shape layers as a simpler 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 7-5). 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], note that this must be read into the TGC file for setting the cell code.
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 which is specified in the geometry control file.

7.4.1 2d_bc Layers

All boundary types listed in Table 7-4 apart from SA and RF are digitised within a 2d_bc layer. The SA and RF boundaries have their own GIS layer and commands, as discussed in Section 7.4. The attributes of the 2d_bc layers are described in Table 7-5. 2d_bc layers are referenced in the .tbc file using the command Read GIS BC.

Up to 10 GIS layers are accepted in a single line for the Read GIS BC command. This is needed for shapefile layers where points and lines are both used (for example, if using HX lines and CN points), requiring multiple layers for the one command. The format is similar to the Read GIS Z Shape command with the different GIS layers separated by the "|" character. For example:

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

Also note that Blank BC Type can be used to set the default BC type to apply when no BC type is specified for an object. This command maybe be repeated throughout the .tbc file to change the default setting for different 2d_bc layers. See Blank BC Type for more details.

Table 7-5 2D Boundary Conditions (2d_bc) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS BC Command			
1	Type	The type of BC using one of the two letter flags described in Table 7-4. Also see Blank BC Type.	Char(2)
2	Flags	Optional flags as follows: **HT, QT, SD, SH, VG, VT:** "S" – Fit a cubic spline curve to the data. **HX:** "A" – 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" – The "S" flag is a legacy and is not recommended unless Adjust Head at ESTRY Interface has been set to ON. "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	Char(3)

No.	Default GIS Attribute Name	Description	Type
		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" – Do not use – not recommended or supported. Applies the original QT boundary, which required flow volume and direction inputs. **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. 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 permits the user to define a maximum change in ZC elevation. 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. **SX:** "U" – No longer recommended or supported – replaced by the SD and SH boundary options. Was originally used to indicate 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 1d_bc "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.	
3	Name	**HS, HT, QT, RF, SD, SH, ST, VG, VT, WT:** The name of the BC in the BC Database (see Section 7.5). **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. **CD, CN, HX, QC, SX, VC, ZP:** Not used.	Char(100)
4	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 7-8). **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). An f value of zero (less than 0.001) is set to one (1).	Float

No.	Default GIS Attribute Name	Description	Type
		CD: The code value (see Table 6-3) to be assigned to cells falling on or within the object. **SX:** n offset for determining cell invert levels for distributing flows. **ZP:** 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 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.	
5	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. This feature is under development and further testing, and may be 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 7-8). **HX, ZP:** 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 6-8). 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
6	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 7-8). **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
7	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 nth 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.	Float

No.	Default GIS Attribute Name	Description	Type
		HS: Incremental adjustment of the amplitude per cell along the boundary. For the nth 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. 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:** 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, (the X-axis left to right is 0°, Y-axis bottom to top is 90°). **CD, CN, RF ST, SX, ZP:** Not used.	
8	B	**HT, QT (with A Flag), 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 7-8). **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. Also see Blank HQ Slope. **HS:** Incremental amount added per cell to the mean water level. **CD, CN, QC, SX, VC, ZP:** Not used. **HX:** Adjusts the WrF value along HX lines to adjust or calibrate the flow rate across a 1D/2D HX link when upstream controlled weir flow occurs. This is similar to using the "a" attribute for applying additional energy (form) loss along the HX line.	Float

7.4.2 2d_sa Layers

A 2d_sa layer is used to define a **S**ource – **A**rea boundary (flow or rainfall vs time), applying the flow directly onto the cells within the digitised polygon. Negative values remove water directly from the cell(s). Table 7-6 lists the attributes for 2d_sa layers which are referenced in the .tbc file using the command Read GIS SA to read in flow hydrographs and Read GIS SA RF to read in rainfall hyetographs, if reading in rainfalls (Read GIS SA RF) four additional attributes are required as outlined in Table 7-6.

7.4.2.1 Approaches to distributing flow

There are a number of options for distributing flows, these are:

- Distributed firstly to the lowest cell and then distributing to between wet cells (Read GIS SA). This is the default option.
- Distributed to 2D cells connected to 1D pits (Read GIS SA PITS).
- Distributed to all cells equally (Read GIS SA ALL).
- Distributed using to streamlines (Read GIS SA STREAM ONLY).

The lowest cells within each SA layer are output in the 2d_sac_check layer.

When distributing the water between the cells there are two options that can be specified in the .tcf. These are:

- SA Minimum Depth, this sets a minimum depth below which flow will not be distributed to the cell (apart from the lowest cell). The default minimum depth is 0.1m
- SA Proportion to Depth, this command proportions the SA inflows according to the depth of water. The default is ON, which proportions the flow according to the depth of water in the cells with deeper cells getting more flow.

The PITS option directs the inflow to 2D cells that are connected to a 1D pit or node connected to the 2D domain using "SX" for the Conn_1D_2D (previously Topo_ID) 1d_nwk attribute. The inflow is spread equally over the applicable 2D cells. An ERROR occurs if no 2D cells are found within the region.

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 (see Section 11.4) version may be needed as this is a similar approach to direct rainfall modelling.

The STREAMS ONLY option used in combination with the Read GIS Streams command. It will only apply the SA inflows to the streamline cells (ie. no non-streamline wet cells in the SA region will receive an inflow). This is discussed further in the following section.

7.4.2.2 Streamlines

Streamlines allow the user to apply SA inflows along the waterways rather than to the lowest cell (when all cells are dry within the SA region). If streamlines have been specified using the .tbc

Read GIS Streams command, SA inflows are distributed along the 2D cells selected by the stream lines within each SA region.

Read GIS Streams can be used one or more times in the .tbc file to define streamline cells. Streamlines are polyline or line objects, usually representing the path of the waterways. One attribute is required being the Stream Order as an integer. Only objects with a Stream Order greater than zero (0) are used by TUFLOW. Therefore, streams that are not to be used for applying SA inflows can be assigned a stream order of 0 (or deleted from the layer). The 2D cells along a streamline are selected as a series of continuous cells in the same manner as any other boundary line.

GIS and other software have the ability to generate streamlines from DEMs, and usually assign a stream order to each stream polyline. If needed, rearrange (or copy) the attributes so that the first attribute is the stream order one.

By default, any wet cells that are not streamline cells are also included in the distribution of the SA inflow. The commands Read GIS SA STREAM ONLY and Read GIS SA STREAM IGNORE provide options for controlling streamline inflows.

The 2d_sac_check layer will show those cells selected as streamline cells.

Table 7-6 2D Source over Area (2d_sa) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS SA Command			
1	Name	The name of the BC in the BC Database (see Section 7.5). **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: Read GIS SA RF Command			
2	Catchment_Area	The contributing catchment area in m^2. 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 zero rainfall to be applied.	Float
3	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 zero rainfall to be applied.	Float
4	IL	The Initial Loss amount in mm.	Float
5	CL	The Continuing Loss rate in mm/hr.	Float
Additional Attributes for the Trigger Option: Read GIS SA Trigger Command			
2	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)

No.	Default GIS Attribute Name	Description	Type
3	Trigger_Location	Trigger_Location is the PO Label in a 2d_po layer (see Section 9.3.3). The 2d_po Type attribute must also be compatible with the Trigger_Type (ie. it must include Q_ or H_ as appropriate).	Char(40)
4	Trigger_Value	Trigger_Value is the flow or water level value that triggers the start of the SA hydrograph.	Float
Additional Attributes for the PO Option: Read GIS SA PO Command			
2	PO_Type	PO_Type must be set to "Q_" to set the SA flow in/out of a model based on a flow rate, or "H_" to base it on a water level.	Char(16)
3	PO_Label	PO_Label is the Label attribute in a 2d_po layer (see Section 9.3.3). The 2d_po Type attribute must also be compatible with the PO_Type (ie. it must include Q_ or H_ as appropriate).	Char(40)

7.4.2.3 RF Option

Read GIS SA RF (rainfall) option calculates and applies the flow to the model based on catchment area, initial loss and continuing loss information. Boundary condition inputs are specified as a rainfall hyetograph (mm versus hours) instead of flow hydrographs, which is required for the other Read GIS SA options.

7.4.2.4 Trigger Option

Read GIS SA TRIGGER option allows the initiation of inflow hydrographs based on a flow or water level trigger. For example, reservoir failures can be initiated based on when the flood wave reaches the reservoir. An example of this option is:

```
Read GIS SA Trigger == mi\2d_satr_dambreaks.mif
```

The 2d_satr layer is a 2d_sa layer (with one attribute, Name), to which three additional attributes are added, as listed in Table 7-6. These are:

- Trigger_Type
- Trigger_Location
- Trigger_Value

Every timestep during the simulation, the flow (Q_) or water level (H_) of the 2d_po object referenced by Trigger_Location is monitored. When the 2d_po exceeds the Trigger_Value value the SA hydrograph commences.

If applying the hydrograph as a dam break, when digitising the SA trigger polygon, it should preferably be on the downstream side of the dam wall, extending the width of the dam wall and possibly be several cells thick (in the direction of flow). If there are stability issues, enlarging the SA polygon will help,

however applications have indicated the feature typically 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 modelled using 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).

7.4.2.5 Flow Feature

The Read GIS SA PO option models seepage or infiltration based on a varying water level or flow rate elsewhere in the model. For example, modelling the seepage of groundwater into a coastal lagoon that is dependent on the water level in the lagoon.

The feature is set up as follows:

1. Create a 2d_po point object of Type "H_" at the water level location.
2. Add "Read GIS PO == …." to the .tcf file if not already there.
3. Create a new 2d_sa layer (call it 2d_sapo) and add two new attributes, as listed in Table 7-6. The PO option attributes are:
   (i) PO_Type: Char of length 16
   (ii) PO_Label: Char of max length 40
4. Digitise the SA polygon(s) covering the area of seepage or infiltration and for the attributes:
   (i) Set the Name attribute to the name of the water level vs flow curve in the BC database.
   (ii) Set PO_Type to "H_".
   (iii) Set PO_Label to the PO Label of the relevant 2d_po "H_" point to be used to determine the flow from the h vs Q curve.
5. Add "Read GIS SA PO == …" to the .tbc file.

Alternatively, to base the SA flow on the flow elsewhere in the model, use a 2d_po line object of Type "Q_". To check the SA in/outflow:

1. View the _MB.csv files. The SA in/outflow from the seepage or infiltration will be part/all of the SS columns.
2. Add "SS" to the "Map Output Data Types ==" command. This outputs over time the net in/outflow from all source flows (ST, SA, SX and rainfalls).

7.4.3 Rainfall

7.4.3.1 Rainfall Overview

Three approaches are available for applying rainfall directly to the 2D cells. These are listed below and described in the following sections.

- 2d_rf layers, the original approach where polygons are assigned a rainfall gauge and spatial and temporal adjustment factors.
- Gridded rainfall where rainfall distribution over time as a series of grids in ESRI .asc/.flt files or in a NetCDF file are specified. This feature was introduced for the 2016-03 release.
- A rainfall control (.trfc) file that allows the user to specify rainfall gauges/points over the model and a series of commands to control how the rainfall is interpolated. This approach generates a series of rainfall grids available to the user for display or checking.

7.4.3.2 2d_rf Layers

The 2d_rf layer applies a rainfall depth to every active cell within the digitised polygon based on an input rainfall hyetograph. Table 7-7 lists the attributes of 2d_rf layers which are referenced in the .tbc file using the commands Read GIS RF or Read RowCol 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 (the converted hydrograph is reported in the .tlf file for cross-checking). The Map Output Data Types RFR and RFC may be used to view the rainfall rate (mm/hr) and cumulative rainfall (mm) over time respectively.

If the same rainfall is to be applied to all active cells in the model, the .tbc command Global Rainfall BC may be used in lieu of digitising a 2d_rf layer.

Note: If using direct rainfall boundaries then the double precision version may be required (see Section 11.4). The .tcf command Model Precision == Double can be used to ensure that only a double precision version of TUFLOW. This is recommend for direct rainfall models.

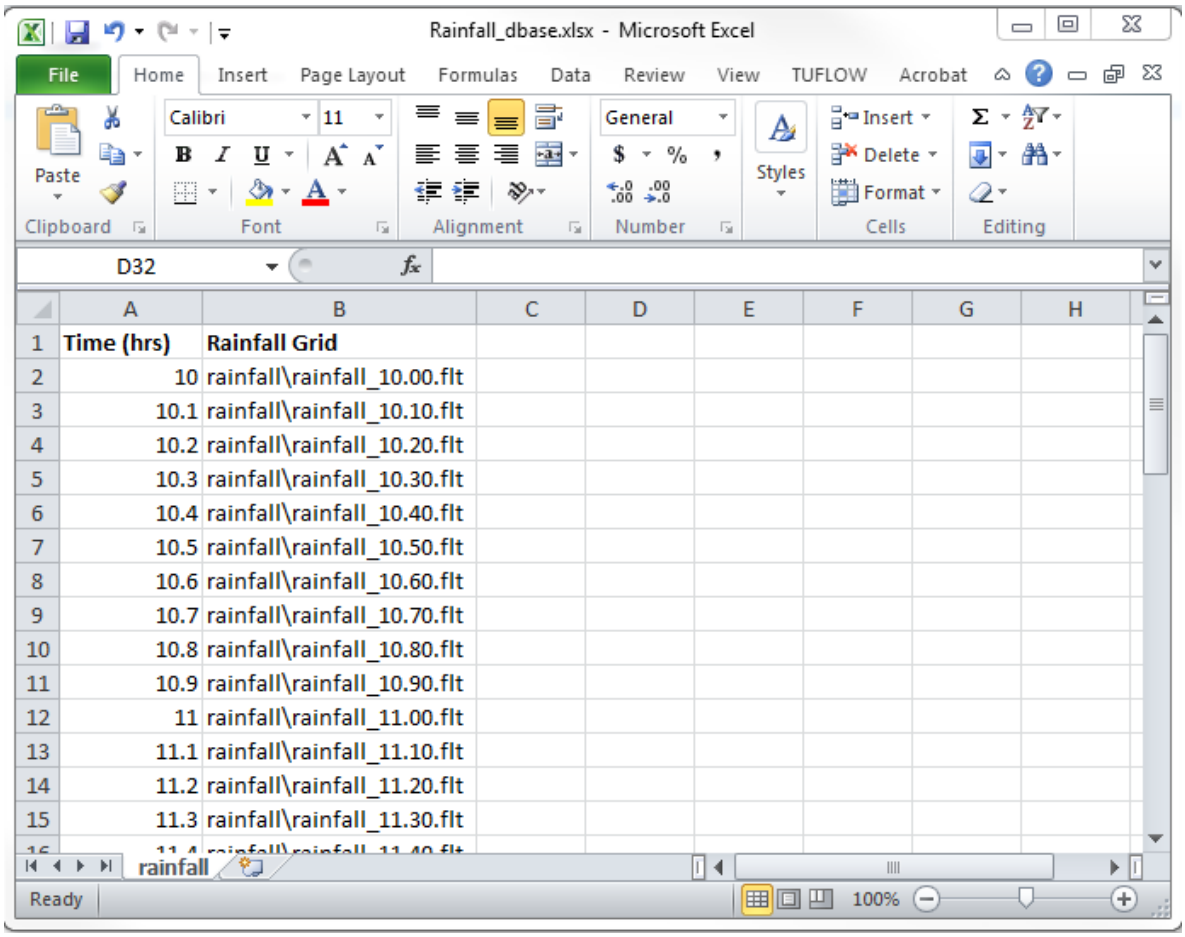
Table 7-7 2D Direct Rainfall[1] over Area (2d_rf) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS RF Command			
1	Name	The name of the rainfall[1] BC in the BC Database (see Section 7.5). **Note: If two or more RF inflows of the same Name cover the same cell, only the first RF inflow is used. It is recommended that each polygon has a unique Name and they do not overlap.**	Char(100)
2	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 zero rainfall to be applied.	Float
3	f2	A second multiplier that allows for adjusting the rainfall spatially. A value of zero will cause zero rainfall to be applied.	Float
[1] Initial and continuing losses can be applied on a material-by-material basis (see Section 6.9.4). Initial and continuing losses are not applied to negative rainfall values (to model infiltration/evaporation).			

7.4.3.3 Gridded Rainfall

Rainfall may also be applied to the model using the .tcf command Read Grid RF. The command references a .csv index containing links to the rainfall grids or a .nc NetCDF file containing all the grids. This allows the applied rainfall to vary spatially across the model without the need to digitise multiple polygons within a 2d_rf GIS layer. If using the .csv file index the rainfall grids may be in ESRI ASCII (.asc) or binary grid (.flt) format, noting that the latter is much faster for TUFLOW to process.

For the .csv format the rainfall database contains two columns, the first being time (in simulation hours) and the second being the rainfall grid. There is no interpolation between times, so a stepped approach is applied. Each rainfall grid applies from its time in the .csv file to the time of the next rainfall grid. Time increments do not have be a constant interval. An example of a rainfall database is shown in Figure 7-2. The values in rainfall grid should be the rainfall depth in mm that have fallen over the period. In the example below the values in the 2nd rainfall grid should be the total rainfall depths from time 10.0 to time 10.1 hours (ie. not the rainfall intensity in mm/h).



	A	B
1	Time (hrs)	Rainfall Grid
2	10	rainfall\rainfall_10.00.flt
3	10.1	rainfall\rainfall_10.10.flt
4	10.2	rainfall\rainfall_10.20.flt
5	10.3	rainfall\rainfall_10.30.flt
6	10.4	rainfall\rainfall_10.40.flt
7	10.5	rainfall\rainfall_10.50.flt
8	10.6	rainfall\rainfall_10.60.flt
9	10.7	rainfall\rainfall_10.70.flt
10	10.8	rainfall\rainfall_10.80.flt
11	10.9	rainfall\rainfall_10.90.flt
12	11	rainfall\rainfall_11.00.flt
13	11.1	rainfall\rainfall_11.10.flt
14	11.2	rainfall\rainfall_11.20.flt
15	11.3	rainfall\rainfall_11.30.flt

Figure 7-2 Example Rainfall Database

Note that the Read Grid RF command is referenced in the .tcf rather than the .tbc file. Unlike the Read GIS RF and Read RowCol RF commands. This is to allow the rainfall to be applied across all 2D domains if multiple domains (see Section 8.3.1) have been used (ie. the Read Grid RF command is domain independent).

7.4.3.4 Rainfall Control File (.trfc file)

As of the 2016-03 release rainfall can be temporally and spatially controlled, including interpolation of rainfall gauges over time, using a series of commands in a rainfall control file (.trfc). The commands generate rainfall grids based on point rainfall gauges, with three methods available for spatial interpolation of the rainfall distribution. The rainfall distribution varies over time according to the rainfall at the gauges. The three methods for spatially interpolating the rainfall are:

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

When the rainfall control file is processed during model initialisation a series of rainfall grids are output, which are then used by the simulation to vary the rainfall over the 2D domain(s). This feature may also be useful simply to generate the series of rainfall grids for other purposes or display. The rainfall grids are pre-processed to reduce memory usage whilst TUFLOW is running.

Of note is that the polygon approach is similar to using a series of rainfall polygons read in via the .tbc Read GIS RF command. However, by pre-processing using a .trfc file, significant more memory efficiency occurs, particularly if a large number of rainfall boundaries is used.

Example .trfc files are being uploaded to this wiki page.

Links to available commands in the rainfall control file are listed below.

IDW Exponent
IDW Maximum Distance
IDW Maximum Points
Maximum Hyetograph Points
Maximum RF Locations
Read GIS RF Points
Read GIS RF Polygons
Read GIS RF Triangles
RF Grid Cell Size
RF Grid Format
RF Grid Origin
RF Grid Size
RF Interpolation Method

7.5 Boundary Condition (BC) Database

A boundary condition (BC) database is created 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 boundary data (eg. flow, level, rainfall 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 7-8. 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 7-8. 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. The Excel TUFLOW Tools.xlm macros can be used to export the worksheets to .csv format. Excel TUFLOW Tools.xlm can be downloaded from www.tuflow.com.

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.

The maximum line length (ie. number of characters including spaces and tabs) in a source file is 100,000 characters.

Table 7-8 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. 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. 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 15.2.4) for converting output from RAFTS, RORB, URBS and WBNM to .ts1 format). The ts1 is a comma separated file, which has additional header data to increase its read efficiency. The .ts1 file format is outlined on the TUFLOW wiki on the TS1 file format page should you wish to use a script to write data using this format. • RAFTS-XP .tot and .loc files. 12, 16 and 20 field output is supported. • WBNM _Meta.out files. • 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. For all other types of Source entry, leave this field blank.	3

Keyword	Description	Default Column[1]
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, flows for a QT boundary or water levels for a HQ boundary. For a Blank Source entry, the constant value to be applied. For example to apply a mean water level to a HT boundary the source can be left blank and the water level entered in this column. For RAFTS-XP (.tot or .loc), WBNM _Meta.out and TUFLOW/ESTRY .ts1 files, the name of the hydrograph location to extract. Note: it is NOT possible to combine the Value and ID keywords in the column label, for example "**Value or ID**". 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. For example this could be used to add a base flow to a QT boundary or sea level rise allowance for a HT boundary. 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.

7.5.1 BC Database Example

Figure 7-3 shows 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.

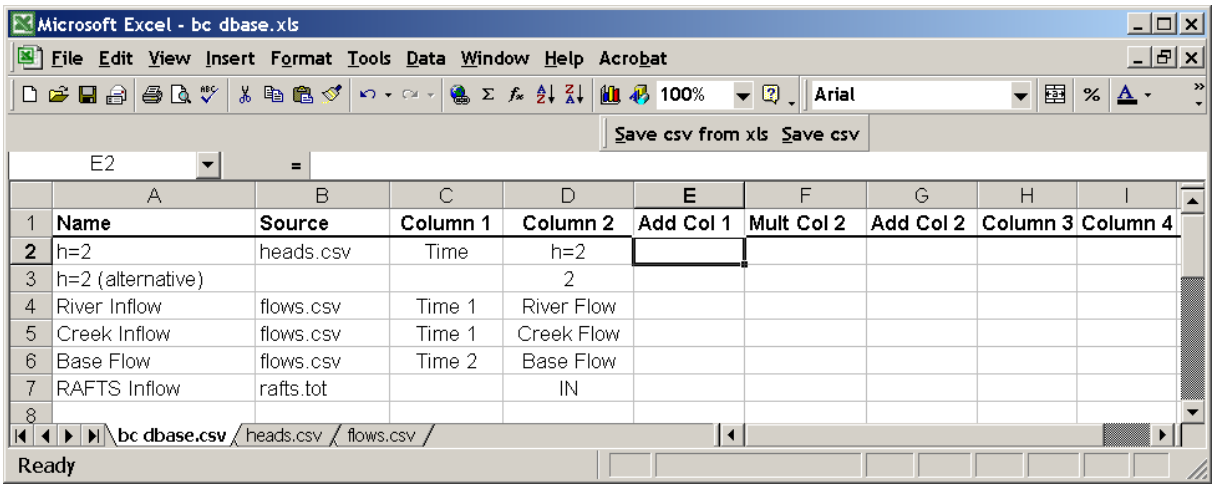
TUFLOW or ESTRY searches through the file until a row is found with the two keywords `Name` and `Source`. `Name` and `Source` do not have to be located in Columns 1 and 2 (although this is recommended).

Table 7-8 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:

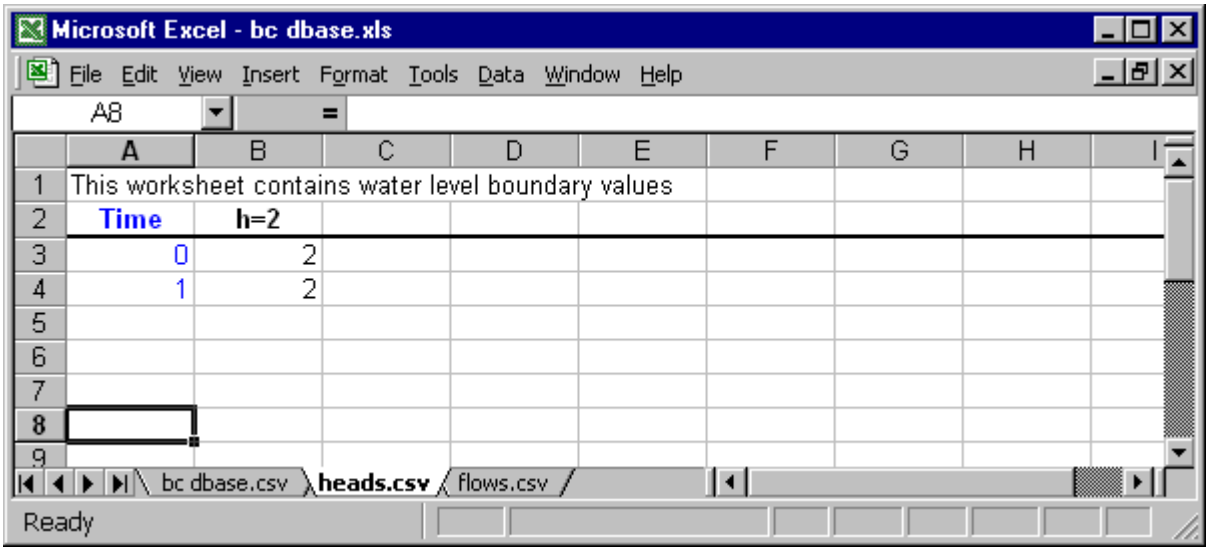
- A BC data event named "h=2" is located in the file heads.csv. The time values are located under a column called "Time" and the BC values are located under a column "h=2".
- As an alternative to "h=2" above, a BC data event "h=2 (alternative)" is set a constant value of 2.
- "River Inflow" is located 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 Figure 7-4).

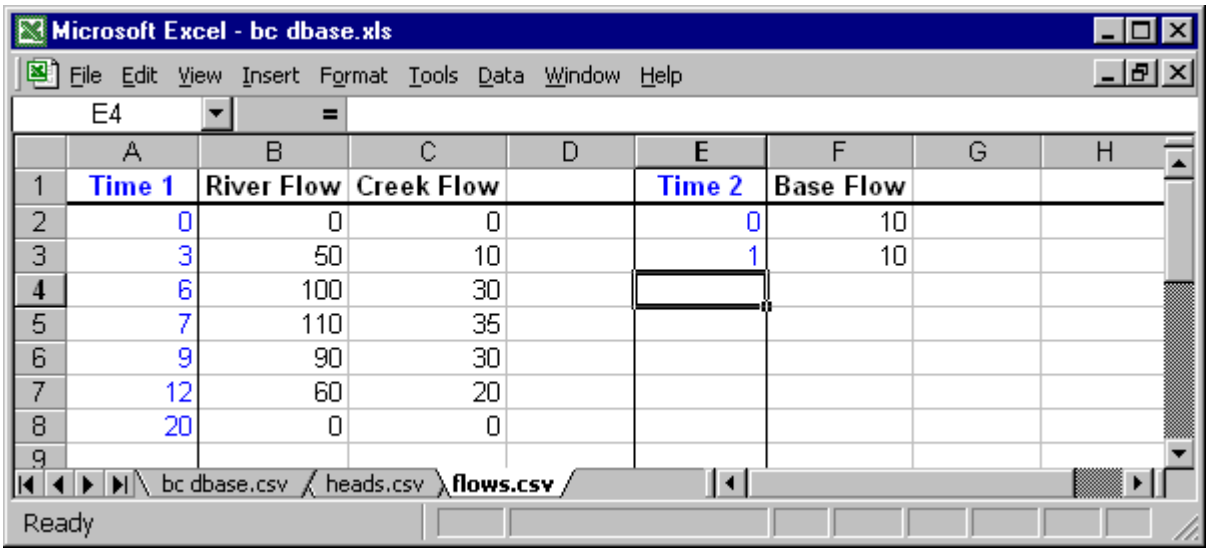


Name	Source	Column 1	Column 2	Add Col 1	Mult Col 2	Add Col 2	Column 3	Column 4
h=2	heads.csv	Time	h=2					
h=2 (alternative)			2					
River Inflow	flows.csv	Time 1	River Flow					
Creek Inflow	flows.csv	Time 1	Creek Flow					
Base Flow	flows.csv	Time 2	Base Flow					
RAFTS Inflow	rafts.tot		IN					

Figure 7-3 Simple BC Database Example



This worksheet contains water level boundary values	
Time	h=2
0	2
1	2



Time 1	River Flow	Creek Flow	Time 2	Base Flow
0	0	0	0	10
3	50	10	1	10
6	100	30		
7	110	35		
9	90	30		
12	60	20		
20	0	0		

Figure 7-4 Example BC Database Source Files

7.5.2 Using the BC Event Name Command

The ability to model multiple events using an Event File (.tef) file offers even more power and flexibility to using BC Event Text and BC Event Name as discussed below. This avoids the need to have a separate .tcf file for each simulation. See Section 11.3 for details on using an event file. The BC Event Name will continue to be supported although users are encouraged to use the event file.

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 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).

The commands allow a wildcard to be set in the BC_database and the text to replace the wildcard. How the commands work is illustrated in the example below.

A BC database file worksheet is created, as illustrated in Figure 7-5, and the following lines are included in all .tcf files.

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

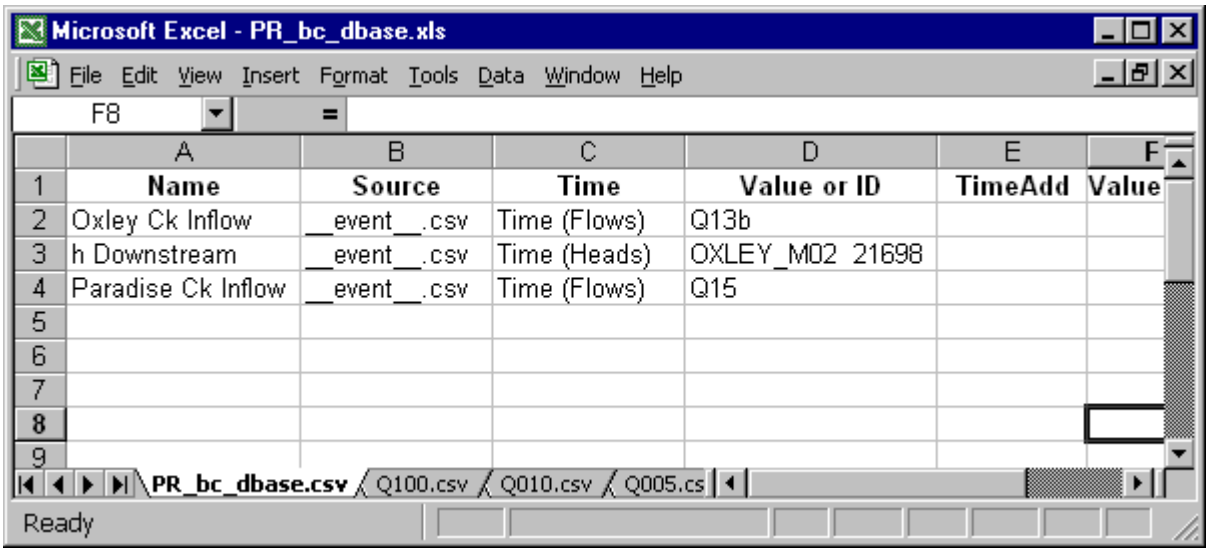
(**Tip:** These lines can be specified in a separate file. The file can be read using the Read File command in all the .tcf files. This saves repeating these lines, and other common commands to all .tcf files.)

The above commands set the active BC Database for TUFLOW and ESTRY to `..\bcdbase\PR_bc_dbase.csv`, and defines the text "`__event__`" as a wildcard to be replaced by the BC Event Name. The following command is added to the .tcf file for the 100 year flood simulation:

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

TUFLOW replaces **every** occurrence of "`__event__`" with "`Q100`" in each line of the BC database. If "`__event__`" does not occur the line remains unchanged. In the example below, the following occurs:

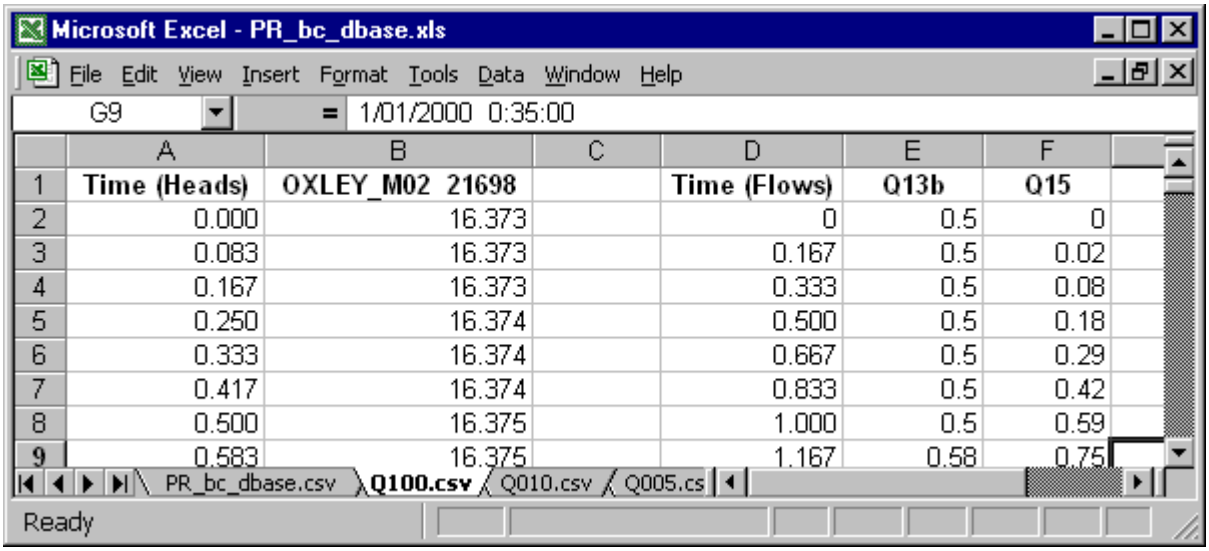
- For the BC event "Oxley Ck Inflow", the BC data is read from file "Q100.csv" rather than "__event__.csv" as indicated in the spreadsheet.
- Similarly, the same applies for "h Downstream" and "Paradise Ck Inflow" BC events.



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

Figure 7-5 Example BC Database Using Event Text

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



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

Figure 7-6 Example BC Database Source Files Using Event Text

To create other simulations, for example a five year flood simulation, it is simply a process of creating the Q005.csv file, saving a copy of the "Q100.tcf" file as "Q005.tcf" and changing BC Event Name to "Q005":

```
BC Event Name == Q005
```

There is no need to create another BC database file. As discussed earlier, the newer Event File option eliminates the need for a separate control file (.tcf) and is the preferred approach, see Section 11.3 for details on using the event file approach.

7.6 Initial Conditions

Two options are available for setting initial conditions in the model. Initial Water Levels (IWLs) can be defined in the 1D and 2D domains to set a global constant or spatially varying water level. Alternatively, the use of restart files sets initial water levels, flow velocities and flow regime based on a previous simulation of the model.

7.6.1 Initial Water Levels (IWL)

7.6.1.1 1D Domains

Similarly for 1D domains, initial water levels (IWL) are set globally as a constant using the .ecf command Set IWL. IWLs can also vary spatially using one or more GIS layers. The default initial water level at 1D nodes is zero (0).

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 GIS Files.
2. Digitise points snapped to nodes and assign each point an initial water level value, alternatively points can be copied from the 1d_nwk_N_check files and assigned an IWL
3. Save the GIS layer
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).

Differences in initial water levels for related features in the 1D and 2D domain can cause model instabilities at the start of a model simulation. If your model becomes unstable in the first few timesteps, review the initial water level values to ensure they are consistent in both domains and also match the water level of any head boundaries that they are be connected to.

Table 7-9 1D Initial Water Level (1d_iwl) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS IWL Command			
1	IWL	Initial water level of object relative to model datum (m).	Float

7.6.1.2 2D Domains

2D IWLs are set globally as a constant using the Set IWL (.tcf file) or Set IWL (.tgc file) commands. IWLs can also vary spatially using one or more GIS and/or grid layers in the .tcf file (Read GIS IWL and Read Grid IWL) or the .tgc file (Read GIS IWL or Read Grid IWL). This is particularly useful for setting initial water levels in lakes, dams, etc.

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 GIS Files.
2. Digitise regions or points and assign each object an initial water level value.
3. Save the GIS layer.
4. Use the .tcf Read GIS IWL command or tgc Read GIS IWL command to read in the IWL values.

The Read Grid IWL command is used to read the IWL from a raster. The grid file containing the initial water levels should be saved in the ESRI ASCII (.asc) or binary grids (.flt) format. This can then be read into the tcf or tgc via the Read Grid IWL command. This can be used to set the IWL based on the water level in a previous simulation, if ASC or FLT is specified in the Map Output Format. The results from one simulation can be directly read by another.

An example which uses both a global set command as well as a read grid command is shown below:

```
Set IWL == 1.1
Read Grid IWL == ..\results\grids\M01_5m_002_h_Max.asc
```

Alternatively, the Read RowCol command can be used, this defines the IWL based on the TUFLOW cell row and column number. As this uses the row and column number of the cell it is therefore dependent on the cell size, model origin, orientation and extent, if any of these changes the file should be re-generated. For this reason the Read Grid IWL is preferred, however, the Read RowCol command can still be used. Instructions for setting this up are 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 7.9):
   (i) Keep the first two columns as the row (n) and column (m) grid references;
   (ii) Remove all other columns;
   (iii) Add a (third) column which is nominally named “IWL” defined as a float or decimal.
4. Using GIS software 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. Save the GIS layer
7. Use the Read RowCol IWL command to read in the IWL values. The Read RowCol command should refer to the .MID file for MapInfo and the .dbf file for shapefiles. The reason for this is

that the spatial data is not required. The location of the model inputs is defined by the model specific row (n) and column (m) grid references.

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 may be 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).

Table 7-10 2D Initial Water Level (2d_iwl) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS IWL Command			
1	IWL	Initial water level of object relative to model datum (m).	Float
Read RowCol IWL Command			
1	N	Row of cell.	Integer
2	M	Column of cell.	Integer
3	IWL	Initial water level at cell relative to model datum (m).	Float

7.6.1.3 Automatic Initial Water Level (Set IWL == AUTO)

As of the 2016-03 release a good option is to automatically set the model's initial water level to the downstream boundary water level using Set IWL == AUTO. This new command is very useful when performing a Monte Carlo assessment that required over 11,000 simulations with each simulation having a unique initial water level!

7.6.2 Restart Files

Gradually varying water surface initial water level conditions are best defined using a restart file. This is achieved by running the model for a warm-up period prior to the main event to create a restart file which establishes the initial water levels and flow velocities. This is achieved using the following commands: Write Restart File at Time, Write Restart File Interval and Read Restart File. The command Write Restart Filename controls whether restart files are overwritten or are time-stamped.

Note: The restart file feature is only available for 2D only, 2D/1D and 2D2D models. It is currently not available for 1D only models. The 2D and 1D restart files use a .trf and .erf extension respectively.

The restart file contains information on the water levels, velocities and flow regimes for the 1D and 2D parts of the model. Therefore, the number of 2D cells and 1D channels must be the same between the original model and the model using the restart file.

Permitted changes between runs could include different boundary conditions, cell elevations and roughness values.

8 Linked 1D and 2D Models

Chapter Contents

8 Linked 1D and 2D Models 8-1

8.1 Introduction 8-2

8.2 Linking 1D and 2D Domains (1D/2D) 8-3

8.2.1 Linking Mechanisms 8-5

8.2.1.1 HX 2D Head Boundary 8-5

8.2.1.2 SX 2D Flow Boundary 8-8

8.2.2 TUFLOW 1D (ESTRY) Domains 8-10

8.2.3 External 1D Solutions (Flood Modeller, XP-Solutions, 12D Solutions) 8-12

8.3 Linking TUFLOW 1D to External 1D Domains 8-15

8.3.1 Flood Modeller 1D/1D Link 8-15

8.4 Linking 2D Domains (2D / 2D) 8-16

8.1 Introduction

This chapter of the manual discusses the methods available for dynamically linking 1D domains to 2D domains, and the linking of a multiple 2D domains to one another. Linked models offer numerous benefits including greater flexibility, greater computational efficiency and performance.

1D-2D linked models are able to utilise the individual benefits of 1D and 2D solution schemes. The 1D scheme is typically used to represent rivers or pipe networks where the flow is essentially one-directional. A 2D scheme is suited to the representation of floodplains where a more detailed flow patterns may occur. TUFLOW may be dynamically linked to 1D networks using the hydrodynamic solutions of ESTRY (TUFLOW 1D), Flood Modeller (previously ISIS), XP-SWMM and 12D Solutions' Dynamic Drainage. Linking of Flood Modeller directly to ESTRY is also possible using the Flood Modeller: TUFLOW - PIPE link.

2D2D linked models allow for a single model to contain multiple 2D domains of different orientation and cell size. The domains may be linked together, nested, or linked via a 1D domain. A smaller cell size (finer resolution) may be used in areas where a more detailed assessment is required with a larger cell size used to represent the remainder of the modelled extent. For example, when modelling a rural catchment containing a number of townships, multiple 2D domain modelling permits high resolution modelling in the urban areas whilst minimising model simulation times by modelling the surrounding rural areas at a coarser resolution.

8.2 Linking 1D and 2D Domains (1D/2D)

TUFLOW 1D and 2D domains can be linked in a variety of ways as illustrated in Figure 8-1 (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 8-1. This approach was developed by Syme (1991) and has been widely applied and ever since.

TUFLOW also permits the following configurations:

- 1D elements through 2D embankments (Sketch 1b in Figure 8-1).
- Nesting or "carving" of a 1D channel through a 2D domain (Sketch 1c in Figure 8-1 and Figure 8-2).
- 1D pipe networks underneath the 2D domain (Sketch 1c in Figure 8-1 and Figure 8-3).

This section of the manual also discusses the linking of TUFLOW 1D (ESTRY) and Flood Modeller to the TUFLOW 2D domain.

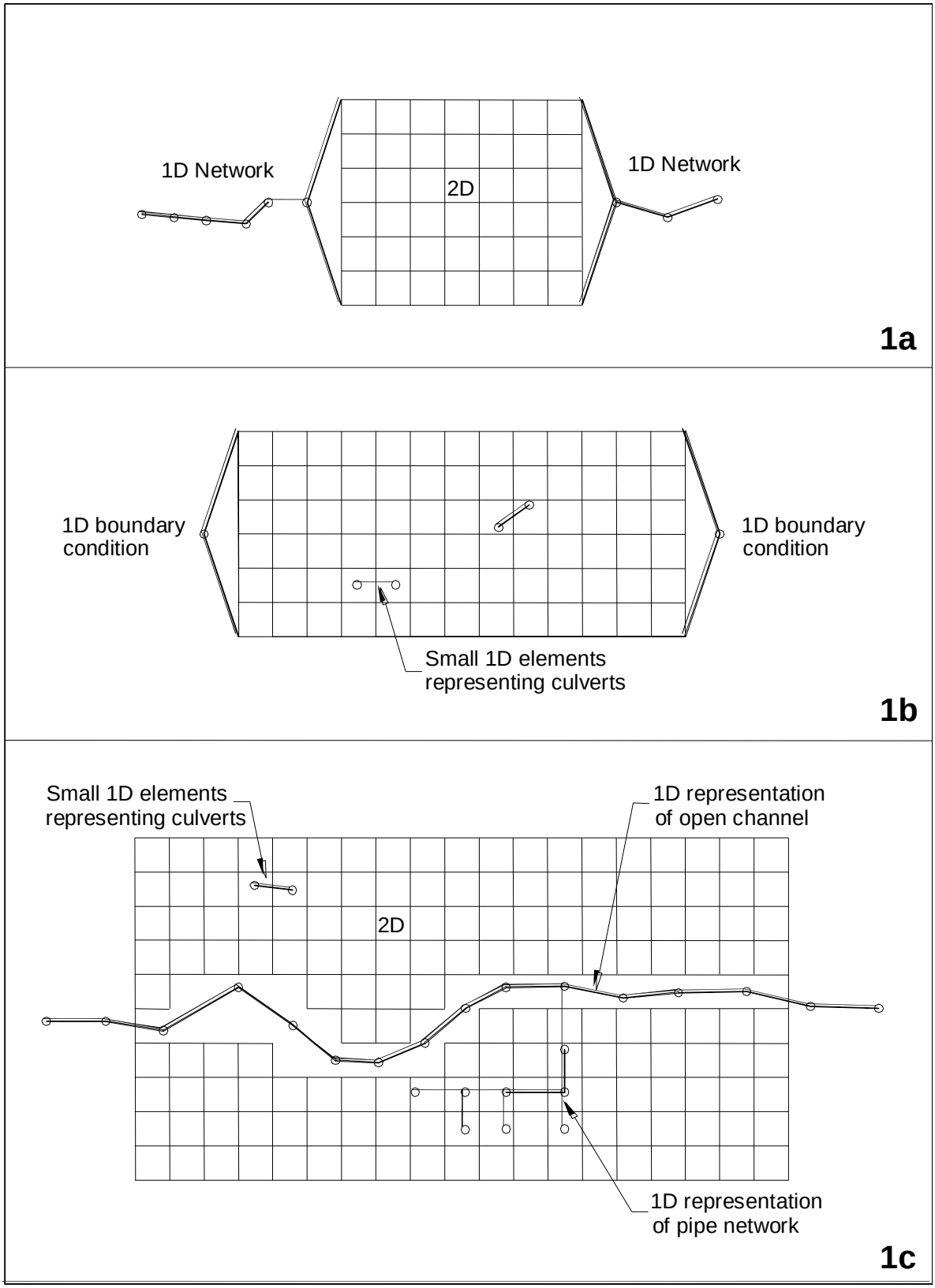


Figure 8-1 1D/2D Linking Mechanisms

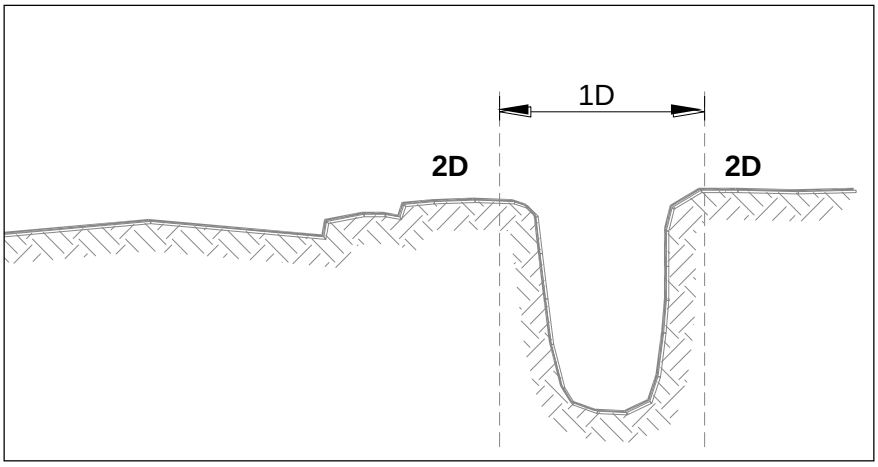


Figure 8-2 Modelling a Channel in 1D and the Floodplain in 2D

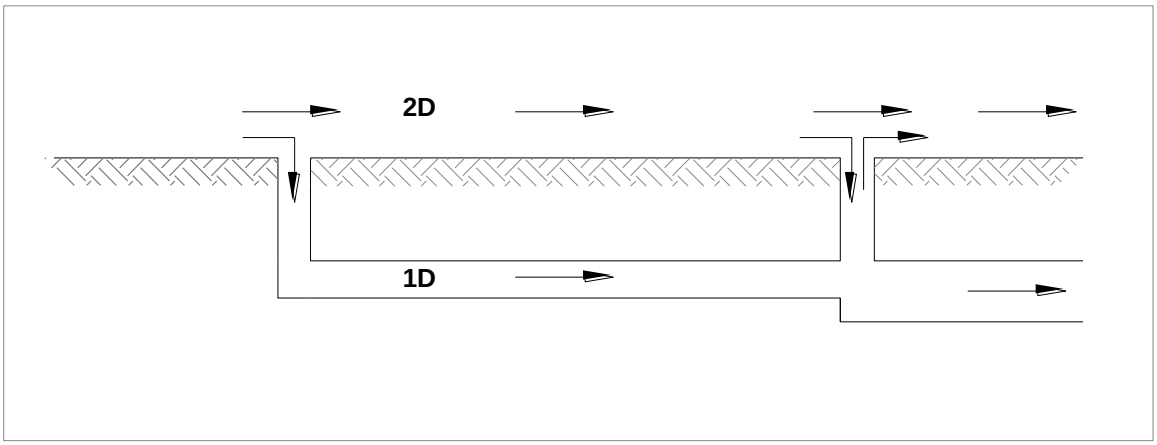


Figure 8-3 Modelling a Pipe System in 1D underneath a 2D Domain

8.2.1 Linking Mechanisms

There are two types of 1D/2D linking option available. These methods are:

1. Head Boundaries to the 2D cells (HX); and
2. Source boundaries to the 2D cells (SX).

These are described in the sections below.

8.2.1.1 HX 2D Head Boundary

The HX terminology is derived from a **H**ead boundary being applied to the 2D cell using data from an e**X**ternal 1D scheme. This type of connection can be used for the 1D schemes of ESTRY, Flood Modeller and XPSWMM.

Depending on the water level in the surrounding 2D cells, flow can either enter or leave the HX cells. The volume of water entering or leaving the 2D boundary is added or subtracted from the 1D model to preserve volume.

The HX lines are connected to the 1D nodes via connection objects, these are CN type objects in the 2d_bc format layer. In Figure 8-4 the HX lines are shown as along the top of the channel as purple lines, the 1D nodes are shown in yellow and the CN lines in green. The CN lines are snapped to vertices along the HX line, at these vertices the water level is connected to the 1D node. Along the HX lines, between the 1D node connections the water levels are linearly interpolated, for this reason it is a minimum requirement that a HX line have a CN line connected to each of its end. If the same node is connected to the HX line at each end, as per diagram 1A in Figure 8-1, the water level in the 2D cells will be the same along the entire HX line and the line should be digitised perpendicular to flow.

The elevations of 2D HX cells determine when water can spill from the 1D into the 2D. Water level is calculated at the cell centres in the 2D engine (see Section 6.2), once the water level in the 1D node exceeds the elevation in the boundary cell water can enter or leave the 2D cell. As such, correct definition of the 2D HX cell elevation is important. If a levee aligns with the HX line, a breakline is recommended along the levee to ensure that the 2D cell elevations are consistent with the levee or spill crest.

In the four panels of Figure 8-5, the 2D HX connection is shown in section view at four stages in a flood:

- In the left panel, the water level in the 1D node is below the 2D cell elevation and no flow is occurring across the connection;
- In the 2nd panel, the water level in the 1D is above the 2D cell elevation along the left bank (but not the right bank). In this case water is spilling from the 1D channel into the 2D floodplain on the left hand side;
- In the 3rd panel, the water level in the 1D is below the water levels in the 2D floodplains and water will be moving from the 2D into the 1D channel, which is typical of a flood recession; and
- In the 4th panel, the flow is still moving from the 2D into the 1D on the left bank, however, on the right bank, the water level in the floodplain is below the 2D HX cell elevation and no flow is occurring (the water is perched).

The HX link preserves momentum in the sense that the velocity field is assumed to be undisturbed across the link. The velocity direction is however not influenced by the direction of the linked 1D channel. Use of HX links at a structure and momentum preservation is also discussed in Section 6.12.

The minimum ZC (cell centre) elevation at or along a HX object should be above the 1D bed level (interpolated between connected 1D nodes). This is necessary to ensure that there is water in the 1D 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, this can cause mass balance issues. See also the tcf command HX ZC Check.

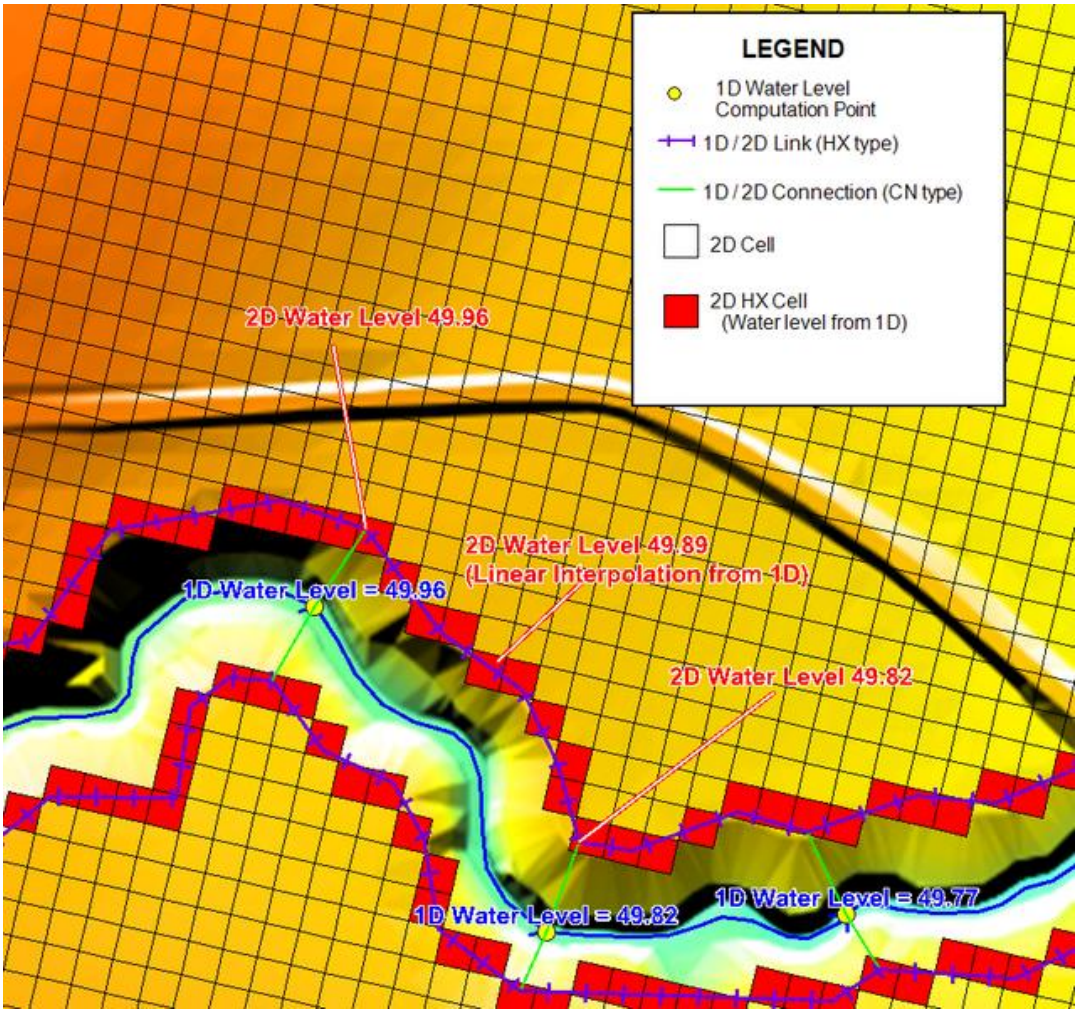


Figure 8-4 HX Schematic – Plan View

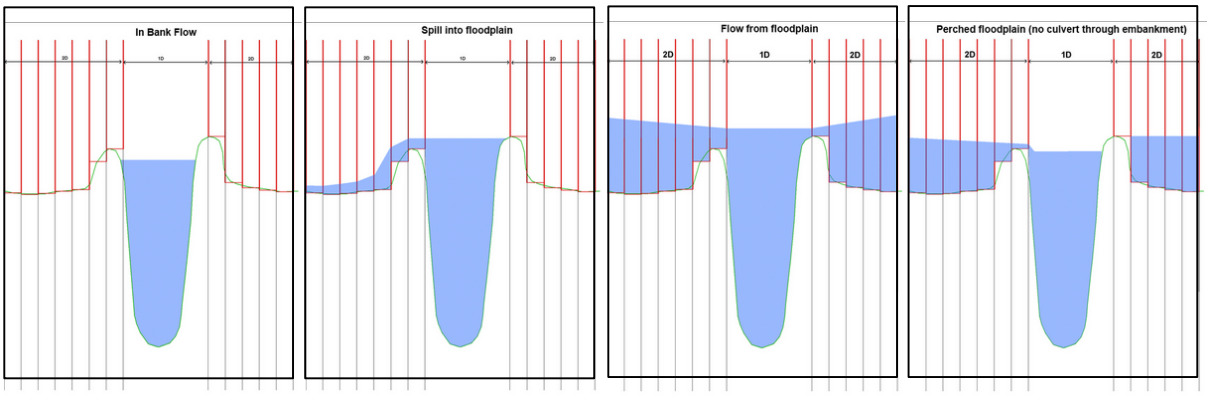


Figure 8-5 HX Schematic – Section View

8.2.1.2 SX 2D Flow Boundary

The SX terminology is derived from a **S**ource boundary being applied to the 2D cell using data from an e**X**ternal 1D scheme. This type of connection can be used for the 1D schemes of ESTRY, Flood Modeller, XPSWMM and 12D Solutions.

Water level in the 1D node is determined based on the average water level along the 2D SX cells. Conversely, water level in the 2D SX cell is determined based on the flow from the 1D node. Flow is proportioned via depth if multiple SX cells are connected to a single node. The SX Flow Distribution Depth Cutoff command can be used to control the depth of water below which a cell does not receive flows from the connected 1D element.

SX objects are digitised in the 2d_bc layer and have the type "SX", these can either be point (a single SX cell) or line objects (multiple SX cells). When using the SX line objects, these are joined to the 1D nodes with a CN type object in the 2d_bc format. This is shown in Figure 8-6. The SX lines (2d_bc) are shown in yellow and the CN lines (2d_bc) are shown in red. These are connected to the 1d network (1d_nwk) at the nodes / end of the culvert. The line objects in this case select three cells upstream and downstream of the structure, flows from the culvert are applied / subtracted from these cells and the average water level along these cells is applied to the 1D channel.

When connecting a SX connection for a structure it is important to ensure that the connected SX cell width matches the size of the structure. For example if a 10m wide 1D structure is connected to a single 2D cell with a cell size of 1m, between timesteps the 1D can convey more flow than the 2D and this can lead to oscillations.

TUFLOW checks whether the minimum ZC elevation at or along a SX connection is below the connected 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, leading to mass balance / stability issues. If this occurs Error 2050 is reported by TUFLOW. See also the tcf command SX ZC Check.

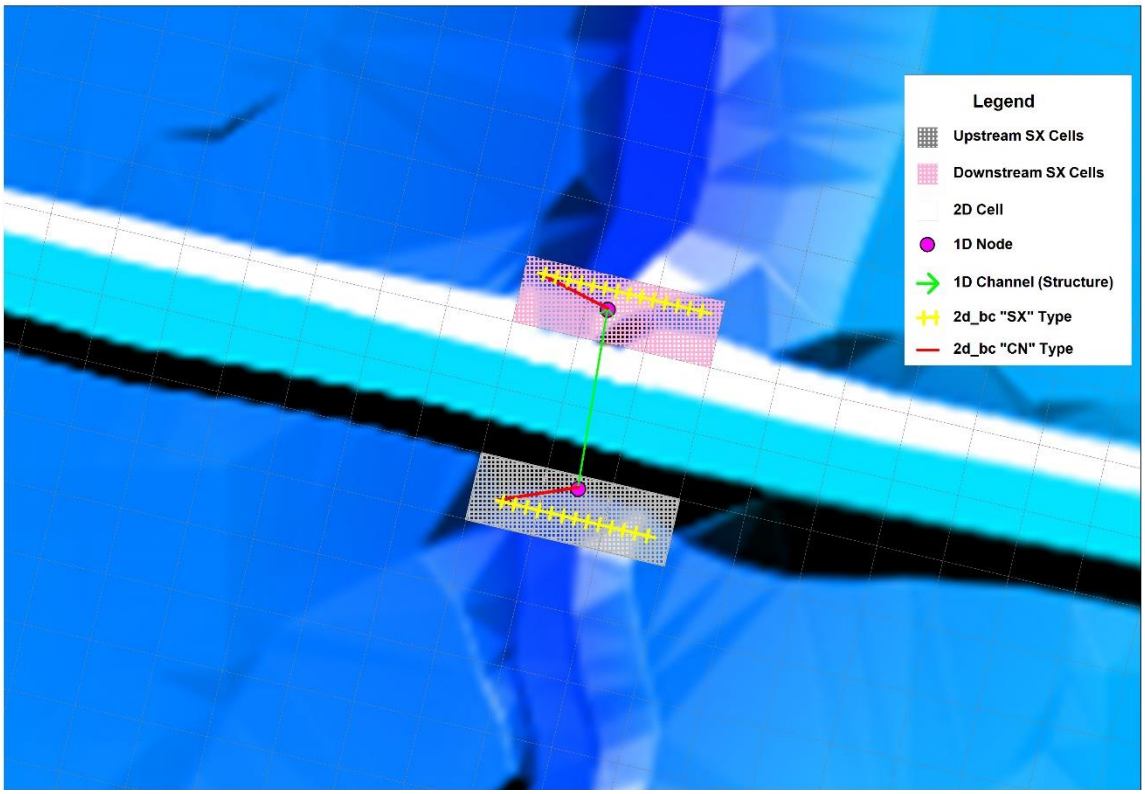


Figure 8-6 SX Schematic – Cross Drainage Structure

A more complex underground SX connection is presented in Figure 8-7. Underground pipe networks can also be modelled within TUFLOW as 1D elements which are dynamically linked with 2D overland flow. SX connections are used to transfer flow between the 1D pit/pipe network and the 2D domain. This is typically modelled using the 1d_nwk "Conn_1D_2D" attribute. This is discussed in detail within Section 5.12.

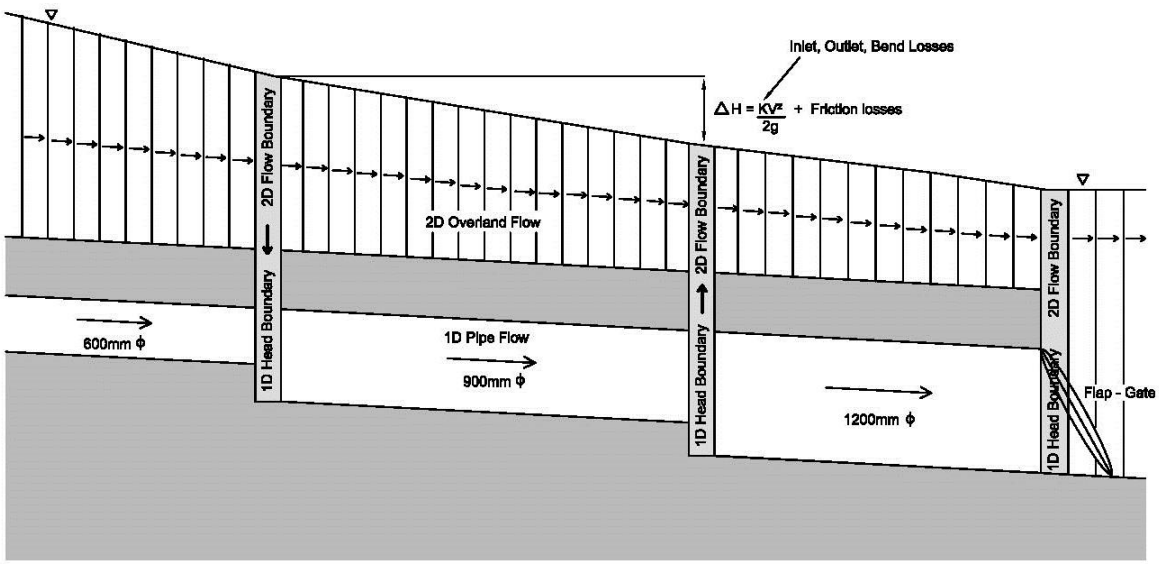


Figure 8-7 SX Schematic – Pipe Network

8.2.2 TUFLOW 1D (ESTRY) Domains

Linked 1D and 2D domain models use 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 7-4 and Table 7-5 in Section 7.4.1. The recommended approach for 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 8-8). 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 6-9).
- 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.

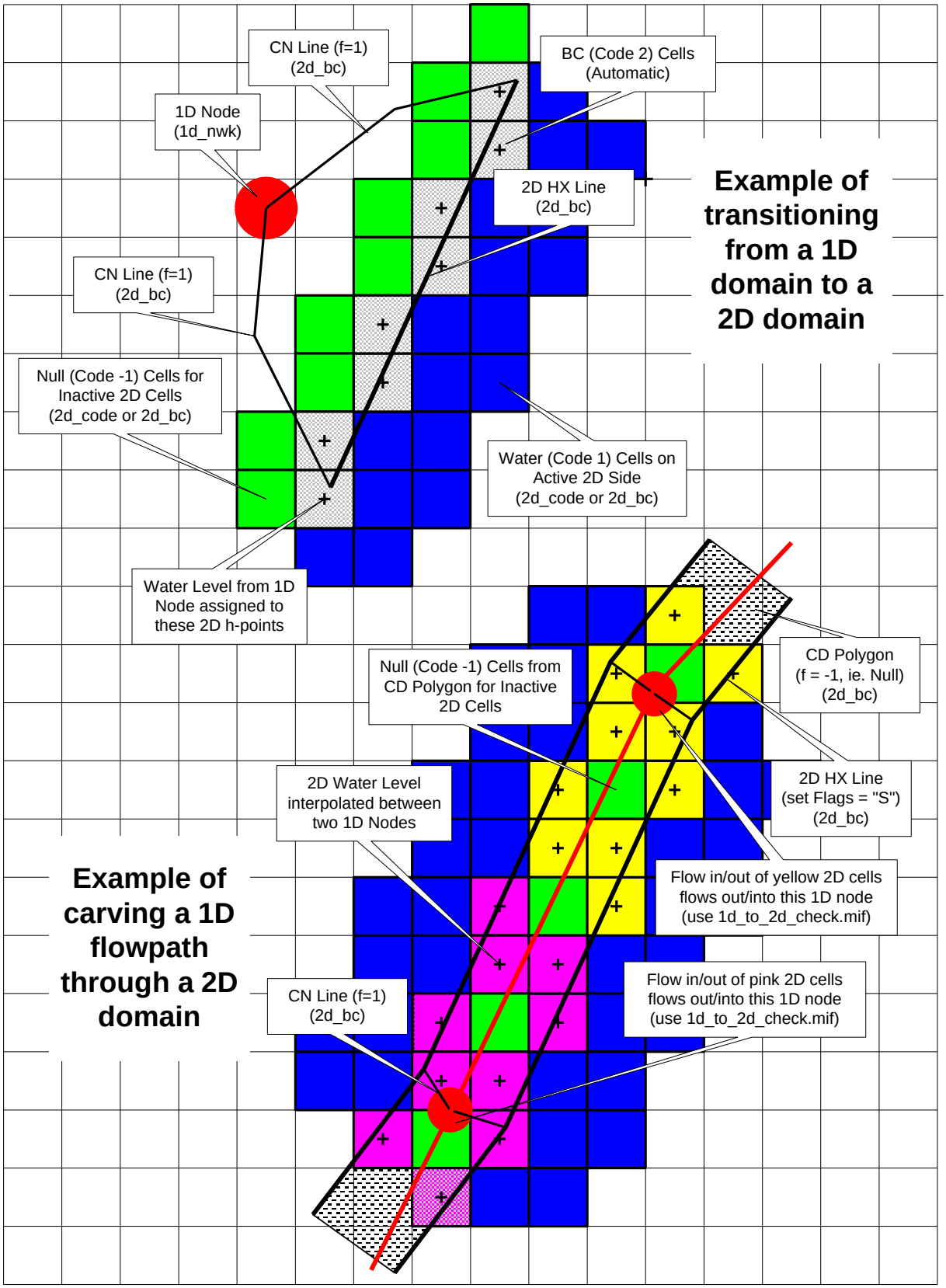


Figure 8-8 Examples of 2D HX Links to 1D Nodes

8.2.3 External 1D Solutions (Flood Modeller, XP-Solutions, 12D Solutions)

TUFLOW 2D domains can be dynamically linked to Flood Modeller (previously known as ISIS), XP-SWMM and 12D Solutions DDA 1D solution schemes. Flood Modeller can also link to TUFLOW's 1D solution (previously known as ISIS-TUFLOW-PIPE or ISIS-ESTRY). To link external 1D schemes a 1d_x1d GIS layer showing the locations of the external scheme's 1D nodes is required. For Flood Modeller and 12D DDA the 1d_x1d layer can either be created by the user or by the Flood Modeller or 12D GUI. For XP-SWMM, this layer is created automatically by the XP-2D or XP-STORM GUI. In all cases if using a GUI, the layer created can be found within the generated input files for the TUFLOW engine and will be referenced within the generated .tcf file.

The external 1D software must be installed and configured to support the linking to TUFLOW. For Flood Modeller the TUFLOW Link must be purchased from CH2MHill unless using the Free versions of both TUFLOW and Flood Modeller. For XP-SWMM and 12D Solutions, the entire process is managed and the files automatically generated by their respective GUIs.

The 1d_x1d layer requires only one attribute, namely a string 12 characters long that contains the unique IDs of the external scheme's 1D nodes. Any other attributes are presently ignored. Creation of this layer manually may be possible through exporting 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 TUFLOW model.

For Flood Modeller the IDs are case sensitive (because Flood Modeller IDs are case sensitive), therefore, the IDs in Flood Modeller and the IDs in the 1d_x1d layer(s) must be identical.

In the .tcf file, use the Read GIS X1D Network command to read the 1d_x1d layer as shown in the example below:

```
Read GIS X1D Network == ..\model\mi\1d_x1d_isis_nodes.mif
```

The linking approach is identical or similar to the ESTRY 1D domain linking. XP and 12D have identical linking as these schemes use a staggered 1D network with flows calculated between the nodes in the same manner as ESTRY. For Flood Modeller a slightly different linking mechanism is used due to the Flood Modeller water level and flows both being calculated at the nodes.

The digitising of channels (polylines) within a 1d_x1d GIS layer is optional, and is only necessary if the model uses 1D Water Level Lines (1d_wll – refer to Section 9.5 for further details). The GIS layer(s) are read into the .tcf file using the command Read GIS X1D Network. Note that "X1D" can be substituted with "ISIS", "XP" or "12D", ie. Read GIS ISIS Network, Read GIS XP Network and Read GIS 12D Network are all aliases for Read GIS X1D Network.

The example below shows the use of all three available commands for external 1D schemes in use with the nodes and channels split into separate layers in this instance.

```
Read GIS X1D Nodes == ..\Model\mi\1d_x1d_ISIS_nodes.MIF
Read GIS X1D Network == ..\Model\mi\1d_x1d_ISIS_channels.MIF
Read GIS X1D WLL == ..\Model\mi\1d_x1d_ISIS_wlls.MIF
```

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

There are no restrictions on connecting XP and 12D external 1D schemes, however, due to the slightly different linking arrangement there are restrictions when connecting to Flood Modeller nodes as follows:

- HX lines should only be connected to Flood Modeller RIVER units. Flood Modeller INTERPOLATE and REPLICATE units are also permitted. This requires that the Flood Modeller unit between consecutive connections along a HX line is always a RIVER unit. If this is not the case, an "ERROR 2043 - 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.
- Special Flood Modeller units may be required for some HX and SX connections (refer to the Flood Modeller documentation).

It is not necessary for all Flood Modeller RIVER units between the upstream and downstream ends of the HX line to be connected. Nodes may be intentionally or accidently omitted as with ESTRY nodes. Please note that this is not recommended because by missing or omitting nodes the 1D water level water surface gradient that is applied along the HX lines will be missing the omitted nodes and will therefore not be correctly represented.

TUFLOW checks whether 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, and if not, an error occurs (see Section 8.2.1.1).

Use the _x1d_nodes_check file to crosscheck the external 1D nodes that were read and associated information passed from the 1D scheme. Also use the 1d_to_2d_check file to crosscheck which HX and SX cells are connected to which nodes. The HX cells in the 1d_to_2d_check layer have been colour coded to represent the external 1D channel/node that the flow in/out across the HX cells is associated with. For Flood Modeller, by default HX cells are assigned to the upstream river unit as a lateral flow.

Note that the start and end simulation times and the timestep are controlled by the external 1D scheme's GUI input fields, therefore, any Start Time, End Time and Timestep commands are ignored in the .tcf file. The external 1D schemes can differ from the TUFLOW timestep noting that for Flood Modeller it is highly recommended that the Flood Modeller timestep is an integer divisor of the TUFLOW timestep.

No .ecf file is required and the ESTRY Control File command should not be specified unless there are also ESTRY 1D domain(s) in addition to the external scheme 1D domain(s). Flood Modeller nodes can also be directly linked to ESTRY 1D domains - refer to Section 8.3.1.

Note that Flood Modeller HX links may benefit from assigning a FLC (typically 0.1 to 0.5 in value) to HX lines using the 2d_bc "a" attribute (see Table 7-5). For HX lines running along the river banks, especially those with high overtopping velocities, improved stability and representation of the energy losses associated with the water peeling off from the river to the floodplain or vice versa can be realised.

The .tcf command Write X1D Check Files writes out check_x1D_H_to_2D.csv and check_2D_Q_to_x1D.csv files that contain the water levels and flows sent to the 2D to/from the external 1D scheme.

8.3 Linking TUFLOW 1D to External 1D Domains

8.3.1 Flood Modeller 1D/1D Link

TUFLOW models can also be configured with Flood Modeller for dynamically linked 1D pipe network 2D overland flow modelling. The main driver for this feature is for Flood Modeller - TUFLOW models to utilise the powerful pipe network and manhole modelling capabilities of TUFLOW (see Section 5.12) and be able to link these networks into a Flood Modeller river model.

Flood Modeller and TUFLOW (ESTRY) nodes will be considered linked if:

1. An ESTRY node in a 1d_nwk layer, and a Flood Modeller node in a Read GIS ISIS Nodes or Read GIS ISIS Network layer are snapped.
2. The ESTRY node has a 1d_nwk Conn_1D_2D attribute of either "X1DH" or "X1DQ".
   (i) If Conn_1D_2D is blank then “X1DH” is assumed.
   (ii) A connector "X" channel type can be used to connect the end of the linked ESTRY channel to the ESTRY node snapped to the Flood Modeller node if the end of the ESTRY channel and the snapped Flood Modeller /ESTRY nodes are not in the same location.
   (iii) Note that the upstream and downstream inverts for the ESTRY node linked to Flood Modeller should be set to -99999 unless the node is also being used to set the inverts of channels snapped to it.
3. An "X1DH" link means a Flood Modeller 1D water level is being applied at the ESTRY node (ie. Flood Modeller sends ESTRY a water level and ESTRY sends back a +/- flow to Flood Modeller).
4. An "X1DQ" link means a Flood Modeller inflow/outflow is being applied at the ESTRY node (ie. Flood Modeller sends ESTRY a +/- flow and ESTRY sends back a water level).
5. An ESTRY X1DH (the default) would be used for most Flood Modeller ESTRY links. An X1DQ might be more appropriate where a Flood Modeller model stops and flows into an ESTRY model.

Generally, an ESTRY timestep will be smaller than the Flood Modeller timestep. In these cases, the total volume is accumulated over all ESTRY timesteps within a Flood Modeller timestep, and applied to the Flood Modeller model as a discharge by dividing the volume by the Flood Modeller timestep.

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

- X1DH V In: The volume of water in via a X1DH link.
- X1DH V Out: The volume of water out via a X1DH link.
- X1DQ V In: The volume of water in via a X1DQ link.
- X1DQ V Out: The volume of water out via a X1DQ link.

The type or existence of a connection can be checked by viewing the Conn_1D_2D attribute in the 1d_nwk_N_check layer. The _messages.mif/.shp layer contains CHECK 1393 messages at each ESTRY node linked to a Flood Modeller node.

8.4 Linking 2D Domains (2D / 2D)

Any number of 2D domains of different cell size and orientation can be combined to form one model. The 2D domains can be linked via 1D domains or directly to each other. 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. Alternatively, direct 2D to 2D linking can be achieved by using the 2d_bc 2D link type (see Table 7-4 and Table 7-5). Examples of these model configurations are shown in Figure 8-9 and Figure 8-10.

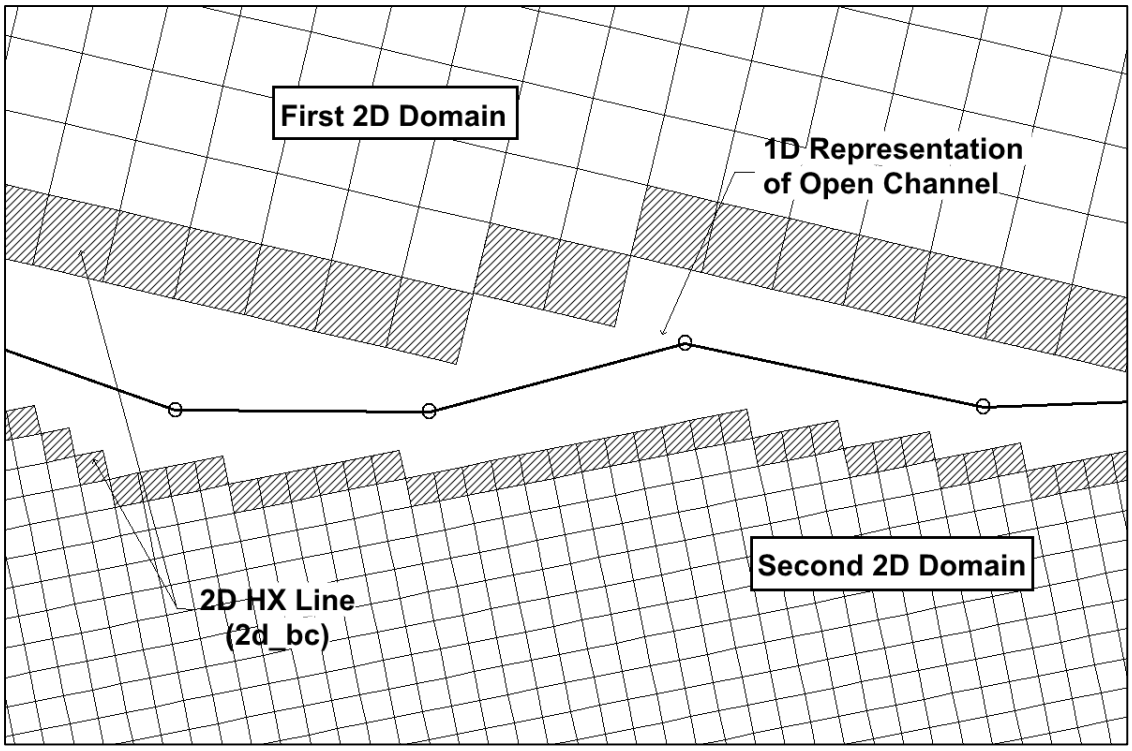


Figure 8-9 Schematic of a Multiple Domain Model linked via a 1D Domain

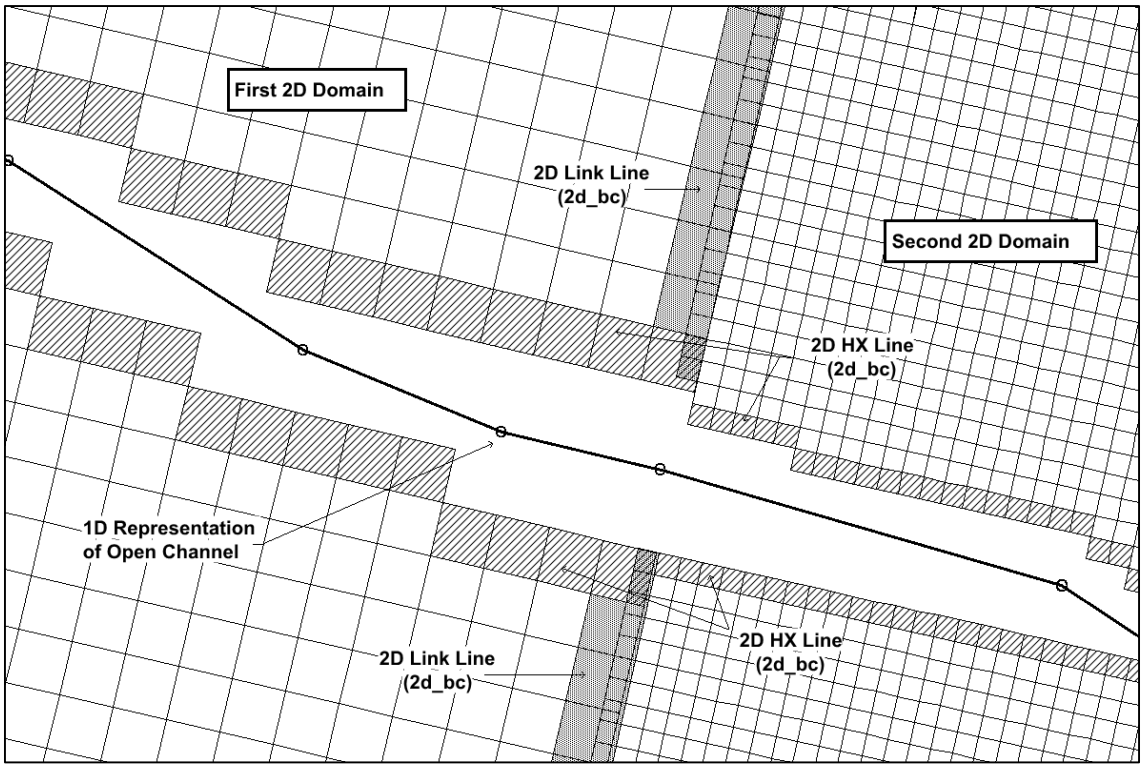


Figure 8-10 Schematic of a Multiple 2D Domain Model using the 2d_bc "2D" Link

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 ESTRY Control File and BC Database commands are independent of the 2D domain block. As such they are typically not included within the 2D domain block.

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

Geometry Control File
BC Control File
Timestep

Optional commands that can be used are:

Cell Wet/Dry Depth
Instability Water Level
Read GIS FC
Read GIS GLO
Read GIS IWL
Read RowCol IWL
Read GIS LP
Read GIS PO

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

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

BD Database == ..\bc_dbase\bcdbase_Q100_2hr_001.csv
ESTRY Control File == ..\model\1D_domain.ecf
Read Materials File == ..\model\n_values.tmf
Start Time (h) == 0.
End Time (h) == 12.
```

Multiple 2D domain models use the 2d_bc "2D" link type (see Table 7-4 and Table 7-5) as the boundary cells which transfer flow between the neighbouring domains. The link type creates hidden 1D nodes at each vertex along the 2D link line and also at a regular interval, as defined by the "d" attribute (see Table 7-5). The hidden 1D nodes act as storage that convey the water from one 2D domain to the other. The water levels along the 2d_bc 2D link line are linearly interpolated using the water levels in the hidden 1D nodes. If the water level profile in reality is not close to linear between vertices, strange flow patterns may occur which can lead to model instability. As such, appropriate resolution of the hidden 1D nodes is an important feature of multiple 2D domain models. The .tcf command Reveal 1D Nodes can be used to view the hidden 1D nodes. The hidden nodes will be written to the nwk_N check layer, as shown in Figure 8-11. For multiple domain models using the 2d_bc "2D" link, note that this GIS layer must be read into the .tbc files of **both** 2D domains. The 2D link can then be checked by viewing the _2d_to_2d_check layer which displays the 2D cells used to link the two domains together. These features, and their check files are shown in Figure 8-11.

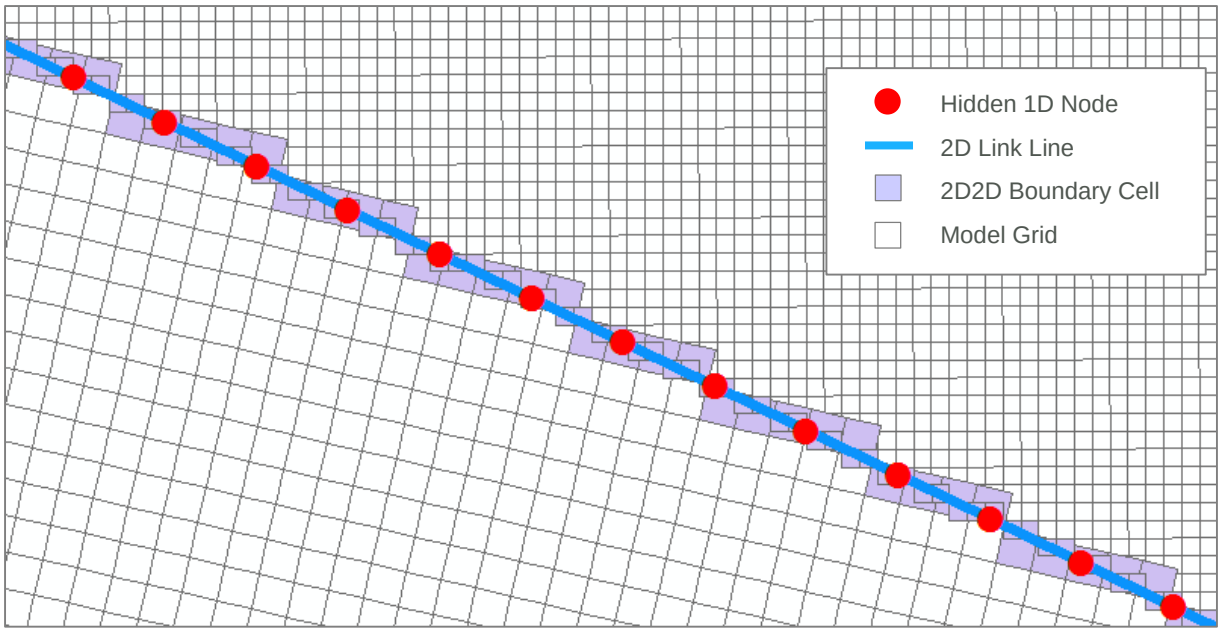


Figure 8-11 Multiple 2D Domain Model "2D" Link Check Files

The following guidance is recommended when defining the location and orientation of the 2d_bc "2D" link.

- 2D link lines will be most stable if digitised perpendicular to the dominant flow direction in high conveyance locations (eg. Rivers and creeks);
- Orientation of the 2D link lines is less critical in low conveyance regions, such as floodplain storage areas (ie. perpendicular orientation is not a necessity). High resolution definition of the hidden 1D nodes may be required. If used, the automated hidden 1D nodes distance attribute "d" should not be set finer than three times the larger cell size.

If the extents of the 2D domains overlap, it will be necessary to deactivate the overlapping cells in the other domain. Using the example above, the active area for the 'West Domain' will need to be deactivated in the .tgc file of the 'East Domain' and vice versa. The Read GIS Code Invert command is useful for this process as it allows for the same GIS layer to be used to active/deactivate cells.

A unique model Timestep is recommended for each 2D domain. The timestep should be defined so that is conforms to the Courant stability criteria which is discussed in Section 3.4.1. The timesteps of all 2D and 1D domains should be an integer multiple of one another.

Significant enhancements to the Multiple 2D domain linking was been introduced for TUFLOW Build 2013-12-AC. **It is recommended that models utilising 2D/2D linking upgrade to Build 2013-12-AC or later where possible.**

The default method is METHOD D as defined by the .tcf command Link 2D2D Approach. Method D incorporates the enhancements and options available within the following .tcf commands:

Link 2D2D Distribute Flow
Boundary Viscosity Factor
Link 2D2D Global Stability Factor
Link 2D2D Adjust Velocity Head Factor

Multiple 2D domain modelling requires the Multiple 2D Domain Module license (visit www.tuflow.com or contact sales@tuflow.com if you do not have this module).

9 Customising Output

Chapter Contents

9 Customising Output 9-1

9.1 Introduction 9-3

9.2 Output Control Commands 9-4

9.3 Configuring Plot Output Options 9-9

9.3.1 Reporting Locations (1D and 2D Combined) 9-9

9.3.2 Grouped Structure (1D and 2D) Output 9-11

9.3.3 2D Time-Series (Plot) Output (2d_po, 2d_lp) 9-12

9.4 Customising Map Output 9-18

9.4.1 Overview 9-18

9.4.2 Changing the Output for Different File Formats 9-18

9.4.3 Output Zones – Enhancing Map Output 9-19

9.4.4 Gauge Level Map Output (2d_glo) 9-22

9.5 Including 1D Results in Map Output 9-24

9.5.1 Overview 9-24

9.5.2 Water Level Lines (WLL, 1d_wll) 9-24

9.5.3 Water Level Line Points (WLLp) 9-26

9.5.4 Manually Adding Triangles into the 1d_WLL Layer 9-28

9.6 Map Output Formats 9-29

9.6.1 Overview 9-29

9.6.2 Which GIS/GUI Supports Which Formats? 9-29

9.6.3 Mesh Based Map Output Formats 9-30

9.6.3.1 DAT and XMDF 9-30

9.6.3.2 TMO 9-31

9.6.3.3 WRB 9-31

9.6.3.4 T3 9-31

9.6.4 Grid Based Map Output Formats 9-32

9.6.4.1 ASC, FLT 9-32

9.6.4.2 NC – NetCDF 9-33

9.6.4.3 TGO 9-33

9.6.4.4 WRR 9-34

9.6.5 Mesh and Grid Combined Map Output 9-34

9.6.5.1 WRC 9-34

9.6.6 GIS Based Map Output Formats 9-34

9.6.7 Mesh Configurations (.2dm File) 9-35

9.6.7.1 Quadrilateral and Triangle Mesh Option 9-35

9.6.7.2 Triangular Mesh Option 9-35
9.6.7.3 2D Cell Corner Interpolation/Extrapolation 9-35
9.7 Map Output Data Types 9-41
9.8 Specialised Outputs 9-56
9.8.1 Recording Gauge Data at Receptors (2d_obj, 2d_rec) 9-56
9.8.2 Evacuation Routes (2d_zshr) 9-58
9.8.3 Calibration Points GIS Layer 9-60

9.1 Introduction

TUFLOW outputs cross-check information and hydraulic calculation results in a range of different file formats for use within a wide choice of text, charting, plotting, GIS and GUI visualisation tools and software.

This chapter discusses options to configure and customise TUFLOW's output prior to carrying out the simulation. There are also a variety of post-processing utilities and viewing options that can be performed on the output after the simulation is completed (see Chapters 12, 13, 14 and 15).

- 12: Check and Log Files
- 13: Viewing and Post-Processing
- 14: Quality Control
- 15: Utilities

For details on the various map formats and content of the hydraulic results refer to Chapter 13.

This chapter includes discussion on:

- List of commands available for customising all the different output, including output to the simulation console window, check files and folder locations (Section 9.2).
- Configuring map output options including 1D results in a 2D form and the powerful output zones feature (Section 9.4.2).
- Options for setting up time-series output from 1D and 2D domains at specified locations (Section 9.3).
- Specialised outputs such as time and depth of inundation along evacuation routes (Section 9.5.1).

Note that all of the customisation options discussed in this chapter require the specification of commands within the control files prior to simulation of the model. Post-processing methods to manipulate TUFLOW results are discussed in Chapter 15.

9.2 Output Control Commands

A wide range and variety of commands allow the user to configure TUFLOW output to be different to the default settings. Table 9-1 lists these commands in different categories followed by a brief description.

Table 9-1 Commands used to Control TUFLOW Output

Command (.tcf file unless specified)	Description
Simulation Console Window and .tlf (log) File Content	
Display Water Level	Display a water level on the console window whilst running.
Force File IO Display	Show file opening information to the console window.
Screen/Log Display Interval	Controls how often information is displayed and written to the .tlf file whilst a simulation is underway.
Verbose	Controls how much information is displayed and written to the .tlf file.
File and Folder Management	
Log Folder	Customise the location for the .tlf and other files.
Simulations Log Folder	Customise the location for logging simulations.
TSF Update Interval	Change when and how often the .tsf file is to be updated.
Output Drive	Override the drive letter for all output files.
Output Folder	Customise the parent folder for the 2D based output files.
Output Folder (1D or .ecf)	Customise the parent folder for the 1D based output files.
Output Files	Control which output files are to be written.
Read File	Insert a file into a control file.
Set Variable	Define user variables for use within the control files. Useful for setting output folder names based on a scenario.
Time-Series and Other Tabular Output	
CSV Header Line CSV Maximum Number Columns	These are legacy commands for use with changing how time-series .csv files are written.
CSV Time	Change the units of the time column.
Excel Start Date	Start the time column at a specific date for use in Excel.
Order Output (1D or .ecf)	Controls whether 1D nodes and channels are output in alphanumeric order or in the order they are processed during input.

Command (.tcf file unless specified)	Description
Output Interval (1D or .ecf) Output Times Same as 2D (1D or .ecf)	Use these commands to have a different output interval for 1D time-series output, but not recommended and not compatible with the new 2016-03 time-series output format.
Read GIS LP	Set up 2D longitudinal profile output.
Read GIS PO	Set up 2D time-series output locations.
Read GIS Reporting Location	Set up combined 1D and 2D time-series output locations.
Start Time Series Output	Change the start of time-series output to be different to the simulation start time.
Time Series Output Interval	Mandatory command that controls the frequency of time-series output.
Write PO Online	Write time-series output at each time map output is written (rather than just at the end of the simulation).
Map Output	
BSS Cutoff Depth	Change the default cutoff depth setting for bed shear stress output.
End Map Output	Change the end time for map output to be different to the simulation end time. Useful for output zones where a shorter output period is required.
Map Cutoff Depth	Change the default cutoff depth for setting whether a 2D cell is wet or dry in the map output. Useful for direct rainfall modelling. Only used for map output, ie. it does not affect the hydraulic computations like Cell Wet/Dry Depth does.
Map Output Corner Interpolation	Apply different approaches to interpolating results to 2D cell corners. Primarily provided for backward compatibility for legacy models.
Map Output Data Types	Controls the map data types to be output. Default is just water level and velocity, but there are many more!
Map Output Entire Model	Switch on or off the map output for the entire model.
Map Output Format	Specify one or more map output formats to be written.
Map Output Interval	Mandatory command that controls the frequency of map output, or set to zero to only output maximums.
Maximum Velocity Cutoff Depth	Change the default depth at which the approach to tracking maximum velocities is changed.
Read GIS GLO	Change from regular time based map output to map output produced at different water levels at a gauge. GLO stands for Gauge Level Output.
Read GIS WLL (1D or .ecf) Read GIS X1D WLL	Include the results from 1D elements in the map output using WLLs.

Command (.tcf file unless specified)	Description
Read GIS WLL Points (1D or .ecf) Read GIS X1D WLL Points	Change the ground elevations and materials at WLL points along WLLs.
Start Map Output	Change the start of map output to be different to the simulation start time.
Time Output Cutoff	Add time of exceedance and time of duration for one or more depth or hazard values to the map output data set.
WLLp Interpolate Bed (1D or .ecf)	Change how the bed level along a WLL is assigned.
Tracking of Maximums and Minimums	
Maximums and Minimums	Change the default setting on whether to track maximums and/or minimums.
Maximums and Minimums Only for Grids	Controls whether grid based map output is only written out as maximums, ie. switch on or off the grid temporal map output.
Maximums and Minimums Time Series	Controls whether to track maximums for time-series outputs.
Setting Up Output Zones	
Define Output Zone … End Define	Set up a definition block of commands for an output zone.
Model Output Zones	Controls which output zones are output.
Read GIS Output Zone	Defines the region for the output zone.
GIS and Grid Output, and Check Files and GIS Check Layers	
Calibration Points MI File	Add the maximum water levels to a MIF GIS layer of calibration or gauge points.
GIS Format	Controls the format for output GIS layers.
Grid Format	Controls the format for output GIS grid layers.
Grid Output Cell Size	Sets the cell size of output grids, including check grids. This can be different to the 2D cell size(s).
Grid Output Origin	Change how the grid output origin is determined.
MI Projection	Define the GIS projection for the output of MIF GIS layers.
Pit Channel Offset (1D or .ecf)	Change the default distance that a pit and its results are displayed relative the 1D node the pit flows into.
SHP Projection	Define the GIS projection for the output of SHP GIS layers.
Write Check Files	Customise which check files/layers to output.
Write X1D Check Files	Set to ON to output additional check files for linked external 1D schemes.

Command (.tcf file unless specified)	Description
Write GIS Domain (.tgc)	Output a GIS layer of the 2D domains processed thus far in the .tgc file.
Write GIS Grid (.tgc)	Output a GIS layer of the current active 2D cells and their attributes at that point in the .tgc file.
Write GIS Zpts (.tgc)	Output a GIS layer of the current 2D Zpts elevations at that point in the .tgc file.
Flood Hazard Categories	
UK Hazard Debris Factor UK Hazard Formula UK Hazard Land Use	Change the default settings on how the UK flood hazard output is calculated.
ZP Hazard Cutoff Depth	Change the cutoff depth for outputting People Hazard categories (ZPA, ZPC and ZPI).
Mass Balance Output	
Mass Balance Output	Switch mass balance tracking and outputting on or off.
Mass Balance Output Interval	Set the time interval for outputting mass balance information.
Advanced Outputs	
Set Route Cut Off Type	Controls the parameter to use when determining whether a route is closed. For example, depth or hazard.
Set Route Cut Off Values	Define one or more values for route closure categories.
Read GIS Gauge Output (.tgc)	Add gauge levels and other information to GIS receptor layer(s).
Restart Files	
Write Restart File at Time	Controls how and when restart files are output.
Write Restart File Interval	Sets the interval at which to write a restart file.
Write Restart File Version	Controls which version of the restart file feature to output.
Write Restart Filename	Controls the approach on whether to keep overwriting or give a unique name for each restart file.
XF Files	
XF Files	Global setting on whether or not to use and output XF files.
XF Files Include in Filename	Set a unique text to be included in XF filenames.
Miscellaneous	
Meshparts	Adds flags to the .2dm file for use by some map output viewers such as SMS to control the display of different parts of a model.
Reveal 1D Nodes	Add the hidden 1D nodes used in 2D/2D links to the 1D output.

Command (.tcf file unless specified)	Description
Write Empty GIS Files	Output empty GIS layers. Very useful command when setting up a new model.
Backward Compatibility to Prior Output Formats	
Defaults Output Approach	To set output formats and defaults to those of prior releases for backward compatibility.

9.3 Configuring Plot Output Options

9.3.1 Reporting Locations (1D and 2D Combined)

Read GIS Reporting Location is a new feature in the 2016-03 release of TUFLOW, allowing plotting of time-series results across both the 1D and 2D sections of the model. For example it is possible to digitise a reporting location flow line across 1D and 2D domains, including multiple 2D domains, and TUFLOW will sum the flow across any 1D channels intersected by the line and all the 2D cells.

Reporting location lines are digitised into a 0d_RL GIS layer containing only a single attribute, the name of the reporting location, as outlined in Table 9-2. Both points and lines / polylines can be digitised in the reporting locations. The points and lines can be in the same layer or different layers. Points will be treated as water level output and lines as flow output. For a point snapped to a 1D node, the 1D water level is used, if no 1D node is snapped a 2D water level is output.

The flow line can cross 1D and 2D sections of the model, for the 1D channels it does not have to snap to any vertices on the channels, it just has to intersect them.

Table 9-2 0d_RL Reporting Location Attributes

No.	Default GIS Attribute Name	Description	Type
Read GIS Reporting Location **Command**			
1	Name	The name of the reporting location. Lines and points can share the same name.	Char (32)

The RL outputs are contained in the plot\csv\ folder. The following files are produced:

- _RLL_Q.csv - flow time-series;
- _RLL_Qmx.csv - maximum flow information;
- _RLP_H.csv – water level time series; and
- _RLP_Hmx.csv – maximum water level information.

As well as the maximum water level and flow information, the time that these occur, the water level at maximum flow and vice versa, and the maximum change between timesteps are also output to the mx.csv files.

The RLs are also output to the plot\gis PLOT GIS layers and can be viewed and their time-series data displayed using the new TuPlot QGIS Plugin as illustrated in Figure 9-1.

It is intended to add other plot output data types to this feature in future releases.

See Section 13.2 for further information on plotting of Reporting Location results.

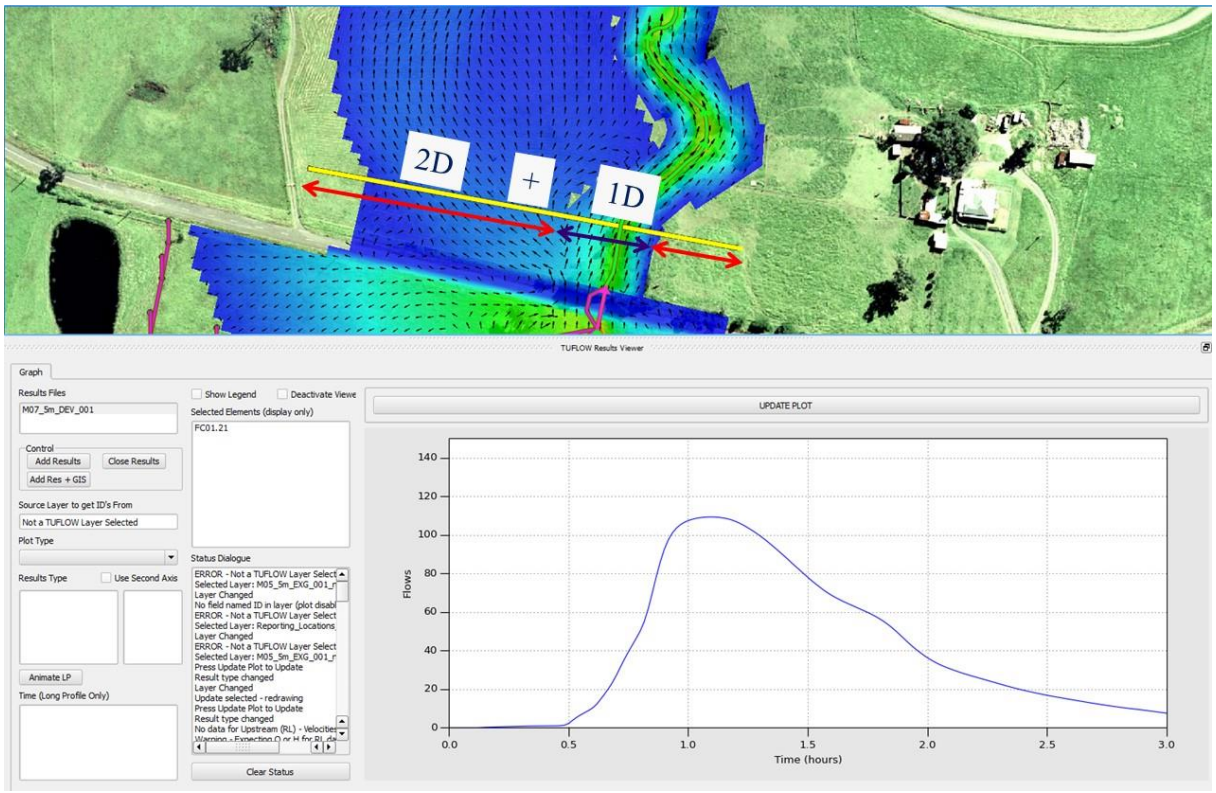


Figure 9-1 Example of the QGIS TuPlot Plugin for a Reporting Location

9.3.2 Grouped Structure (1D and 2D) Output

The new Structure Reporting feature in the 2016-03 release outputs time-series and summary data for single and combined structures. 1D and 2D and 1D/2D structures are all output together to give a complete set of results. The summary output at the flood peak provides the flow split between below and above deck, including for 2D Layered Flow Constrictions, along with other information such as the head drop. The structure output is particularly useful in the reporting of hydraulic structure flows and afflux.

Structures are classified according to the following logic:

1D Structures:

- 1D structures that are in parallel (ie. they have the same upstream and downstream nodes) are automatically grouped together and treated as a single structure for this output. The ID assigned to the group is the structure with the lowest bed elevation. Note that the directions of the digitised channels is important, that is, to form a group all channels must be digitised in the same direction.
- For 1D structures that have no parallel 1D structure channel, these are also included in the output so as to provide a complete set of results for all 1D structures.
- The flow split between below and above deck is based on all structures except weirs contributing to the below deck flow, and weirs to the above deck flow.

2D and 1D/2D Structures:

- To create a structure output that includes 2D flow and optionally any 1D structures, the new "QS" PO line is digitised in a 2d_po layer (see Section 9.3.3). All 2D flow across this line (including multiple 2D domains) and any 1D structures that intersect this line are grouped together. The 1D structure's 1d_nwk line does not have to snap with the QS line; they only have to cross over each other. The ID assigned to the structure output group is the 2d_po QS Label (see Table 9-4).
- If 2d_po QS line selects cell sides that are modified by the Layered Flow Constriction feature, the summary output will split the flow into a below and above "deck" component based on Layers 1 to 3 being below "deck" and Layer 4 above.
- 2d_po HU and HD lines or points can be used to define the upstream and downstream water levels of the structure. HU objects should be located upstream of the structure and HD downstream. The average 2D water level along these objects will be used to populate the upstream and downstream water level data in the output. Lines can have more than two vertices (ie. polylines are accepted). To associate the HU and HD objects with the QS line, all three (QS, HU and HD) must have the same ID for the 2d_po Label attribute Label (see Table 9-4). If a QS line has no HU object associated with it, output that cannot be produced, such as the water level drop across the structure, is given a -99999 value in the _SHmx.csv output file described below.

The structure output is located in the new plot/csv folder and includes:

- _SQ.csv file that contains time-series data of the flow through the structure. This file is similar to other time-series .csv output, but it only contains 1D and/or 2D structures as described above.
- _SHmx.csv file that contains a summary of each structure when the upstream water level reached its maximum. To generate this output the flow and other information is tracked every timestep for grouped structures. The output columns include: flow, area and average velocity for below and above deck; total flow, area and average velocity for the whole structure; upstream and downstream water levels; the head drop across the structure (ie. upstream minus downstream water level); and the time this data was recorded (ie. the time the upstream water level peaked).
- Time-series output for the upstream and downstream water levels are available through the 1d_H.csv, 2d_HD.csv and 2d_HU.csv files. Note that for the 2d_HD.csv and 2d_HU.csv files if all 2D cells are dry a -99999 is output.

See Section 13.2 for further information on plotting of grouped structure results.

9.3.3 2D Time-Series (Plot) Output (2d_po, 2d_lp)

Time-series data output from 2D domains is available for a range of hydraulic parameters (as listed in Table 9-3). Output takes the form of time-series hydrographs (referred to as PO – Plot Output) or longitudinal profiles over time (LP). The data types available for PO are as listed in Table 9-3 and only H_ (water level) and V_ (velocity) outputs are available for LP.

Note that to combine flows for 1D and 2D models the Read GIS Reporting Location command can be used, see Section 9.3.1 for discussion on Reporting Locations.

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 9-2 illustrates how 2d_po objects are interpreted.

The start time for PO and LP output and the output interval is set separately to Map 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 written to a .csv file and also to the _TS layer (refer to Section 13.2.3). 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 9-4 and Table 9-5 describes the GIS attributes of the 2d_po and 2d_lp layers. Of note for **flow** flags of PO output, TUFLOW sums time-series with the same label (this does not apply if the label is left blank).

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.

Tracking of maximums and minimums at every timestep for PO and LP outputs is the default approach. This can be disabled by setting the Maximums and Minimums Time Series command to OFF in the .tcf file.

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

The _TS GIS layer also contains the tracked values. Note, the max/min values in the _TS GIS layer are not tracked at every timestep if this feature is turned off, they are instead tracked at the Time Series Output Interval.

Table 9-3 Time-Series (PO) Data Types

Flag	Description	Point	Line (or Polyline)
G_	Gauge Level	Water level of the h-point of the nearest cell. If the cell is dry, the ground level (ZC) is output. Used for Read GIS Objects to record gauge levels when a receptor is first inundated. Refer to Section 9.5.1.	N/A
H_	Water Level (Head)	Water level 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. Note: If a polyline is used, the average water level along each line segment is output, therefore, use of polylines is not recommended for this output type at present.
HD HU	Downstream and Upstream Structure Water Levels	**Point, Line or Polyline:** Used to define the upstream (HU) and downstream (HD) water levels of a QS structure (see QS below and Section 9.3.2). The 2D water level is used to populate the upstream and downstream water level data in the new structure output feature. For a point the water level at the nearest 2D cell centre is used. For a line or polyline, the average water level along the entire line is calculated. Note, unlike H_ above, polylines are not output segment by segment, with the water level being the average along the entire line. To associate the HU and HD objects with the QS line, all three (QS, HU and HD) must have the same ID for the 2d_po Label attribute Label (see Table 9-4). Note that the water levels over time are output to a 2D PO .csv file and the summary information at the flood peak to the new _SHmx.csv file. If the point or line is completely dry, -99999 is output to the .csv files.	

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 9-2). The flow is positive if the water is flowing away from you when looking in a direction with the start of the PO line on your left and the end of the line on your right. If digitising a flow line across a 1D channel that is carved through the 2D domain, ensure that the line is digitised so that it crosses the 1D channel where there is a change in colour of the linked 2D HX cells as shown in the 1d_to_2d_check or _TSMB1d2d layers.
QA	Flow Area	N/A (zero area results).	The flow area is calculated using the same cell sides as for Q_. An adjustment for obliques lines is made.
QI	Integral Flow	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.
QS	Structure Flow	N/A (zero flow results).	Same as Q_ above, but also used to set up a 2D structure output (see Section 9.3.2) that will include in addition to the 2D flow any flows from intersected 1D structures and the split between below and above deck flows. Note the flow output to the 2D PO .csv files is only the 2D flow, while that to the new _SQ.csv file is the combined 1D/2D structure flow.
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).

Flag	Description	Point	Line (or Polyline)
V_	Velocity	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 or Uu	u-point velocity	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	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	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_.
VY	Velocity in Y-direction	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_.

Table 9-4 2D Plot Output (2d_po) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS PO Command			
1	Type	Any combination of the **two** letter flags listed in Column 1 of Table 9-3 (limit of 10 flags per entry). For example, to output water level and flow time-series for the same line, enter "H_Q_" for the type attribute of the line.	Char (20)
2	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. Read GIS Reporting Location is the preferred method for cumulating flows in 2D1D models. See Section 9.3.1 for discussion on Reporting Locations.	Char (30)
3	Comment	Optional field for entering comments. Not used.	Char (250)

Table 9-5 2D Longitudinal Profiles (2d_lp) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS LP Command			
1	Type	Specify "H_" to output water level. This is the only available option.	Char (20)
2	Label	Label up to 30 characters defining the name of the longitudinal profile. The label appears at the top of the columns of data in the _LP.csv file. Spaces are permitted. Commas are not permitted.	Char (30)
3	Comment	Optional field for entering comments. Not used.	Char (250)

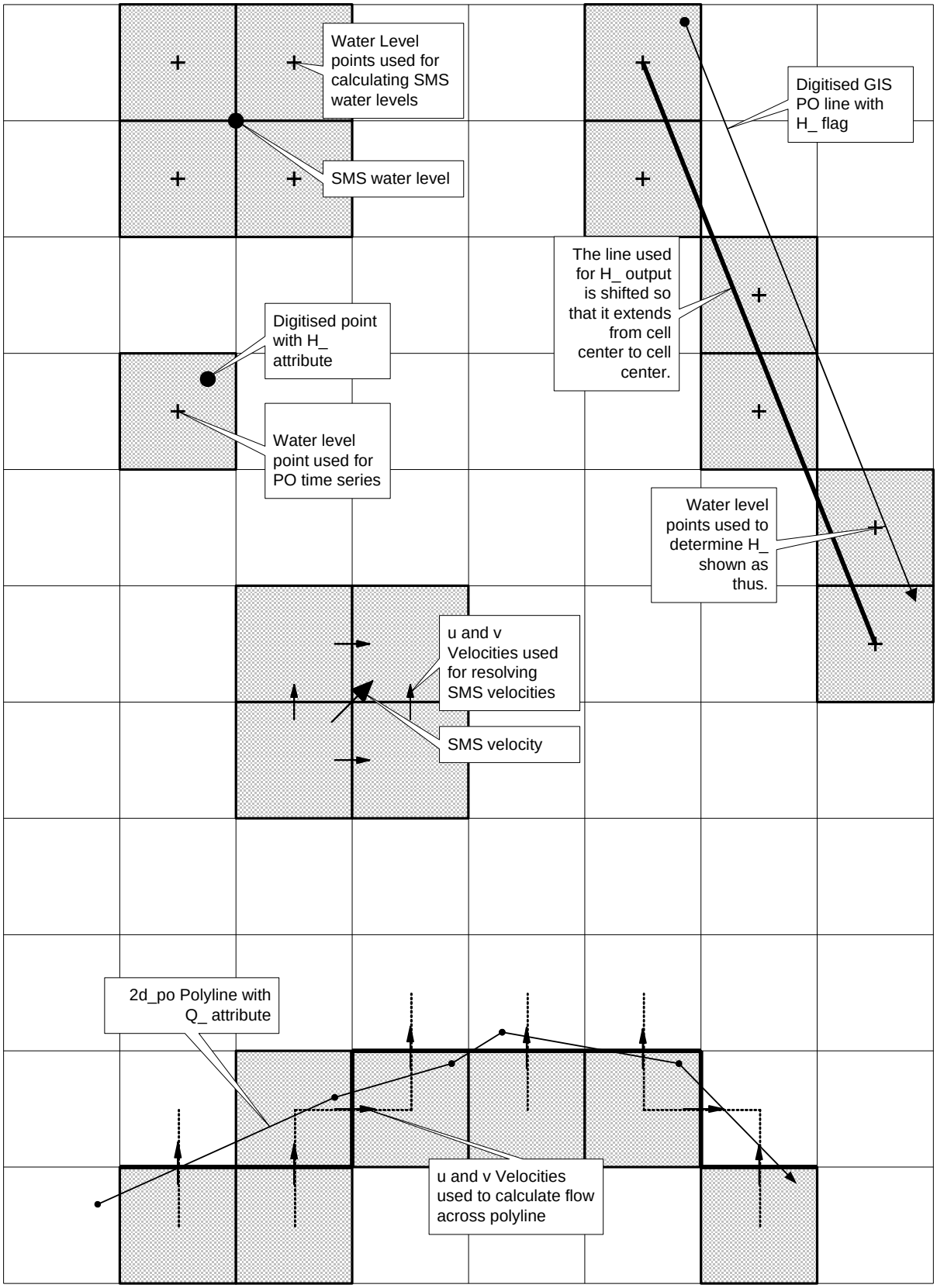


Figure 9-2 Interpretation of PO Objects and SMS Output

9.4 Customising Map Output

9.4.1 Overview

TUFLOW has a range of commands that allows the user to control which formats, data types, frequency, sub-areas (output zones) of a model to produce output. The options are highly flexible, and can be customised to be different for different output formats and/or output zones as discussed in the following sections.

For example, time based map output maybe produced using one format, and just the peak flood level surface for another output. Output over a township maybe at a higher output interval frequency than for the whole model so as to produce a higher quality animation of flooding through the township. For details refer to the information on each command. The main commands of interest are:

- Map output file format(s) to be written (Map Output Format).
- Map output data types, such as water level, stream power, etc (Map Output Data Types).
- When and how frequently to write map output (Start Map Output, Map Output Interval).
- Whether to track the maximums and minimums (Maximums and Minimums).

For example, application of the above commands may look like the below:

```
Map Output Data Types == h V d  ! Output water level, velocities and depths
Start Map Output == 10  !  Start map output at 10 hours
Map Output Interval (s) == 300  !  Write map output every 5 minutes (300s)
Map Output Format == xmdf asc  !  Produce map output in XMDF and ASC formats
```

9.4.2 Changing the Output for Different File Formats

A variety of map output formats can be output for a single simulation. The supported formats are discussed in detail in Section 9.7 and are specified using the .tcf command Map Output Format.

In addition, output for each map output format may be customised to be different to the default or global settings by including the map output format acronym at the start of the command. For example, WRR Map Output Interval == 360 will set the output frequency to 360s just for the WRR format. The format acronym must be identical to that used in Map Output Format. The following commands can be applied in this manner.

<format> Start Map Output
<format> End Map Output
<format> Map Output Interval
<format> Map Output Data Types

Note: to apply the same setting to more than one format, the command needs to be repeated for each format. If a format is not customised the default setting or the setting applied to the whole of the model is used.

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

```
XMDF Map Output Interval == 360
```

To only output depths (d) and Bed Shear Stress (BSS) in ASC format:

```
ASC Map Output Data Types == d BSS
```

Note the order of commands is important. Ensure that commands defining the map output settings for all format types are read in prior to any commands specific to a certain format type.

For example, the following commands set a Map Output Interval of 120 seconds for the XMDF and DAT formats and 3600 seconds for the ASC format.

```
Map Output Format == ASC XMDF DAT
Map Output Interval == 120
ASC Map Output Interval == 3600
```

Swapping the order of the second and third command lines will set a Map Output Interval of 120 seconds for all output formats as the third command overwrites the ASC Map Output Interval command.

```
Map Output Format == ASC XMDF DAT
ASC Map Output Interval == 3600
Map Output Interval == 120
```

9.4.3 Output Zones – Enhancing Map Output

Output Zones are a powerful feature that allows users to generate map and check file outputs for sub-regions of a model. Different Output Zones can have different output formats, start/end times, frequency of output, and output data types. Any number of Output Zones can be defined for a model, and all, a selection or none of the Output Zones can be activated when the model is run.

Output zones are very useful, particularly for large models, where higher frequency map output is required for a portion of a model, for example to create an animation for an urban region. Another useful purpose is to create an Output Zone for a problematic section of model that requires closer examination, by generating output on a fine time interval during the period in question. Another benefit is simply reducing the size of the output files by only outputting where information is required, and disabling the output for the whole model using Map Output Entire Model == OFF. Output zones currently only apply to TUFLOW's map and GIS check file outputs (ie. does not apply to time series and other plot outputs).

Note the Model Output Zones command is used to select which Output Zones are to be used. If this command is omitted, no output from the Output Zones is written. Separate multiple Output Zones using a "|" (vertical bar or pipe character). For example, to output from zones ZoneA and ZoneC specify:

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

If there are numerous Output Zones, it is recommended that the definition se are placed in one or more separate files and use the Read File command to reference these file(s) so as to keep the size of the .tcf file minimal. This however is not a requirement.

Each Output Zone is defined using a definition block as follows:

```
Define Output Zone == <ZoneA>

  …..

End Define
```

The following commands can be used within an Output Zone definition. With the exception of Read GIS Output Zone, all commands are optional. The optional commands can be used to change the default setting or the setting applied outside the Output Zone definitions.

Read GIS Output Zone
Map Output Format
Start Map Output
End Map Output
Map Output Interval
Map Output Data Types
Output Folder
Maximums and Minimums
Maximums and Minimums Only for GRID
Write Check Files

The Read GIS Output Zone reads a GIS layer containing one or more polygons that define the regions to be output. The attributes of the layer are not used.

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

Map Output Entire Model can be used outside an Output Zone definition block to turn on or off map output for the entire model (the default is ON). If set to OFF only map output for any Output Zones is written. Map output commands that occur outside Output Zone definitions apply to the entire model output (ie. as is the case in previous releases).

Example 1: Defining an Output Zone

The example below defines Output Zone "ZoneA". In the example, a DEM_Z check grid will be the only check layer written for the Output Zone, and Grid, WRB and XMDF outputs will be produced. The Grid output will consist of water level (h) and VxD (Z0) results, starting at time 0.5 hours and ending at 4 hours at an interval of 0.5 hours. The WRB and XMDF output files will contain water levels

(h), velocities (V) and the ZMBRC hazard categories starting at the simulation beginning and finishing at 6 hours at an interval of 6 minutes.

```
! Entire Model settings
Map Output Format == ASC XMDF
Map Output Interval == 3600  !default output interval is hourly
ASC Map Output Interval == 0 !maximums only for .asc output
Map Output Data Types == h v q d z0 !default output types

Model Output Zones == ZoneA | ZoneB !Output both of the zones below

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

9.4.4 Gauge Level Map Output (2d_glo)

The 2d_glo GIS layer uses the .tcf command Read GIS GLO and writes map based output data when the water level at the gauge reaches user defined levels.

Note: Map output based on reaching gauge levels replaces the conventional approach of using a Start Map Output time and a Map Output Interval.

The 2d_glo GIS layer is used to define the location of the gauge within the modelled extent. The gauge is digitised as a point object within a 2d_glo layer with the attributes as described in Table 9-6 and referenced within the .tcf using the command Read GIS GLO.

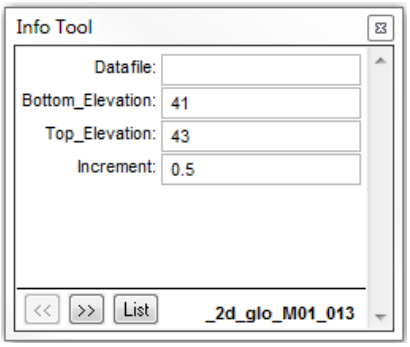
The water level at the gauge location is tracked by TUFLOW throughout the simulation and output to the Console DOS Window (see Section 12.2) preceded by the letters "GL". When the water level is within a user specified range, the map output results (see Section 9.4) at that simulation time will be written. The results are therefore useful for mapping the predicted area of inundation for specified gauge heights.

The range of water levels at the gauge for which the results are written may be defined using one of two methods:

- By referencing a .csv file within the 2d_glo GIS layer using the first attribute. The .csv file contains a single column of levels, and comment lines are allowed using the "!" or "#" character. If a .csv file is specified in the first attribute, the remaining attributes are not used, but dummy or empty values for these attributes must exist.
- By populating the remaining attributes of the 2d_glo GIS layer as described below.

When using the attributes of the 2d_glo GIS layer to define the gauge heights, the map output results are first written when the water level at the gauge reaches the specified "Bottom_elevation". Subsequent results are written as the water level at the gauge rises based on the value of the "increment" attribute. The map output ceases once the water level at the gauge reaches the "Top_elevation".

For example, if the 2d_glo GIS layer has been defined as shown below, map output results will be written when the water level at the gauge reaches 41m, 41.5m, 42m, 42.5m and 43m.



Only one gauge location may be specified per model simulation. If more than one object exists within the 2d_glo layer, the gauge that is monitored by TUFLOW will be the last digitised object. Similarly, if the Read GIS GLO command is used more than once, only the last occurrence of the command will be used.

Table 9-6 2D Gauge Level Output (2d_glo) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS GLO Command			
1	Datafile	Filename (and relative or full path if needed) of the file containing the gauge levels to trigger map output. Must be a comma or space delimited text file such as a .csv file. Only the first column is used and this column must contain the gauge levels. Comment lines using a leading "!" or "#" can be used at any point within the file. If this attribute is blank, the following three attributes are used to define the gauge levels to trigger map output. If this attribute is not blank, the following three attributes are not used, but they must exist and can simply be populated with empty or default values.	Char (254)
2	Bottom_Elevation	The water level in m above datum at the gauge at which the writing of map output results will commence.	Float
3	Top_Elevation	The water level in m above datum at the gauge at which writing of the map output results will cease.	Float
4	Increment	The water level increments in metres between the Bottom_Elevation and Top_Elevation at which the map output results will be written.	Float

9.5 Including 1D Results in Map Output

9.5.1 Overview

1D domain results can be output in combination with 2D domain(s) by using the 1d_wll GIS layer and the Read GIS WLL 1D command. 1d_wll GIS layer(s) are used to define and control the 1D map output. The layer(s) contain lines (called Water Level Lines or WLLs) 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 WLLs is important. They must be digitised from left to right looking in the positive direction of the channel.

When viewing the results, if the 1D WLLs and 2D domains overlap, the 1D results are displayed on top of the 2D results. However, depending on the viewing platform, when observing the scalar and vector magnitudes as the cursor is moved around, the 2D values maybe given precedence over the 1D where the overlap occurs.

Two WLL options are available. The preferred method can be specified by using the command WLL Approach. Of the two options, Method A and Method B, Method B allows more advanced and accurate mapping of 1D results in map formats and is the default and recommended method, and that described below. For documentation on Method A, refer to the TUFLOW 2010 Manual.

It should be noted that water level lines do **NOT** change the 1D hydraulic computations, they are purely used in order to display the 1D results in plan (2D) formats.

9.5.2 Water Level Lines (WLL, 1d_wll)

Ground elevations and optionally material (Manning's n) values can be assigned to points along a WLL. A more accurate representation of 1D domain velocity and flood hazard can be mapped using this approach. 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.

WLLs can have any number of vertices. The treatment of WLLs is as follows:

- To pick up the water level 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.
- If a WLL crosses two or more channels, the channel closest to the middle vertices (3 point line) or half-way point (2 point line) is used.

- If a WLL middle vertex snaps to a node with two or more channels on the upstream side, the channel that is closest in angle to the WLL's perpendicular (based on the WLL's two end points) is used.
- For 4 or more vertices, one of the middle vertices (ie. not an end vertex) must snap to a vertex on the channel line.

There is one attribute required as described in Table 9-7. The attribute, Dist_for_Add_Points, is the minimum distance in metres along which to generate elevation points for that line. If Dist_for_Add_Points is zero, only elevations at the vertices along the WLL are generated.

Table 9-7 1D Water Level Line (1d_wll) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS WLL Command			
1	Dist_for_Add_Points	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

Use the 1D command Read GIS WLL to specify the 1d_wll layer and automatically create 1D map output for TUFLOW (ESTRY) 1D domains. For Flood Modeller or XP-2D linked TUFLOW models, use Read GIS ISIS WLL or Read GIS XP WLL in the **.tcf** file to read the 1d_wll layer(s). The Flood Modeller 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.

Several 1d_wll layers can be specified covering different sections of the 1D domain(s) if required. A TIN of triangular elements are created between the WLLs in the .2dm mesh file – view these in SMS or a GIS to check they have been created correctly.

The default approach uses the processed cross-section data (height –width data) from the channel for setting elevations at each WLL point. For Flood Modeller and XP-SWMM, this data is automatically fed through to TUFLOW and is based on the cross-section information entered into the Flood Modeller/XP-SWMM model.

If a WLL is snapped to a node, the processed data used for setting any bed elevations is 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).

9.5.3 Water Level Line Points (WLLp)

If Write Check Files in the .tcf file is specified, two GIS check layers are created from the WLLs. These are labelled 1d_WLLo and 1d_WLLp. 1d_WLLo (Water Level Line Objects) reproduces the WLLs with attributes containing the channel and node the WLL is allocated to for cross-checking purposes.

1d_WLLp layers (Water Level Line Points) contain all of the elevation points generated based on the Dist_for_Add_Points attribute. This layer can then be used to allocate elevations (first attribute) to each point from a DEM (in the same manner the 2D Zpts can be 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 integer material value at each point by using a GIS to assign values from material polygons. The material value must exist in the .tmf file or a materials .csv file (see Read Materials File).

The attributes of a 1d_WLLp 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 9-8. A parallel channel analysis is used to provide varying velocities across the water level line. Note that the WLL level lines do **NOT** change the 1D hydraulic computations, they are purely used in order to display the 1D results in plan (2D) formats.

For Flood Modeller and XP-SWMM, the layers are essentially the same, but are named using xWLLo and xWLLp.

Note: If using Read GIS WLL Points or Read GIS X1D 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 Dist_for_Add_Points 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 Dist_for_Add_Points value.

Table 9-8 1D Water Level Line Point (1d_wllp) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS WLL Points Command			
1	Z	Ground elevation of the point. Automatically generated from the channel cross-section processed data or updated based on a point inspected from a DTM.	Float
2	RR or Material	In the 1d_WLLp check file, the relative resistance of the point. A value of 1 is assigned if the elevation was estimated from the channel's processed data. 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 or Read GIS X1D WLL Points, this column should either be set to: • zero (0) to force a relative resistance of 1; or • An integer 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

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 15.2.1) when extracting obverts of structures for longitudinal profiles.

A useful tip at a junction of 1D channels is to use a connector for any side channels (Type = "X" – see Section 5.8.3). 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.

When using Method B triangular regions can also be used in the 1d_WLL format. This will manually create the triangulation, rather than relying on the automated TUFLOW triangulation as described in the following section. This is particularly useful at junctions or between parallel channels to avoid steps in the output.

9.5.4 Manually Adding Triangles into the 1d_WLL Layer

Triangle region (polygon) objects with three vertices can be digitised in the 1d_WLL layer. This is particularly useful at junctions or between parallel channels to infill areas and avoid steps in the output surfaces. The triangles must snap to the ends of WLL line objects. In the example in Figure 9-3 the water level lines are shown in red and the triangles are shown as yellow. These are connected to the ends of the water level lines. The map output surfaces will interpolate over the triangle based on the values at the snapped WLLs.

If the region object is not correctly snapped an ERROR 1311 message occurs, pointing to the vertex on the triangle that is not snapped.

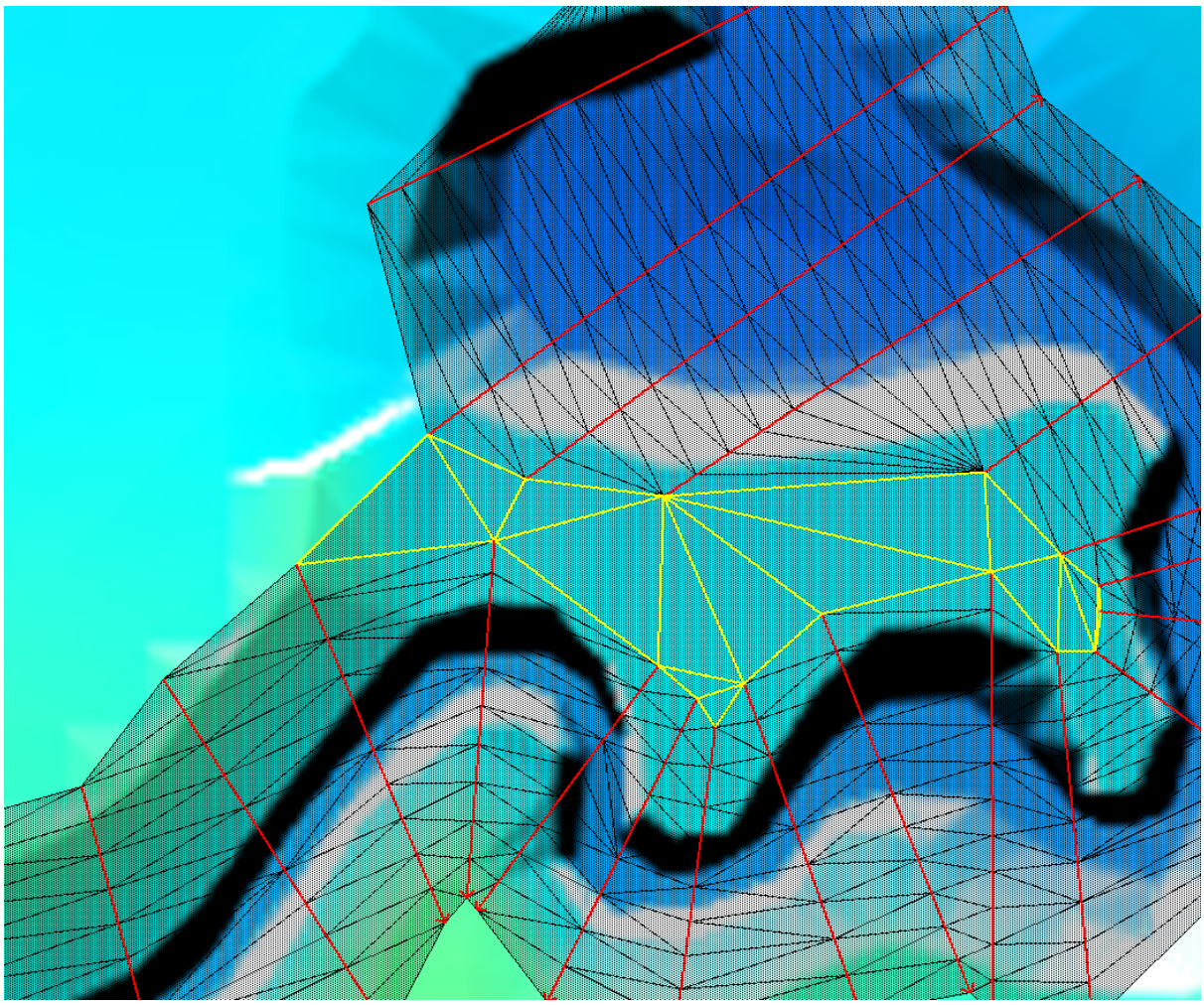


Figure 9-3 Adding Triangles into 1d_WLL Layer to Infill Areas

9.6 Map Output Formats

9.6.1 Overview

The TUFLOW 2016-03 release offers a wide range of map output formats, all publicly available, to cater for a range of GIS and GUI software. There are no constraints over how many of these output formats a single simulation can produce, and different formats can have different output settings and different regions of the model can output in different formats as well as vary other map output controls.

Map output is offered in the following forms:

- **Mesh Based** (see Section 9.6.3): The output is based on a quadrilateral and/or triangular mesh of the 1D and 2D domains. This output is essentially the closest reproduction of the hydraulic calculations, with minimal interpolation from the 1D and 2D computational points. These formats include: DAT, T3, TMO, WRB, and XMDF.
- **Grid Based** (see Section 9.6.4): The output is over a regular north-south grid in a similar manner to a raster DEM. The hydraulic output at each output grid cell is interpolated from the computational mesh using the Mesh Based output above. These formats include: ASC, FLT, NC (NetCDF), TGO, WRR.
- **Combination Mesh and Grid** (see Section 9.6.5). These formats include: WRC.
- **GIS Based** (see Section 9.6.6): GIS layers (in a similar manner to using the TUFLOW_to_GIS utility) can now be written as the simulation proceeds. The format is set by GIS Format.

The selection of map output formats is controlled by the .tcf command Map Output Format. One or more output formats may be specified for the whole model or for an Output Zone (refer to Section 9.4.3). The only restriction is that the SMS HIGH RES option cannot be specified in combination with any of the other formats.

If no output format has been specified, the results are written by default for the 2016-03 release using the XMDF format. If Defaults == Pre 2016 is set the default is the DAT format, which has been the default for all versions of TUFLOW prior to the 2016-03 release. The following sections describe the supported map output formats.

Table 9-9 at the end Section 9.6 provides a summary of all Map Output Formats.

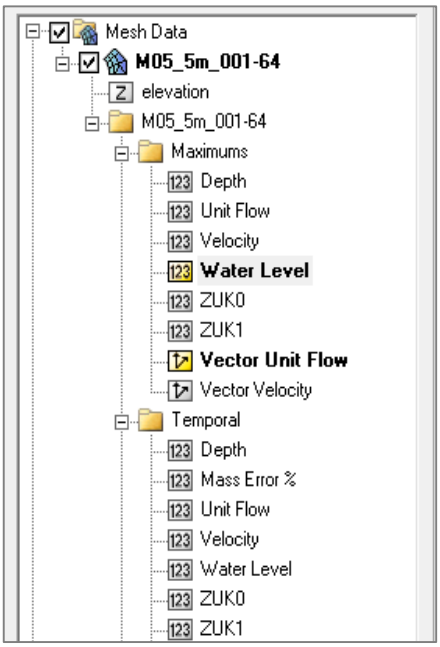
9.6.2 Which GIS/GUI Supports Which Formats?

The formats supported by commonly used GIS and GUI software are provided on this TUFLOW Wiki page. Note that GIS and GUI software are continually adding and supporting new formats, so we will attempt to keep the wiki page up-to-date!

9.6.3 Mesh Based Map Output Formats

9.6.3.1 DAT and XMDF

SMS formatted data can be output in .dat or .xmdf format, or both. If the .dat output format is used, a separate file is written for each output type specified using the command Map Output Data Types (see Table 9-10). If the .xmdf output format is specified, only a single .xmdf file is output containing all map output types. For both .dat and .xmdf formats the .sup and .2dm files are output. These file types are described in Table 2-3.



The .xmdf output has a number of advantages over the .dat format:

- All map output is contained within a single file (ie. all .dat files are consolidated within a single .xmdf file).
- It is much faster to access and loads instantly when first opened in SMS 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 is stored as either static or temporal (time) based.

The envelope of maximum and/or minimum values is available for some output types using the options in Maximums and Minimums. Minimums are assigned a time of –99999.0 and maximums a time of 99999.0 in the .dat file format, for the .xmdf files these are stored in separate "Maximums" and "Minimums" folders. For the .xmdf format the maximums and minimums are automatically filtered out as separate datasets, therefore the 99999 times do not appear in the time list if viewed in SMS.

For the .dat format, the _Times.dat file is automatically written. It contains the time when the maximum water level and velocity value occurred. This data is also written to the .xmdf file and will appear under a separate "Times" folder.

Additional time outputs are available using Time Output Cutoff Depths or Time Output Cutoff Hazards to output maps of the duration of inundation and the time of first inundation above specified depth(s) or VxD(s) to the _Times.dat or .xmdf file.

The Map Output Format 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 proportion of upstream controlled flow regimes. Some of the utility programs such as res_to_res.exe do not yet recognise this format (TUFLOW_to_GIS is compatible with this format). This option also requires

extra RAM and the SMS .dat or .xmdf files are roughly four times the size of the standard default output format. Note that the SMS HIGH RES option cannot be used in combination with other output formats.

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 file size. This format is compatible with the TUFLOW utilities.

9.6.3.2 TMO

The .tmo output format is utilised by 12D Solutions for their TUFLOW GUI interface. If using 12D to display/view results this format will need to be specified using Map Output Format. This format contains 2D domain cell centred outputs from the model.

Note, the Output Zones feature (refer to Section 9.4.3) is currently not yet available for the TMO format.

9.6.3.3 WRB

WaterRIDE by Worley Parsons is commercial software for visualising and post-processing hydraulic modelling results. TUFLOW supports the WRB (this section), WRC (Section 9.6.5.1) and WRR (Section 9.6.4.4) WaterRIDE formats.

One .wrb file is produced for each simulation that contains the model's ground/bathymetric elevations, water levels, velocities (scalar and vector), and optionally the Z0 (VxD product) and one hazard category. The .wrb format is restricted to these data types, other data types specified using Map Output Data Types are ignored if WaterRIDE output is used. WaterRIDE can only display a single hazard category (the first one is used if more than one is specified). If Z0 and/or a hazard category are not specified for WRB output, WaterRIDE can optionally post-process these hazard values. Other data types such as depth are also post-processed by WaterRIDE.

If maximums are tracked (see Maximums and Minimums) these are also added to the .wrb file for the data types mentioned above. Note that if WaterRIDE is used to post-process maximums the values will be different to those provided by TUFLOW. TUFLOW tracks maximums every timestep, whilst WaterRIDE post-processes maximums using the values in the .wrb file which only occur every Map Output Interval.

The .wrb output utilises the triangular mesh output for as discussed in Section 9.6.7.2.

9.6.3.4 T3

Blue Kenue from the National Research Council Canada is free software for visualising and animating time varying hydraulic model map output. TUFLOW Blue Kenue output utilises the .t3s and .t3v formats. The .t3s file contains fixed or temporal scalar data and the .t3v file contains vector data.

Five files are produced per simulation as follows:

- .Z.t3s: Contains the ground elevations/bathymetry of the model. This file is the ASCII form of the .t3s format so can be viewed using a text editor.

- .Ts.t3s: Contains the temporal scalar map output for all data types specified using the binary format of the .t3s format.
- .Tv.t3v: Contains the temporal vector map output. At present this format can only support one data type, so if, for example, V and q data types have been specified, only V will be included (V and q as scalar output will appear in the .Ts.t3s file). Discussions with Blue Kenue developers are underway to allow this format to support multiple data types. The file is a binary format.
- .Ms.t3s: If maximums are being tracked, this file contains the maximums for scalar data types in binary format.
- .Mv.t3v: If maximums are being tracked, this file contains the maximums for the vector data type in binary format. The comment above for .Tv.t3v files also applies to this file.

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

Blue Kenue only supports triangular meshes, so all Blue Kenue output at present utilises the triangular mesh output as described in Section 9.6.7.2.

9.6.4 Grid Based Map Output Formats

9.6.4.1 ASC, FLT

The Map Output Format ASC and FLT options write results out in these standard GIS grid formats. This handles rotated 2D domains, multiple 2D domains and 1D WLL meshes. All Map Output Data Types are supported, and the resolution of the output grids can be set using the command Grid Output Cell Size. By default, the resolution is half the smallest 2D cell size. At present only one output grid resolution is possible, so the last occurrence of the command will prevail. It is planned to allow different resolutions to be specified for different map output formats and different Output Zones.

The command Maximums and Minimums Only for GRID can be used to control whether only grids of the maximums are required. Alternatively the ASC or FLT Map Output Interval can be set to zero as follows to trigger only outputting the maximums.

```
ASC Map Output Interval == 0   ! Only output result maximums for ASC grids
FLT Map Output Interval == 0   ! Only output result maximums for FLT grids
```

The approach utilises a similar method to that used by TUFLOW_to_GIS (see Section 15.2.1) using the –asc option. TUFLOW pre-processes and stores in memory the interpolation factors at each GIS Grid cell. This significantly reduces the computational time to write each grid, so writing via TUFLOW is much faster than via TUFLOW_to_GIS. This approach does, however, consume more RAM for a simulation.

Alternatively, instead of including ASC or FLT in the Map Output Format list, this can also be specified using the Map Output Format == GRID. In this case the grids are written by default in FLT binary

format. This can be changed to ASC by specifying the .tcf command Grid Format == ASC. Also note that a DEM of the final Zpts is now automatically output if writing check files (unless it is excluded using Write Check Files Exclude == DEM_Z).

The resolution of the output grid can be specified with the command Grid Output Cell Size command. The origin of the output grid is rounded to the nearest cell size. This can be set to the exact model origin using the Grid Output Origin == Model Origin.

All time outputs are supported by the grid map output formats. The grid file extensions are:

- _TDur_<cutoff> for duration of inundation;
- _TExc_<cutoff> for time <cutoff> is exceeded;
- _TMax_h for time of peak water level; and
- _TMax_V for time of peak velocity.

9.6.4.2 NC – NetCDF

The NETCDF (Network Common Data Format) is a commonly used format for storing modelling and scientific data. TUFLOW supports the output of raster data into the NetCDF file format via the NC option. A single NetCDF file is created that contains both the time varying and static output (eg. maximums, and time outputs). The output is a north-south aligned raster and includes outputs from multiple domains and 1D WLLs. The interpolation is the same used by the ASC, FLT, WRR and TGO formats.

A number of NetCDF specific commands are supported as listed below. The TUFLOW Wiki page TUFLOW NetCDF Raster Format also provides additional information.

- NetCDF Output Compression
- NetCDF Output Start Date
- NetCDF Output Time Unit
- NetCDF Output Direction
- NetCDF Output Format

If using FEWS to view NC output, please ensure that NetCDF Output Format == FEWS has been set.

9.6.4.3 TGO

The TGO format is utilised by 12D Solutions for their TUFLOW interface. The output is a north-south aligned raster and includes outputs from multiple domains and water level lines. The interpolation is the same used by the ASC, FLT, WRR and NC formats, and unlike the .tmo format, all Map Output Data Types are supported, and the resolution of the output grids can be set using the command Grid Output Cell Size.

9.6.4.4 WRR

A WaterRIDE format that contains the time varying grid output is a north-south aligned raster and includes outputs from multiple domains and 1D WLLs. The interpolation and grids is the same as would be produced using the ASC and FLT formats. All Map Output Data Types are supported, and the resolution of the output grids can be set using the command Grid Output Cell Size.

9.6.5 Mesh and Grid Combined Map Output

9.6.5.1 WRC

The WRC format produces a master (.wrc) file and one or more WRR and WRB files. The approach adopted by TUFLOW is as follows:

- Each 2D domain is output as a rotated WRR format grid with the cell size equal to the TUFLOW cell size. As discussed previously, the WRR format is significantly faster to operate with than the WRB format due to its grid based formatting. The output values are the cell centre values for each 2D cell.
- Any 1D WLL triangulations are output as a separate WRB file.
- The WRC "master" file is output to the specified results folder, while the WRR and WRB files are written to a "waterRIDE" sub folder.

9.6.6 GIS Based Map Output Formats

Gridded output format as GIS layers can be written directly from TUFLOW during the simulation by including "GIS" in the Map Output Data Types command. This offers a similar functionality to that using the TUFLOW_to_GIS utility via the –mif or –shp options. The output GIS format is controlled by the GIS Format command. For scalar outputs, these are output as point a GIS file with a separate file for each output time. For vector outputs, this can be either as a point or region GIS file. Specific commands to this output type are:

- GIS Grid Vector Type
- GIS Grid Vector Direction
- GIS Grid Vector SF
- GIS Grid Vector TTF

The commands above can be applied to all vector outputs or can be specific to the data type by prefixing with a "v", "q" or "W", for velocity, unit flow or wind respectively. The example below sets the scale factor to 1 for all outputs except unit flow, which has a smaller factor of 0.1.

```
GIS Grid Vector SF == 1.0    ! Set scale factor for all GIS vector output to 1.0
q GIS Grid Vector SF == 0.1    ! Set scale factor for unit flow GIS output to 0.1
```

9.6.7 Mesh Configurations (.2dm File)

9.6.7.1 Quadrilateral and Triangle Mesh Option

The default mesh used by the DAT and XMDF formats uses quadrilaterals (squares) for the 2D cells and triangles for representing any 1D WLLs as discussed in Section 9.5. Each 2D domain and the 1D WLL triangles are treated as separate meshes, although they will appear as one when viewing the .2dm file.

The advantage of this format is that the amount of data output is optimised keeping file sizes small with little loss of accuracy in translating results from the computational mesh.

Only the DAT and XMDF formats support the combination of quadrilaterals and triangles.

9.6.7.2 Triangular Mesh Option

Except for the DAT and XMDF formats, all the other mesh based formats use a triangular mesh to represent the computational 1D and 2D domains. The triangular mesh is also used by the grid based formats (ASC, FLT, NC, TGO, WRR) to interpolate from the triangular mesh the values at the grid's cell centres (this is the output grid's cell centres, not the 2D domain grid cells).

To utilise this format using the DAT and XMDF formats, specify SMS TRIANGLES anywhere in entry for the Map Output Format command.

The triangular mesh option incorporates output at the cell centres, so the exact water level calculated by TUFLOW at the 2D domain cell centres is used when translating results, thereby providing a slightly more representative surface of the hydraulic calculations. Each 2D cell is represented as four triangles with a common vertex at the cell centre, giving a higher resolution spatial output than just using the 2D cell corners.

However, the number of nodes in the mesh for the 2D domains increases by 20% and the number of elements by a factor of 4 compared with the Quadrilateral and Triangle Mesh Option, therefore, the output file sizes will be larger and the viewing and processing times possibly slower. Any triangular elements from 1D WLLs are not affected and remain the same in either mesh arrangement.

9.6.7.3 2D Cell Corner Interpolation/Extrapolation

Irrespective of the mesh option being used, the Map Output Data Type values at the 2D cell corners need to be interpolated (or extrapolated if at the wet-dry interface). Improved methods for extrapolating values at the 2D cell corners at the wet/dry interface were introduced as part of the 2013-12 release. Previous methods had issues with tracking the maximum hazard outputs, particularly on steep slopes at the wet/dry interface where the extrapolated value at the cell corners on the dry side was exaggerated as it was translated horizontally.

Two new methods were developed that can be set using the command Map Output Corner Interpolation == METHOD B or METHOD C. Method C (the default) is recommended as it is simpler and extensive testing has indicated it resolves the issues. METHOD B largely resolves the issues and has the added

advantage that the effect of thin breaklines is better handled whilst they are dry or upstream controlled flow occurs. The pre 2013-12 release approach can be used by specifying Map Output Corner Interpolation == METHOD A (this is also invoked if Defaults == PRE 2013 is specified).

For model results where the water has risen upwards (eg. river flooding), the Method B or C approaches should cause no significant changes in results (ie. fractions of a mm). Where the flow is downwards over steep slopes, some changes in results at cell corners will occur, but usually only slightly. However, maximum hazard values on very steep slopes may experience a more significant change.

Table 9-9 Map Output Format Options

Map Output Format Option	Description
Mesh Based Output Formats Mesh based map outputs use a mesh of quadrilaterals and/or triangles of the 2D cells and any 1D WLLs to manage and store the model results. The mesh is defined in the .2dm file that is written with the map output files.	
DAT (Section 9.6.3.1)	A publicly documented format developed by Aquaveo for the SMS GUI software. This was the default output format prior to the 2016-03 release if the command Map Output Format was not specified. This format, whilst popular for many years, does not utilise a header index (therefore slow to open the file) and does not support compression. Also a .dat file is produced for every Map Output Data Type specified, which has its pros and cons.
T3 (Section 9.6.3.4)	Published format used by the free Blue Kenue software produced by the National Research Council Canada for visualising and animating time varying hydraulic model map output. Results are only output in the cell centred triangular mesh option (see Section 9.6.7.2) as Blue Kenue does not support quadrilateral elements in a mesh.
TMO (Section 9.6.3.2)	TUFLOW Map Output format developed for use by the 12D Solutions TUFLOW interface.
WRB (Section 9.6.3.3)	WaterRIDE triangulation format for visualising and post-processing hydraulic modelling results. Results are only output in the cell centred triangular mesh arrangement (see Section 9.6.7.2) as WaterRIDE does not support quadrilateral elements in a mesh.
XMDF (Section 9.6.3.1)	XMDF was developed by Aquaveo as a faster and more space efficient replacement to the DAT format. The XMDF format complies with the HDF5 standard. The advantages of the XMDF format over the DAT format are: all specified Map Output Data Types are in the one file; clearer labelling of output types and maximum outputs; an internal folder structure is available for use by GUIs; and supports compression (see XMDF Output Compression). The XMDF format is also now supported by most GUIs that support the DAT format, GIS Plugins like Crayfish and the TUFLOW Utilities. For these reasons XMDF is now the default setting for the 2016-03 release.

Map Output Format Option	Description
Grid Based Output Formats Grid based formats output the results over a regular grid in similar manner and formats to DEMs and other raster based data. The output grid is not necessarily the same as the 2D grid and will also use any 1D WLL triangulation to interpolate output. Grid based formats are highly suited for use in GIS, and especially for handling output from very large models (eg. GPU Models). The resolution of the output grid is controlled by the Grid Output Cell Size command.	
ASC (Section 9.6.4.1)	ESRI ASCII (.asc) grid format, an industry standard format often used for transferring 3D surfaces between GIS software. The output grids are placed in a sub-folder called "grids". The resolution of the grid output can be set using the command Grid Output Cell Size. By default, the grid output resolution is half the smallest 2D cell size. Grids are written every map output interval and for the maximums/minimums at the end if maximums and/or minimums are being tracked. To only output the maximum/minimum grids at the end of the simulation use Maximums and Minimums Only For Grids. If SHP Projection was specified a .prj file is written, which will be used by GIS software that recognise this format to assign the GIS projection to the 3D surface. Advantages of this format are that it is very simple and can be viewed in a text editor. Limitations are that for large grids the file maybe slow write, slow to open and work with, file sizes are large and a separate file needs to be written for every Map Output Data Type specified and for every output time (that can be a lot of files!).
FLT (Section 9.6.4.1)	ESRI binary (float) version of the ASC format described above. The file header containing the dimensions of the grid is output to a .hdr (text) file and contains the same header as for an .asc file. The remainder of the output, the 3D surface values, is written to a .flt file as a binary dump rather than as a text file. The output grids are placed in a sub-folder called "grids". The resolution of the GRID outputs can be set using the command Grid Output Cell Size. By default, the resolution is half the smallest 2D cell size. Grids are written every map output interval and for the maximums/minimums at the end if maximums and/or minimums are being tracked. To only output the maximum/minimum grids at the end of the simulation use Maximums and Minimums Only For Grids. If SHP Projection was specified a .prj file is written, which will be used by GIS software that recognise this format to assign the GIS projection to the 3D surface. Advantages of this format are that it is very simple, and is extremely fast to write and open. Main limitations are that file sizes are large, and a separate file needs to be written for every Map Output Data Type specified and for every output time (that can be a lot of files!).

Map Output Format Option	Description
GRID (Section 9.6.4.1)	Outputs grid files in either .asc or .flt format. The .flt format is the default unless Grid Format == ASC has been specified in the .tcf. Refer to description above for the ASC and FLT formats.
NC (Section 9.6.4.2)	Writes a NetCDF .nc file containing the same grids as would be generated using the ASC or FLT options. There a range of commands to vary the attribute information written to the .nc file header (see Section 9.6.4.2)
TGO (Section 9.6.4.3)	
WRR (Section 9.6.4.4)	
Mesh and Grid Based Output Formats Combination of a mesh and grid based approach.	
WRC (Section 9.6.69.6.5.1)	WaterRIDE format that writes a .wrc index file that references one or more .wrb and .wrr files. A .wrb file is produced for any 1D WLL triangulations and a .wrr file for each 2D domain rotated and sized to be identical to the 2D domain computational grid.
GIS Based Output Formats GIS based map outputs use the format specified by GIS Format to write out map output as a series of GIS layers.	
GIS (Section 9.6.6)	Outputs GIS layers using the format specified by the GIS Format command. These layers are similar to those that can be generated using the TUFLOW_to_GIS.exe utility.
Options for DAT and XMDF Formats The following options only apply to the DAT and XMDF formats. Only one (or none) can be specified.	
SMS	This is the default SMS option and outputs ground elevations and results at the cell corners only.
SMS HIGH RES	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 or .xmdf files are roughly four times the size.

Map Output Format Option	Description
SMS HIGH RES CORNERS ONLY	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 RES_to_RES.exe utility programs (refer to Section 15.2).
SMS TRIANGLES	Outputs 2D cells as four triangles rather than as a quadrilateral if DAT and/or XMDF are specified. The triangles are constructed so that the 2D cell centre is a common vertex to all four triangles. This means that the mesh is entirely constructed of triangles (four triangles per 2D cell and any 1D WLL triangles). **Note: For the formats that rely on this triangle only mesh (eg. ASC, FLT, NC, T3, TGO, WRB, WRR), the SMS TRIANGLES option does not need to be invoked. Only specify SMS TRIANGLES if you require your DAT or XMDF output to be based on a triangle only mesh rather than the default mesh of quadrilaterals for 2D cells and triangles for 1D WLLs. See Section 9.6.7.**

9.7 Map Output Data Types

TUFLOW can output a wide range of output types in map format. Table 9-10 describes all the non-hazard map output types, while Table 9-11 contains all the numerous flood hazard category (Z) map output types. The map output types produced by a simulation are controlled using the .tcf command Map Output Data Types.

The map output types' flags are listed in the first column of the tables, and are used to denote the type(s) to be output. They can occur in any combination or order and are not case-sensitive. For example to output water level, velocity and unit flow, enter the following line in the .tcf file:

```
Map Output Data Types == h V q
```

Although optional, it is strongly recommended that spaces are used between each data type for clarity.

The output types are available in a wide range of Map Output Formats and can be varied for different formats and between output zones (see Chapter 9). The various map output formats are described in Section 9.6. Note, not all Map Output Data Types are available for all Map Output Formats due to limitations or constraints of the format. The type/format compatibility is documented in Table 9-10 and Table 9-11.

Note that it is possible to get different output types for different output formats as discussed in Section 9.4.2. For example:

```
ASC Map Output Data Types == h V Z0
XMDF Map Output Data Types == h V q d ci z0
```

Table 9-11 presents the hazard category outputs. Of note is that each hazard is tracked every timestep for its maximum if Maximums and Minimums is set to ON or ON MAXIMUMS ONLY (the default) to ensure that the peak hazard category is recorded during the simulation. Up to ten (10) different hazard categories per simulation can be specified for map output.

Users also have the option to customise hazard output via the TUFLOW_USER_DEFINED.dll (refer to Section 11.4.4). Please email support@tuflow.com if you would like to code a unique hazard output that is currently not included as an output type option.

Table 9-10 Map Output Types (Excluding Hazard (Z) Types)

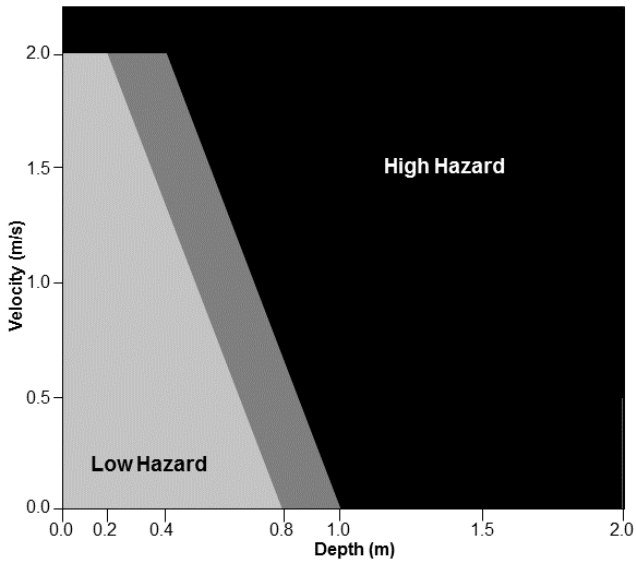
Flag	Map Output Data Type	Supported Formats	Description
AP	Atmospheric Pressure	All formats excluding WaterRIDE	Atmospheric pressure in hPa. Atmospheric Pressure is only applied if using the Read GIS Cyclone or Read GIS Hurricane commands. Maximum and minimum output is not available.
BSS	Bed Shear Stress	All formats excluding WaterRIDE	Bed Shear Stress as given by the equation below where ρ is density, g gravity, V velocity, n Manning's n and y depth: Metric Units: $\tau_{bed} = \frac{\rho g V^2 n^2}{y^{\frac{1}{3}}}$ English Units: $\tau_{bed} = \frac{\rho g V^2 n^2}{2.208\, y^{\frac{1}{3}}}$ The Bed Shear Stress map output can be misleading at very shallow depths as the BSS formula divides by the depth. The BSS and SP outputs are linearly reduced to zero once the depth is below a threshold (by default, 0.1m). This threshold can be changed using the .tcf command BSS Cutoff Depth. Maximum and minimum output is not available.
CI	Cumulative Infiltration	All formats excluding WaterRIDE	The cumulative infiltration over the entire simulation in mm or inches when a soils infiltration method has been used (see Section 6.10). See also the IR (infiltration rate) map output type below.
Cr	Courant Number	All formats excluding WaterRIDE	Courant number (2D domains only at present)
CWF	Cell Width Factor	SMS HIGH RES only	Cell side flow width factor. Used to monitor the changes in cell side widths over time if using Read GIS Layered FC Shape.
d	Depth	All formats excluding WaterRIDE	Water depths. For the cell cornered results formats (see Section 9.6.7.3) 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). Both maximum and minimum output is available.
dGW	Depth to Groundwater	All formats excluding WaterRIDE	Depth to groundwater (from the ground surface) over time in metres or feet when a groundwater depth or level has been defined (see Section 6.10.5).
E	Energy	All formats excluding WaterRIDE	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

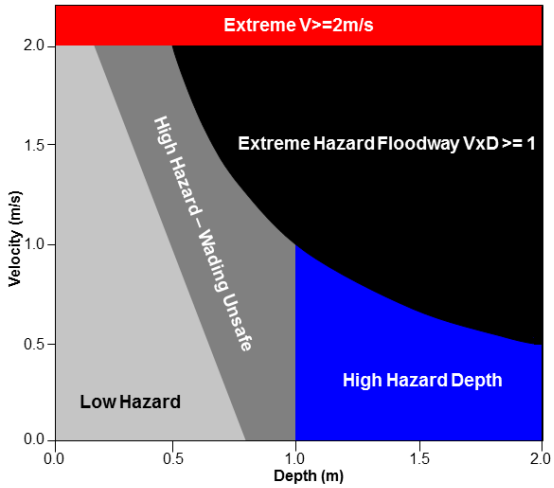
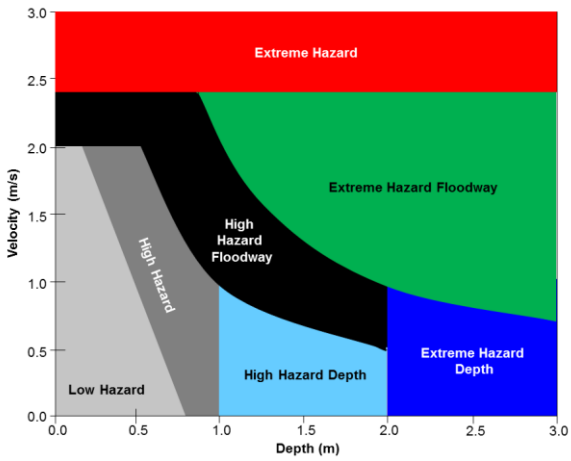
Flag	Map Output Data Type	Supported Formats	Description
			energy without interpolation is being trialled using the HIGH RES options (see Map Output Format). For 1D areas, this output should be treated with caution as it is derived from interpolation of water levels and approximations of the channel velocities across the WLLs, which can be problematic in 1D channels with high velocities. The energy output for 1D nodes is available as part of the plotting output (Section 13.2) Maximum energy levels 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.)**
F	Froude Number	All formats excluding WaterRIDE	Froude number output. No maximum and minimum output is available at this stage.
FLC	Form Loss Coefficient	SMS HIGH RES only	Form Loss coefficient. Used to monitor the changes in the form loss coefficient over time if using Read GIS Layered FC Shape.
h	Water Level	All formats	Water level output. For the cell cornered results formats (see Section 9.6.7.3) the water levels are interpolated from the water levels calculated at the cell centres. Both maximum and minimum outputs are available. 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).
IR	Infiltration Rate	All formats excluding WaterRIDE	The infiltration rate in mm/hr or inches/hr over time when a soils infiltration method has been used (see Section 6.10). See also the CI (cumulative infiltration) map output type above.
MB1	Mass Balance	All formats excluding WaterRIDE	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. This output does not include 1D output from WLLs.
MB2	Mass Balance	All formats excluding WaterRIDE	Same as MB1 above, but is accumulated over the entire simulation. This output does not include 1D output from WLLs.

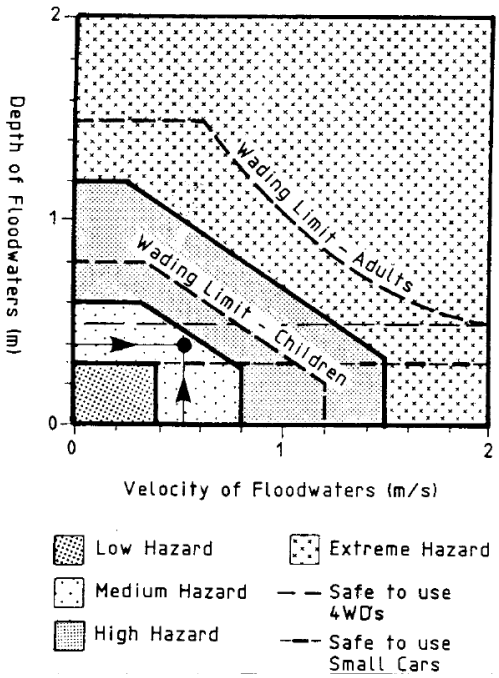
Flag	Map Output Data Type	Supported Formats	Description
n	Manning's n	All formats excluding WaterRIDE	Manning's n values. The n values only vary over time for materials using the Manning's n varying with depth feature. The n values at the cell corners in the _n.dat file are interpolated from the surrounding four cell mid-sides.
q	Vector Unit Flow	All formats excluding WaterRIDE	Unit flow (m^2/s, 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.
R	Flow Regime	All formats excluding WaterRIDE	Flow regime. The output 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. When using the default cell corner approach, the flow regime is a weighted average of the flow regimes at the four adjoining cell mid-sides, which can therefore be misleading. Using the Map Output Format == SMS HIGH RES option will output the exact flow regime occurring at the cell mid-sides.
RC	Route Category	All formats excluding WaterRIDE	The route category output over time for evacuation routes. The definition and number of categories is based on the values specified within the Cut_Off_Values attribute of the 2d_zshr GIS layer (see Section 9.5.1). The RC values are output as an integer representing the closure category specified by the user. The maximum RC category value is tracked every timestep and output (if tracking maximums is switched on, which is the default).
RFC	Cumulative Rainfall	All formats excluding WaterRIDE	The cumulative rainfall in mm or inches over time when direct rainfall has been applied to the model (refer to Sections 0 and 7.4.3.2). See also the RFR (rainfall rate) map output type below.

Flag	Map Output Data Type	Supported Formats	Description
RFR	Rainfall Rate	All formats excluding WaterRIDE	The rainfall rate in mm/hr or inches/hr over time when direct rainfall has been applied to the model (refer to Sections 0 and 7.4.3.2). See also the RFC (cumulative rainfall) map output type above.
SP	Stream Power	All formats excluding WaterRIDE	Stream Power as given by the equation below where τbed is bed shear stress (see BSS above) and V is velocity. $Stream\ Power = \lvert V \rvert \tau_{bed}$ The Stream Power map output can be misleading at very shallow depths as the BSS formula divides by the depth. The BSS and SP outputs are linearly reduced to zero once the depth is below a threshold (by default, 0.1m). This threshold can be changed using the .tcf command BSS Cutoff Depth.
SS	Sink / Source Flow	All formats excluding WaterRIDE	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).
t	Viscosity Coeff	All formats excluding WaterRIDE	Eddy viscosity coefficient. This is useful for checking the Smagorinsky coefficient values. No maximum and minimum output is available at this stage.
V	Vector Velocity	All formats	Flow velocity. The resulting velocity vector is calculated from the surrounding u and v-points. Note: The maximum and minimum velocities are tracked over time. By default the maximum velocities are tracked over 0.1m depth, below this depth the velocity at maximum water level is used. See the Maximum Velocity Cutoff Depth command for more information.

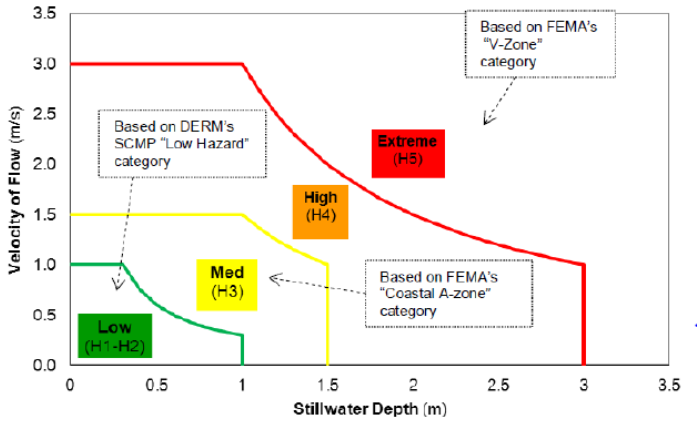
Table 9-11 Map Output Hazard (Z) Types

Flag	Map Output Hazard Type	Supported Formats	Description
Z0	Z0	All formats	Velocity x Depth product
Z1	Z1	All formats	Flood hazard category based on the Australian NSW Floodplain Management Manual (NSWG, 2005). The output is an integer 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 is monitored throughout the simulation and is not necessarily when the maximum water level occurs as with some other output. 

Flag	Map Output Hazard Type	Supported Formats	Description
Z2	Z2	All formats	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 ≤ 2.64: Category 2 (High Hazard) Otherwise Category 1 (Low Hazard) Dry points are assigned Category 0 
Z3	Z3	All formats	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 ≤ 2.64: Category 2 (High Hazard) Otherwise Category 1 (Low Hazard) Dry points are assigned Category 0 

Flag	Map Output Hazard Type	Supported Formats	Description
Z4	Z4	All formats	Flood hazard mapping approach based on the Australian Guidelines (CSIRO, 2000) using the following logic: D ≤ 0.3 AND V ≤ 0.38: Category 1 (Low Hazard) D ≤ 0.6 AND V ≤ 0.8 AND D + 0.64*V ≤ 0.82: Category 2 (Medium Hazard) D ≤ 1.2 AND V ≤ 1.5 AND D + 0.69*V ≤ 1.38: Category 3 (High Hazard) Otherwise Category 4 (Extreme Hazard) Dry points are assigned Category 0 
Z5	Z5	All formats	Under development for a client and subject to change. Not recommended for use unless instructed by the client or modified for own use.
Z6	Z6	All formats	Under development for a client and subject to change. Not recommended for use unless instructed by the client or modified for own use.
Z7	Z7	All formats	Under development for a client and subject to change. Not recommended for use unless instructed by the client or modified for own use.
Z8	Z8	All formats	Under development for a client and subject to change. Not recommended for use unless instructed by the client or modified for own use.

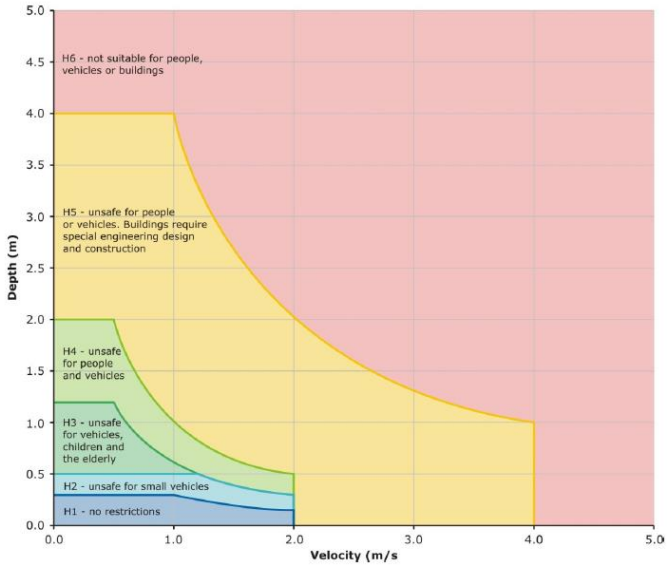
<table>
<tr><th>Flag</th><th>Map Output Hazard Type</th><th>Supported Formats</th><th>Description</th></tr>
<tr><td>Z9</td><td>Z9</td><td>All formats</td><td>Flood hazard mapping approach based on the draft storm tide hydraulic hazard categorisation developed for Moreton Bay Regional Council (GHD, 2011). Z9 output values are 0 (zero) for no hazard and 1 to 5 for H1 to H5 respectively.



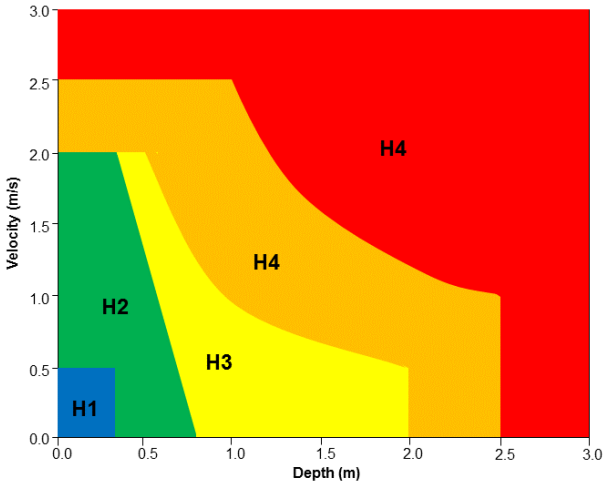
<table>
<tr><th colspan="2">Low Risk to Life and property</th><th colspan="3">High Risk to Life and property</th></tr>
<tr><th>H1</th><th>H2</th><th>H3</th><th>H4</th><th>H5</th></tr>
<tr><th>Insignificant [1]</th><th>Minor [1]</th><th>Moderate [1]</th><th>Major [1]</th><th>Catastrophic [1]</th></tr>
<tr><td>No significant life risk
Property risk only to items which come in direct contact with floodwaters such as building contents</td><td>Low life risk. Able bodied adults can walk safely.
Cars can float and precautions must be followed to keep them out of floodwaters</td><td>Moderate life risk.
Able bodied adults cannot safely walk
Only large vehicles (trucks) can safely travel.</td><td>Major life risk
Light frame buildings (e.g. houses) can fail structurally</td><td>Extreme life risk
Majority of buildings could fail</td></tr>
</table>
</td></tr>
<tr><td>ZH</td><td>Bathymetry</td><td>All formats excluding WaterRIDE</td><td>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 because of variable geometry (VG) boundaries or for morphological modelling. This output is very useful if you are comparing two or more runs that have different topography (eg. before and after scenarios), and you wish to easily view or compare the topography for each scenario within SMS or post-process using TUFLOW_to_GIS (refer to Section 15.2.1).

If the topography in the model does not change over time (ie. no variable Z shapes or morphological changes), the ZH Zpt values are output once, rather than every timestep, thereby not consuming disk space unnecessarily. This feature is only available if using the XMDF format. The ZH map output will appear under a XMDF folder "Fixed".

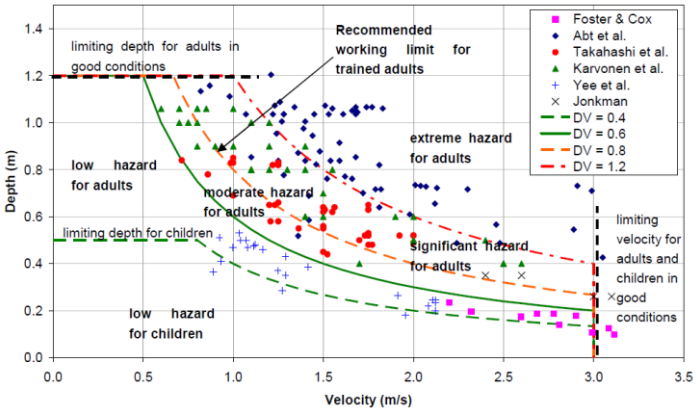
No maximum and minimum output is available at this stage.</td></tr>
</table>

Flag	Map Output Hazard Type	Supported Formats	Description
ZAEM1	ZAEM1	All formats	Flood hazard category as outlined by Australian Emergency Management Institute in 2014. ZAEM1 output values are 0 (zero) for no hazard and 1 to 6 for H1 to H6 respectively. 

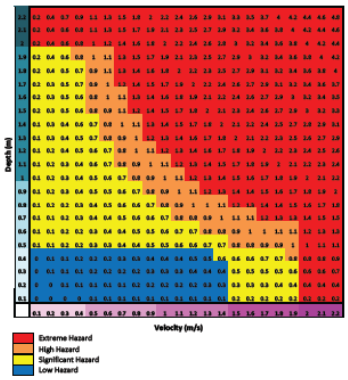
Hazard Classification	Description
H1	Relatively benign flow conditions. No vulnerability constraints.
H2	Unsafe for small vehicles.
H3	Unsafe for all vehicles, children and the elderly.
H4	Unsafe for all people and all vehicles.
H5	Unsafe for all people and all vehicles. Buildings require special engineering design and construction.
H6	Unconditionally dangerous. Not suitable for any type of development or evacuation access. All building types considered vulnerable to failure.

Flag	Map Output Hazard Type	Supported Formats	Description
ZMBRC	ZMBRC	All formats	Flood hazard output used by Moreton Bay Regional Council. Where: V > 2.5 or D > 2.5 or V*D > 2.5: Category 5 (H5) V > 2.0 or D > 2.0 or V*D > 1.0: Category 4 (H4) V > 3.2 - 4D: Category 3 (H3) V > 0.5 or D > 0.3: Category 2 (H2) Otherwise Category 1 (H1) Dry points are assigned Category 0  H1: Hydraulically suitable for parked or moving cars. H2: Hydraulically suitable for parked or moving heavy vehicles and wading by able-bodied adults. H3: Hydraulically suitable for light construction (eg. Timber frame and brick veneer). H4: Hydraulically suitable for heavy construction (eg. steel frame and reinforced concrete). H5: Generally unsuitable
ZMW1	ZMW1	All formats	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

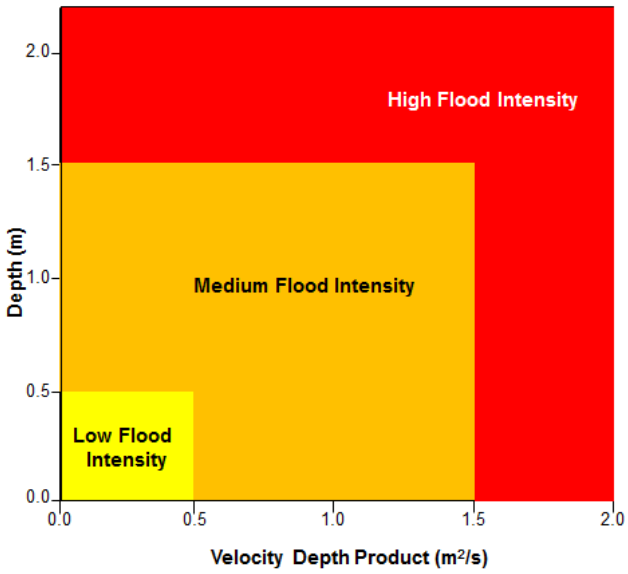
Flag	Map Output Hazard Type	Supported Formats	Description
ZMW2	ZMW2	All formats	Based on Melbourne Water FM&M Technical Specifications to quantify Safety Risk in Roads. The logic used is as follows: D*V < 0.2 and D < 0.2: Category 1 D*V < 0.4 and D < 0.4: Category 2 D*V < 0.6 and D < 0.6: Category 3 D*V < 0.84 and D < 0.84: Category 4 Otherwise Category 5 Dry points are assigned Category 0

<table>
<tr><th>Flag</th><th>Map Output Hazard Type</th><th>Supported Formats</th><th>Description</th></tr>
<tr><td>ZPA</td><td>ZPA</td><td>All formats</td><td>
People Hazard category “Hazard to Adults” based on the Australian Rainfall and Runoff (ARR) Project 10 Stage One report, published by Engineers Australia (Cox *et al*, 2010). The values within the ZPA output are:
0 = Safe (no hazard)
1 = Low Hazard
2 = Moderate Hazard
3 = Significant Hazard
4 = Extreme Hazard
It is possible to specify a cut-off depth/s representing when the Safe category applies by using the .tcf command ZP Hazard Cutoff Depth. Up to three values can be defined which are the cut-off depths for ZPA, ZPC and ZPI respectively.
The Engineers Australia report may be downloaded from http://www.tuflow.com/Download/Publications/2010.04 - ARR Project 10 Stage1 report Final.pdf. The relevant tables and figures are reproduced below:
<table>
<tr><th>DV (m^2s^{-1})</th><th>Infants, small children (H.M ≤ 25) and frail/older persons</th><th>Children (H.M = 25 to 50)</th><th>Adults (H.M > 50)</th></tr>
<tr><td>0</td><td>Safe</td><td>Safe</td><td>Safe</td></tr>
<tr><td>0 – 0.4</td><td rowspan="5">Extreme Hazard; Dangerous to all</td><td>Low Hazard[1]</td><td rowspan="2">Low Hazard[1]</td></tr>
<tr><td>0.4 – 0.6</td><td>Significant Hazard; Dangerous to most</td></tr>
<tr><td>0.6 – 0.8</td><td rowspan="3">Extreme Hazard; Dangerous to all</td><td>Moderate Hazard; Dangerous to some[2]</td></tr>
<tr><td>0.8 – 1.2</td><td>Significant Hazard; Dangerous to most[3]</td></tr>
<tr><td>> 1.2</td><td>Extreme Hazard; Dangerous to all</td></tr>
</table>
[1] Stability uncompromised for persons within laboratory testing program at these flows (to maximum flow depth of 0.5 m for children and 1.2 m for adults and a maximum velocity of 3.0 ms^{-1} at shallow depths).
[2] Working limit for trained safety workers or experienced and well equipped persons (D.V < 0.8 m^2s^{-1})
[3] Upper limit of stability observed during most investigations (D.V > 1.2 m^2s^{-1})


</td></tr>
<tr><td>ZPC</td><td>ZPC</td><td>All formats</td><td>People Hazard category “Hazard to Children” based on the Australian Rainfall and Runoff (ARR) Project 10 Stage One report (Cox *et al*, 2010), published by Engineers Australia. Refer to the description of output values for ZPA above.</td></tr>
</table>

Flag	Map Output Hazard Type	Supported Formats	Description
ZPI	ZPI	All formats	People Hazard category “Hazard to Infants and frail/elderly People” based on the Australian Rainfall and Runoff (ARR) Project 10 Stage One report (Cox *et al*, 2010), published by Engineers Australia. Refer to the description of output values for ZPA above.
ZQRA	ZQRA	All formats	Hazard categories for the Queensland Reconstruction Authority. Refer to http://qldreconstruction.org.au/u/lib/cms2/resilient-floodplains-part2-full.pdf (Queensland Reconstruction Authority, 2011-2012). **Schedule 4 - Flood hazard criteria** **Indicative flood hazard criteria** The following indicative flood hazard criteria have been prepared for use in preparing flood investigations (level 2), and planning evaluations based on latest available engineering guidance. In the absence of other more appropriate flood hazard definitions, the criteria below may be used.  Depth (m) Velocity (m/s) Extreme Hazard High Hazard Significant Hazard Low Hazard **Rules** Depth: Low <0.5, Significant <2, High <2, Extreme 2+ Velocity: Low <1.5, Significant <2, High <2, Extreme 2+ DxV Ratio: Low <0.6, Significant 0.6 to <0.8, High 0.8 to <1.2, Extreme 1.2 + **Rationale** 1. **Low** – self evacuation possible for adults and children, vehicle stability within tolerance for large 4WD 2. **Significant** – working limit for trained safety workers, Vehicle evac unsuitable, Building Code limitation 3. **High** – limit of uncompromised stability for adults (dangerous to most) 4. **Extreme** – in excess of known stability limits **References** 1. ARR Revision Project 10: Appropriate Safety Criteria for People a. Children – Significant Hazard DV ≤ 0.6 & D ≤ 0.5 b. Adult – Moderate Hazard DV ≥ 0.6 c. Working limit for trained safety workers or experienced and well equipped persons DV < 0.8 2. ARR Revision Project 10 State 2 Report: Appropriate Safety Criteria for Vehicles (Draft) a. Large 4WD DV ≤ 0.6 & D ≤ 0.5 3. Dale et al. (2004) Structural flood vulnerability and the Australianisation of Black’s Curves a. Fibro/Tile construction D < 0.5 & V < 2 b. Draft QDC for flood hazard areas for Deemed to Satisfy provisions – V < 1.5 4. BMT WBM (2012) Newcastle City-wide Floodplain Risk management Study and Plan P.81-82 a. Hydraulically suitable for wading by able-bodied adults V < 2 & D < 0.8 b. Hydraulically suitable for light construction (e.g. timber frame and brick veneer) V < 2 and D < 2 5. Jonkman et al. (2008) Methods for the estimation of loss of life due to floods: A literature review and proposal for a new method Natural Hazards P. 364 a. Level of hazard to people can be categorized as low, moderate, significant or extreme.
ZUK0	ZUK0	All formats	The value of the UK Hazard formula (see UK Hazard Formula, UK Hazard Land Use and UK Hazard Debris Factor).
ZUK1	ZUK1	All formats	The UK Hazard category (see UK Hazard Formula, UK Hazard Land Use and UK Hazard Debris Factor).

Flag	Map Output Hazard Type	Supported Formats	Description
ZUSA1	ZUSA1	All formats	The flood intensity category as defined by the flow depth and/or the depth velocity product as shown in the figure and table below. ZUSA1 output values are 0 (zero) for no hazard, 1 for Low Intensity, 2 for Medium and 3 for High. The methodology has been developed by Garcia et al (2003, 2005) and is described in the Flo-2D Mapper Manual 2009 (Refer to http://www.flo-2d.com/)



Definition of Water Flood Intensity			
Flood Intensity	Maximum depth h (m)		Product of max depth h (m) and max velocity v (m^2/s)
High	h > 1.5 m	OR	v h > 1.5 m^2/s
Medium	0.5 m < h < 1.5 m	OR	0.5 m^2/s < v h < 1.5 m^2/s
Low	0.1 m < h < 0.5 m	AND	0.1 m^2/s < v h < 0.5 m^2/s

Flood hazard may then be calculated by relating flood intensity to probability.

9.8 Specialised Outputs

9.8.1 Recording Gauge Data at Receptors (2d_obj, 2d_rec)

Read GIS Objects RECORD GAUGE DATA records the flood level and simulation time at one or more gauge(s) when receptors are first inundated above their trigger inundation levels (eg. floor levels). Read GIS Receptors can be used as an alias to Read GIS Objects. Level output associates a flood level at one or more gauges with the time of first inundation at properties, buildings, or other areas of interest within the modelled extent. When the water level at the property reaches a user-defined trigger inundation level, the gauge height and simulation time is recorded and tagged to the receptor. This is particularly useful for flood warning and forecasting studies where property specific information on the likelihood of a property being inundated for a given gauge height can be generated.

Gauges are defined as a point within a 2d_po GIS layer with type "G_" (see Section 9.3.3 and Table 9-3). The levels from all gauges are recorded at each receptor once inundated.

Receptors must be GIS point and/or polygon objects located in one or more GIS layers nominally prefixed by 2d_obj or 2d_rec. Each object within the layer represents a receptor, property or other object of interest. For information on the attributes of the GIS layer see .tgc Read GIS Objects command. The command also includes options to set the Zpt elevations to the receptor level or the first attribute in the layer (for example, to set the Zpts to the floor level of the buildings), or to alternatively use the existing ZC elevations.

Once simulated, a GIS layer is written to the location as specified using the .tcf command Output Folder, and has a _GDO extension standing for Gauge Data Output. The layer contains point objects (for regions the centroid is used). The attribute data for the layer is described in Table 9-12. Figure 9-4 shows an example of how the output GIS layers may be used to illustrate the flooding of properties in relation to the water level at a gauge.

An example model making use of this feature may be downloaded from this TUFLOW Forum Post.

Table 9-12 2d_GDO_ Gauge Data Output Attributes

GIS Attribute	Description
Trigger_Level	The trigger inundation level at the receptor. The level will be either the first attribute in the GIS layer (typically either the ground level of the property or the floor level of the building), or the lowest ZC value within the property or building polygon if the "USE ZC" option is specified (ie. Read GIS Gauge Output USE ZC).
Time	The simulation time in hours when the water level at the receptor first reaches the Trigger_Level.
Use_ZC	"Y" if the ZC 2D cell elevation was used for the Trigger_Level; "N" if not.
<Gauge_1>	The water level in metres above datum at gauge <Gauge 1> as defined within a 2d_po layer, when the water level first reaches the Trigger_Level at the property/building/object.
<Gauge_N>…	If a second, third, fourth… gauge exists, the water level in metres above datum at each gauge.

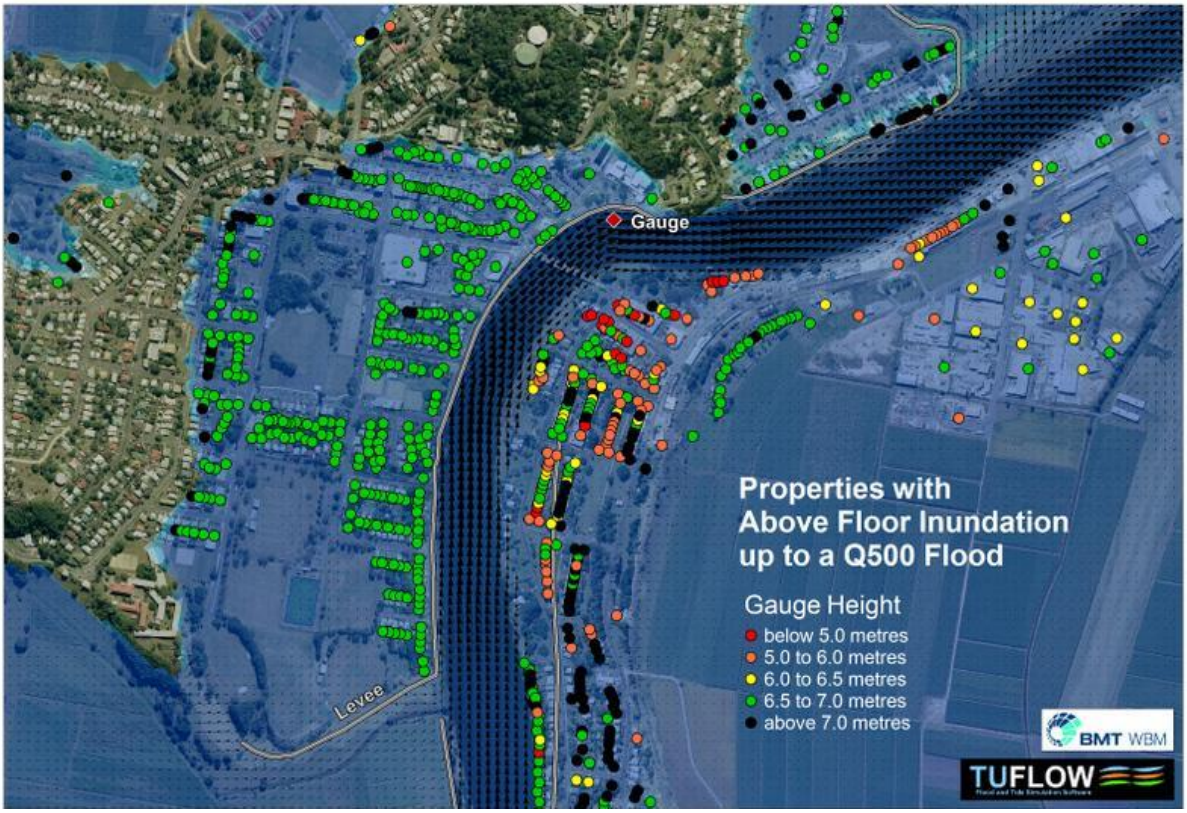


Figure 9-4 Example Use of Gauge Data Output Layer

9.8.2 Evacuation Routes (2d_zshr)

Evacuation routes can be specified to provide help define:

- Evacuation route suitability;
- Warning times;
- Risks;
- Degree of route inundation; and
- Duration of inundation.

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 Shape options. The 2d_zshr layer is the same as a 2d_zsh layer, but with three additional attributes, as shown in Table 9-13.

The commands Set Route Cut Off Values and Set Route Cut Off Type can be used in the .tcf and/or .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.

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 attributes as shown in

Table 9-14.

The _RC.dat output file contains the Route Categories over time and is written when adding RC to the .tcf command Map Output Data Types. This file can be used to view and animate the route category values.

Table 9-13 2D Z-Shape Route (2d_zshr) Attribute Descriptions

No.	Default GIS Attribute Name	Description	Type
Read GIS Z Shape Route Command			
1	Z	Unless the BRIDGE Shape_Options is specified (see below), the Z Shape lines adjust the Zpts as described in Table 6-8 for the same attribute.	Float
2	dZ	Unless the BRIDGE Shape_Options is specified (see below), the Z Shape lines adjust the Zpts as described in Table 6-8 for the same attribute.	Float
3	Shape_Width_or_dMax	Refer to the same attribute in Table 6-8.	Float

No.	Default GIS Attribute Name	Description	Type
4	Shape_Options	**BRIDGE:** Does not use the Z attribute to adjust the Zpts, instead uses it to assign 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_Options of "BRIDGE".	Char(20)
5	Route_Name	Used to label the evacuation 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.	Char(40)
6	Cut_Off_Type	Options are as follows: **Blank:** Uses the value from the latest Set Route Cut Off Type command is applied at that point in the .tgc file. **Depth:** The depth (default). **VxD, Z0** or **Hazard:** The velocity x depth hazard product **V** or **Velocity:** The velocity. More than one 2d_zshr layer may be used if different cut-off types are required.	Char(40)
7	Cut_Off_Values	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 _RC.dat file, and summarised in the _RCP layer. Multiple cut-off values allow for the assessment of different risk levels (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).	Char(80)

Table 9-14 _RCP Output (2d_zshr) Attribute Descriptions

GIS Attribute	Description
Route_Name	The name of the route
Cut_Off_Value	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	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

9.8.3 Calibration Points GIS Layer

Reads an input GIS layer of point objects (.mif format only), and outputs a copy of the GIS layer appended with the peak water level calculated during the simulation as an extra attribute. If the 2D cell on which the point object falls remains dry for the duration of the model simulation, a value of "99999" is reported. The output layer is written to the same location as the 2D map output results.

The input GIS layer does not require any attributes, and if any are entered, these are ignored by TUFLOW. Only point objects within the GIS layer are used with all other object types being ignored.

The calibration points GIS layer is referenced in the .tcf using the command Calibration Points MI File. Currently only .mif/.mid format is supported. If a .shp file is referenced, an ERROR 2451 message will be output.

10 TUFLOW GPU Solver

Chapter Contents

10 TUFLOW GPU Solver 10-1

10.1 Introduction 10-2

10.1.1 Will the Results be the same as TUFLOW Classic? 10-2

10.2 Solution Scheme 10-4

10.2.1 Conservativeness 10-4

10.2.2 Stability and Adaptive Timestepping 10-4

10.3 Getting Started 10-6

10.3.1 Compatible Graphics Cards 10-6

10.3.1.1 Compute Capability 10-9

10.3.2 Updating NVIDIA Drivers 10-9

10.4 Running TUFLOW GPU 10-10

10.4.1 Single and Double Precision 10-10

10.5 TUFLOW GPU .tcf Commands 10-11

10.6 Data Input 10-12

10.6.1 Cell Codes 10-12

10.6.2 Topography 10-12

10.6.3 Bed Resistance 10-12

10.6.4 Initial Water Levels 10-12

10.6.5 Boundary Conditions 10-12

10.6.6 Infiltration 10-13

10.6.7 Virtual Pipe Feature 10-13

10.6.8 Excluded TUFLOW Functionality 10-16

10.7 Data Output 10-17

10.7.1 Time-Series Output (2d_po) 10-17

10.8 Troubleshooting 10-18

10.1 Introduction

TUFLOW's GPU Module is a powerful solver built into the existing TUFLOW software. As its name implies it utilises the substantial parallel computing ability of modern Graphics Processor Units (GPU).

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

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

The front end is still TUFLOW, and very little modification of the model input is needed to utilise the GPU solver. Similarly the output data is still written by TUFLOW and the same output formats and data types are available. TUFLOW GPU was submitted for the 2012 UK Environment Agency 2D Benchmark Tests (see Section 0), where along with TUFLOW and TUFLOW FV, were top performers in terms of consistent results and run times. The scheme has also been benchmarked against theoretical solutions and through comparisons with TUFLOW and TUFLOW FV.

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

10.1.1 Will the Results be the same as TUFLOW Classic?

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

However, the primary reason is that the two solution schemes, whilst solving the same equations, are different. TUFLOW Classic is a second order semi-implicit (matrix) finite difference solution, whilst GPU is an explicit first order spatial finite volume solution. For tests such as those in the 2012 UK Environment Agency 2D Benchmarking, Classic and GPU produce similar results, especially when compared to the range in results produced by all the other schemes. The UK Environment Agency tests, however, are predominantly over shallow flow, where bed friction tends to dominate. For deep, fast flowing water the bed resistance term is less dominating and the representation of other terms such as eddy viscosity, the degree of numerical diffusion, whether 1st order or 2nd order, and other influences will have a more influential effect on the results. Observations to date between Classic and GPU are that similar levels result for shallow, friction dominated areas, while for deep, fast flowing models with

river bends and other flow complexities TUFLOW GPU tends to produce higher water levels for the same Manning's values (for example, 0.2m higher in flow that is 10m deep and 3m/s velocity).

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

10.2 Solution Scheme

10.2.1 Conservativeness

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

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

10.2.2 Stability and Adaptive Timestepping

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

- The volume lost from a cell in any given timestep cannot leave the cell with negative volume. If we express the Courant number as $Nu = \frac{udt}{dx}$, then keeping $Nu < 0.25$ guarantees that a cell cannot be depleted to zero volume in a single timestep. In practice we have found $Nu < 1.0$ to be a suitable upper limit for Nu since cells that have fluid leaving on all boundaries do not usually have the maximum velocity in the model.
- The Shallow Water Equations admit harmonic solutions (ie. shallow water waves with speed $c = \sqrt{gh}$). This gives rise to a non-dimensional wave speed number $Nc = \frac{cdt}{dx}$. For the 4th order solver (the default and recommended solution), we have found the solution to remain stable for $Nc \leq 2.0$. For Build 2013-12-AB, the limiting default for Nc was reduced to 1.6 due to the occasional instability in very deep fast flowing water.
- Momentum diffusion. The sub-grid scale eddy viscosity term causes diffusion of momentum. This gives rise to non-dimensional diffusion number $Nd = \frac{\nu dt}{dx^2}$. Again for the 4th order solver we have found the solution to remain stable for $Nd \leq 0.3$.

After each timestep the solver checks for model instability (NaNs or sudden increases in *Nu, Nc, Nd*) and if necessary will reduce the timestep and repeat the step. If adaptive timestepping is off the timestep will not be increased again, and may be decreased again if further instabilities are encountered. If adaptive timestepping is on the solver controls the timestep to maintain *Nu, Nc, Nd* at or just under the limits quoted above of 1.0, 1.6, and 0.3 respectively. If desired the user may adjust these limiting values using Maximum Courant Number. If NaNs are encountered this overall scalar will be reduced slightly and the timestep repeated; the overall scalar will not be increased again, but may be decreased again if further instabilities are encountered. If a sudden increase in *Nu, Nc, Nd* is encountered the timestep will be temporarily decreased and the timestep repeated. The timestep will then be allowed to gradually

ramp up again until one of *Nu, Nc, Nd* reaches its control limit. The timestep also can't exceed the plot output or map output time intervals.

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

10.3 Getting Started

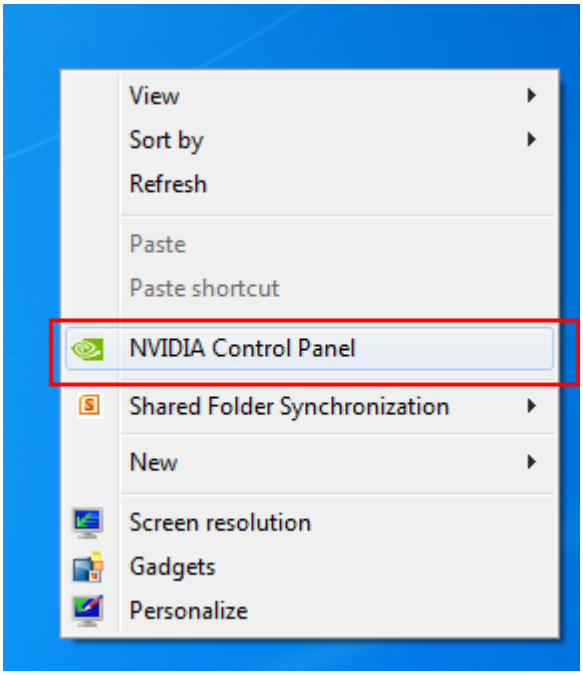
10.3.1 Compatible Graphics Cards

TUFLOW GPU requires an NVIDIA CUDA enabled GPU. A list of CUDA enabled GPUs can be found on the following website: http://developer.nvidia.com/cuda-gpus.

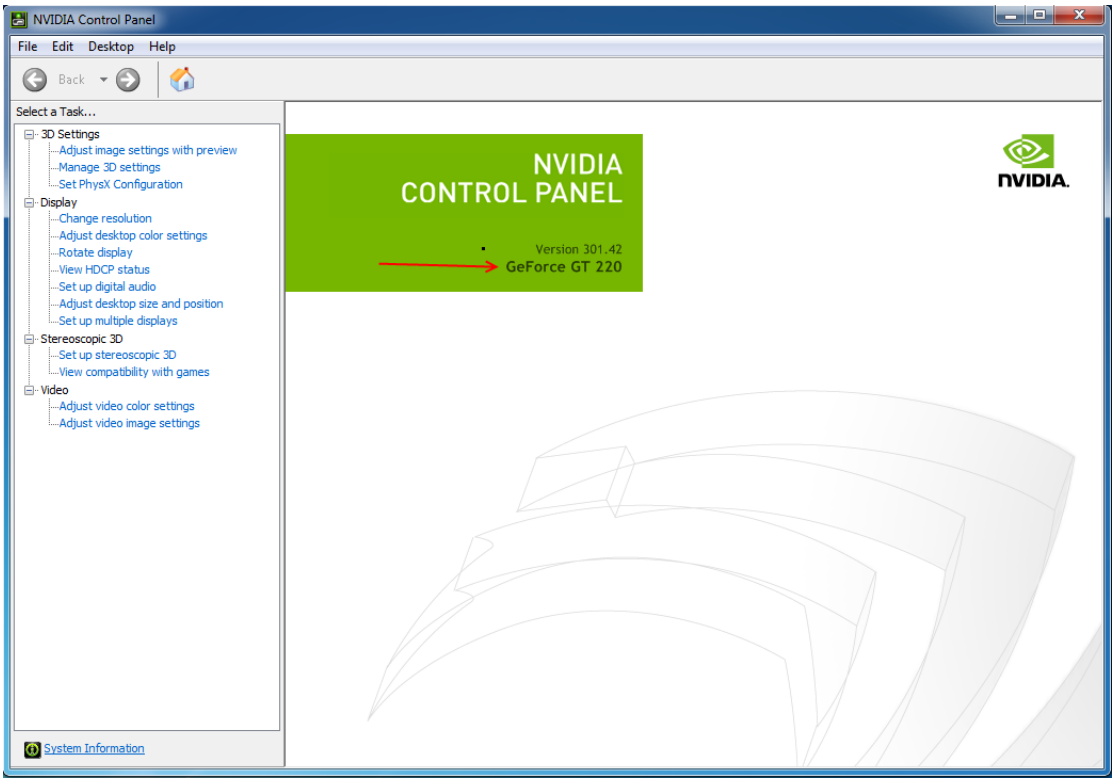
To check if your computer has an NVIDA GPU and if it is CUDA enabled:

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

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



Screenshot: Accessing NVIDIA Control Panel from the desktop



Screenshot: NVIDIA GPU Model

CUDA-Enabled GeForce Products

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

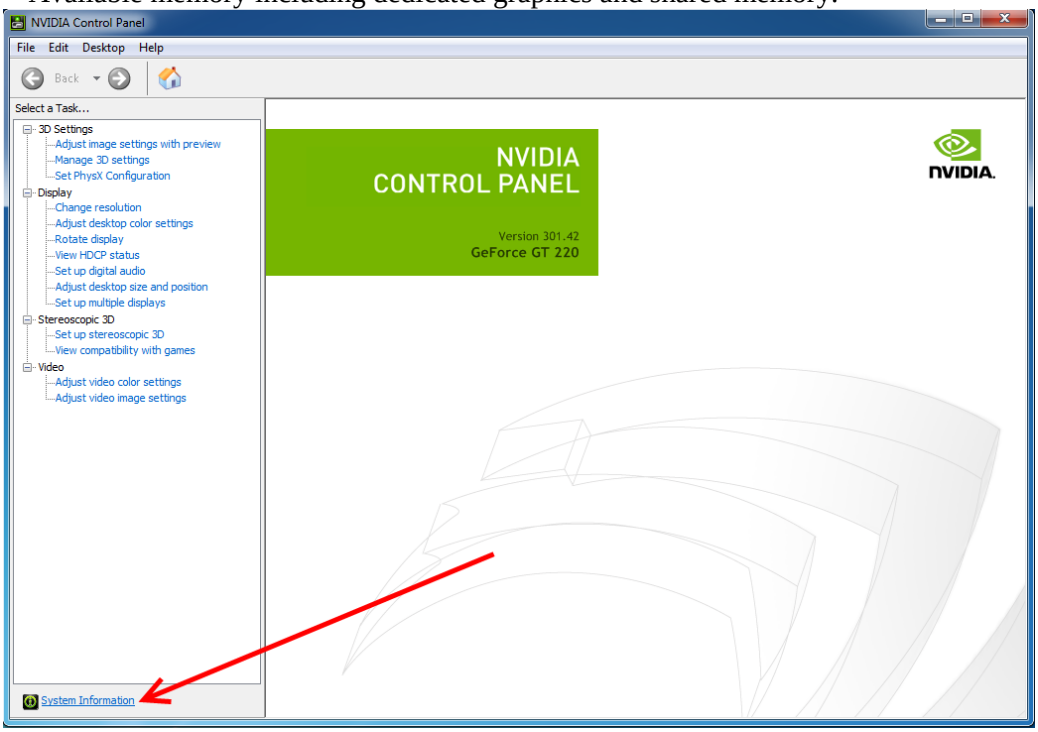
GeForce Desktop Products		GeForce Notebook Products	
GPU	Compute Capability	GPU	Compute Capability
GeForce GTX 690	3.0	GeForce GTX 680M	3.0
GeForce GTX 680	3.0	GeForce GTX 675M	2.1
GeForce GTX 670	3.0	GeForce GTX 670M	2.1
GeForce GTX 560 Ti	2.1	GeForce GTX 660M	3.0
GeForce GTX 550 Ti	2.1	GeForce GT 650M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

Screenshot: Check the Website for your NVIDIA Card

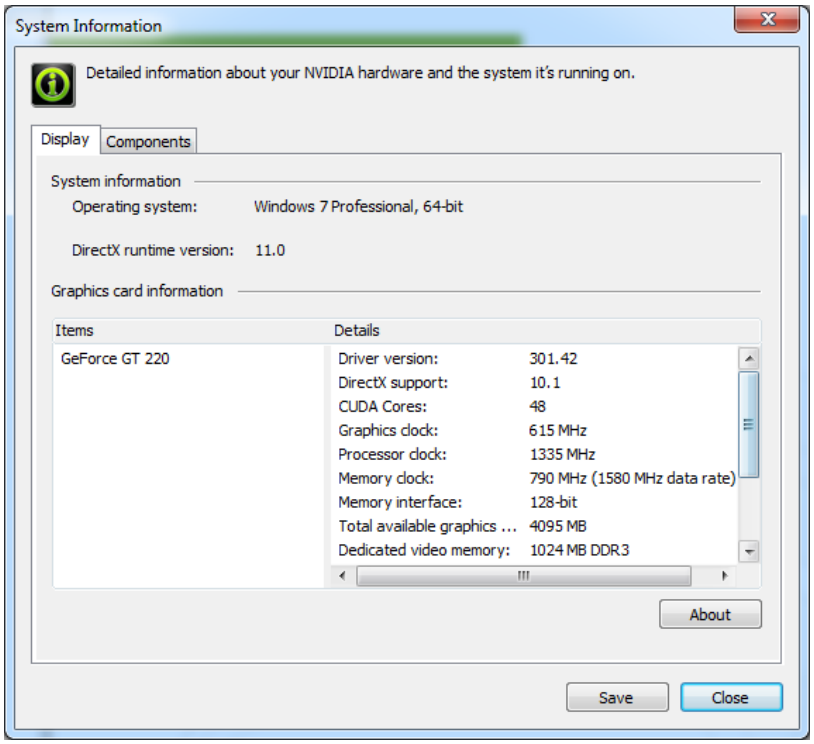
More information on the card can be found in the "System Information" section, which is accessed from the NVIDIA Control Panel. The system information contains more details on the following:

- The number of CUDA cores;
- Frequency of the graphics, processors and memory; and

- Available memory including dedicated graphics and shared memory.



Screenshot: Accessing System Information from NVIDIA Control Panel



Screenshot: NVIDA System Information

10.3.1.1 Compute Capability

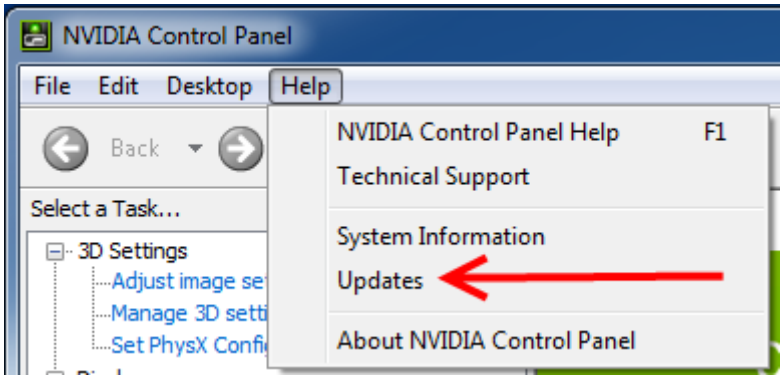
On the NVIDA website each CUDA enabled graphics card has a "Compute Capability" listed. For cards with a compute capability of 1.2 or less, only the single precision version of the GPU Module can be utilised. **However, benchmarking has indicated that the double precision version is NOT required and that for all GPU simulations TUFLOW_iSP exes should be used. Refer to Section** 10.4.1**.**

10.3.2 Updating NVIDIA Drivers

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

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

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



Screenshot: Accessing Driver Updates from NVIDIA Control Panel

10.4 Running TUFLOW GPU

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

To simulate the model via the GPU solver the command GPU Solver == ON is required in the .tcf. The TUFLOW GPU simulation is then started in the same manner as a standard TUFLOW simulation. This can be done via a batch file, right click functionality in Windows or from a text editor. Refer to Section 11.5.2 for instructions in creating a batch file and the TUFLOW Wiki for all other options.

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

A list of available control file commands is given Section 10.5.

10.4.1 Single and Double Precision

TUFLOW GPU is available in both single precision and double precision. If the simulation is started with the single precision version of TUFLOW, the GPU solver will utilise a single precision version. If the simulation is started with the double precision version of TUFLOW, error message ERROR 2420 will be output by default stating that the single precision version should be used. This default can be overwritten by specifying GPU DP Check == OFF.

The calculation method in the GPU uses depth due to its explicit nature, unlike TUFLOW Classic that uses water level due to its implicit scheme. This means that precision issues associated with applying a very small rainfall to a high water level are not applicable.

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

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

10.5 TUFLOW GPU .tcf Commands

The following .tcf commands (apart from Maximum Courant Number and Timestep) are specific to the TUFLOW GPU solver:

GPU Device IDs	Controls the GPU device or devices to be used for the simulation if multiple CUDA enabled GPU cards are available in the computer or on the GPU itself.
GPU DP Check	The default setting causes TUFLOW GPU to output an error message advising that it is recommended to use the single precision version of the GPU solver.
GPU Solver	This command must be set to ON to initialise the GPU solver in TUFLOW. If this command is not present or set to OFF the standard TUFLOW Classic CPU solution is used.
GPU Temporal Scheme	Sets the order of the temporal solution. The default is the recommended 4th order temporal solution, therefore this command is usually not specified. **We recommend the use of the 4th order temporal scheme as it is unconditionally stable with adaptive timestepping turned on, and has been found to give accurate results. Lower order schemes save a little on memory requirements, but are more prone to instability and in some cases unreliable results.**
Maximum Courant Number	This option switches on the adaptive timestepping if set greater than zero. For the 4th order time solver a value of 1.0 (the default) is recommended. The recommendation is not to specify this command and use the default settings. If the user wishes to use reduced timestep control numbers then the scaling factor may be entered here. This is discussed in Section 10.2.2, and is useful for checking timestep convergence.
Timestep	If adaptive timestepping is off, sets the fixed timestep (see Section 10.2.2). The timestep will always be much smaller than TUFLOW Classic's timesteps as the scheme is explicit (TUFLOW uses an implicit scheme). As a general rule of thumb specify a timestep that is around one tenth of the TUFLOW timestep you would use. If adaptive timestepping is active (the default), the timestep value is only used for the very first step. To allow the same command to be used for either a TUFLOW Classic or a GPU simulation the timestep value is divided by 10 for the initial GPU timestep, therefore, enter a timestep value similar to that that you would use for TUFLOW Classic.

10.6 Data Input

10.6.1 Cell Codes

Active (Code 1) and inactive cells (Code 0) are assigned cell codes in the same manner as for TUFLOW. Refer to Section 6.7.

10.6.2 Topography

The GPU solver has only one elevation per cell (ZC). Therefore, thin Z lines are automatically treated as thick Z lines if using the GPU solver (see Section 6.8.3 and also Thin Line as Thick). TUFLOW's topographic manipulation commands are available for use on GPU models keeping in mind that only functionality that modifies the ZC values is applicable. As with all models, it is strongly recommended that a check of the final topography is as expected, has been carried out. Flow constrictions (FC) and layered flow constrictions (LFC) are not as yet utilised by the GPU solver and should not be used. As of Build 2013-12-AB, form loss coefficients (FLC), storage reduction factors (SRF) and cell flow width factors (CWF) are supported using the tgc command Read GIS <option>.

10.6.3 Bed Resistance

Manning's n values are specified in the same manner as for TUFLOW. All materials and Manning's n functionality is available for use with the GPU solver, including depth-varying Manning's n curves. Refer to Section 6.9.1.

10.6.4 Initial Water Levels

Initial water levels (IWL) can be applied in the same manner as for TUFLOW. Refer to Section 7.6.1.

10.6.5 Boundary Conditions

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

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

Table 10-1 TUFLOW GPU Boundary Types

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

Source Inflows	2d_sa: Read GIS SA 2d_rf: Read GIS RF 2d_bc (ST): Read GIS BC Global Rainfall commands: Global Rainfall BC Global Rainfall Area Factor Global Rainfall Continuing Loss Global Rainfall Initial Loss	All SAs are treated as either SA ALL, or if the Read GIS Streams command has been specified SA's are applied as SA STREAM ONLY by default. TUFLOW Classic's ability to distribute the flow over the wet cells and weight the flow according to depth are being investigated for implementation in the GPU solver.

To keep the memory footprint minimal, there is a limit on the number of different source inflows each cell can have. Any number of source inflows can be specified for the model, but a single cell can only have up to four (4) different source inflows. Each inflow can also be weighted, for example, to distribute rainfall hyetographs differently to each cell, through TUFLOW's Read GIS RF (2d_rf) f1 and f2 attributes (refer to Section 7.4.3).

Note that the IL and CL values in the materials file are NOT yet used by the TUFLOW GPU solver for direct rainfall modelling, primarily to keep the memory footprint small (GPUs have small memory capacity compared with CPUs). An IL/CL loss can be modelled using the IL/CL infiltration option (refer to 10.6.6), which is the preferred approach. If not applying infiltration the rainfall applied must therefore be excess rainfall.

10.6.6 Infiltration

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

All TUFLOW Classic's infiltration inputs and outputs are valid and can be used with the GPU solver. Refer to Section 6.10 for further information.

10.6.7 Virtual Pipe Feature

The virtual pipe feature can be used to simulate flows in drainage networks, without full linking to a 1D model (which is under development). A depth flow relationship is applied at the inlet locations and the flow is transferred to the outlet locations. The GIS input is a series of points with the attributes described in Table 10-2.

The flows at the inlet and outlet can be limited, if the outlet becomes limited, the inlet capacity is reduced and pits can surcharge. The order in which pits are limited / surcharge is controlled by the surcharge index attribute in the GIS layer.

The inlet capacities curves are specified in a Pit Inlet Database (see Section 0) as per a "Q" type pit.

The flows and water levels for each inlet and outlet are output into .csv file and also the _TS GIS layers. Note that the downstream pit level is not known nor is needed for the calculations as the pit is always

assumed to have no downstream influence, therefore, the downstream levels are output as 10 m below the upstream level.

Table 10-2 1d_pit (Virtual Pipe) Attributes

No.	Default GIS Attribute Name	Description	Type
Read GIS Pits Command			
1	ID	Unique ID for pit inlet or outlet.	Char (12)
2	Type	"I" (Inlet). Inlet pits are not connected to a 1d_nwk pipe system, but can optionally be linked to an Outlet pit. "O" (Outlet). Outlet location for one or more Inlet ("I") pits that have the same Network_ID as the Outlet pit. More than one Outlet pit can exist for the same Network_ID. Blank or "D" (Dynamic). Reserved to act as a standard pit connected to a pipe network system in future releases. Do not use this option for the virtual pipe network feature.	Char (4)
3	Network_ID	Network ID (as an integer) used to determine which outlet(s) the flow into the inlet pits discharges from. Separate or independent networks should have a different network ID.	Integer
4	Inlet_Type	For an inlet pit, the name of a pit inlet type in the Pit Inlet Database (see Section 0). Not used for outlet pits.	Char (32)
5	Sur_Index	In the event that the total flow into a network's inlet pit(s) exceeds the maximum outflow capacity of the outlet pit(s) (see Qmax below), one or more of the inlet pit flow rates must be modified. The surcharge index is used to determine how inlet pit flows are adjusted. The inlet pits with the lowest surcharge index are given priority. When the total inflow of all inlet pits exceeds the total capacity of all output pit(s) for a network, the inflow into the inlet pit with the lowest surcharge index is reduced. If this pit's inflow is reduced to zero, this pit is permitted to become an outlet with flow rate up to but not exceeding its maximum outflow (see QMax below). If the total inflow of all remaining inlets exceeds the maximum outflow plus the maximum outflow of this inlet, then the inlet with the next highest surcharge index is adjusted using the same methodology. To prevent an inlet pit from surcharging, set QMax to zero. For example, the pit invert level could be used for the surcharge index, in which case the lowest pits will surcharge first. However, if you have reliable data for the order in which they tend to surcharge in reality then this can be used. The virtual pipe network is flow conserving, but not necessarily energy conserving. The highest pit	Float

No.	Default GIS Attribute Name	Description	Type
		inlet could be incorrectly specified with the lowest surcharge index in which case water would be flowing uphill!	
6	QMax	For outlet pits the maximum flow that the pit can discharge back into the 2D domain. For inlet pits the maximum surcharge capacity or flow back on to the 2D domain. To prevent an inlet pit from surcharging enter a QMax of 0.0 (zero). If the total flow capacity (sum of QMax values for outlet pits on the same network) of the outlet(s) is reached, the inflow to inlet pits is restricted and/or surcharged at the inlets using the surcharge index value above.	Float
7	Width	The width of the channels is used to determine the number of 2D cells to connect the inlet / outlet to.	Float
8	Conn_2D	Used to control the connection type. For virtual pipe pits this field must be blank or set to "SX". If set to blank, this will be treated as a SX connection.	Char (8)
9	Conn_No	Overwrite the automatically determined number of 2D cells, see Width attribute above. • 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.	Integer
10	pBlockage	The percentage blockage (%) of the inlets and outlets. Reduces the capacity by the specified amount.	Float

10.6.8 Excluded TUFLOW Functionality

A range of TUFLOW's features are not supported by the GPU solver. Flow constrictions (FC) and layered flow constrictions (LFC), weir factors (WrF), evacuation routes and other functionality are not yet incorporated. If you are unsure, please email support@tuflow.com.

10.7 Data Output

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

For large models (>1,000,000 cells), using SMS or other post-processors maybe slow, and GIS outputs are often the only way to manage the outputs from large GPU models. At present the GRID map output formats (either .asc or .flt) is the most effective way of handling large models, especially the .flt binary format. Other formats should be trialled as to whether practical or not. Map Output options (Section 13.4) and Output Zone (Section 9.4.3) features are all available with the GPU solver.

Note that the maximum hazard from the GPU solver is not tracked every timestep as yet to minimise GPU memory usage, however, this feature is being built in as an option for the next major release. The maximum Hazard values are presently based on the maximum depth and maximum velocity result.

10.7.1 Time-Series Output (2d_po)

2D plot output (PO) can be extracted by the GPU during a simulation for water levels (H_) and flows (Q_) in the same manner as for TUFLOW. Water levels should be point objects and flows polyline objects. Refer to Section 9.3.3.

10.8 Troubleshooting

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

```
TUFLOW GPU: Interrogating CUDA enabled GPUs …
TUFLOW GPU: Error: Non-CUDA Success Code returned
```

Please try the following steps:

1. Check the compatibility of your GPU card and whether the latest drivers are installed (see instructions in Section 10.3.1).
2. Test with a user account that has administrator privileges as these may be required for running computations on the GPU.
3. If multiple monitors are running from the video card, try running with only a single monitor.

If the above steps fail to get the simulation to run, please email the NVIDIA system information (see figure above) and TUFLOW log file (.tlf) to support@tuflow.com.

11 Managing and Starting Simulations

Chapter Contents

11 Managing and Starting Simulations 11-1

11.1 Introduction 11-2

11.2 File Naming 11-3

11.3 Simulation Management 11-5

11.3.1 Events 11-5

11.3.2 Scenarios 11-9

11.3.3 Variables 11-12

11.4 TUFLOW Executable Download 11-14

11.4.1 Overview and Where to Install 11-14

11.4.2 Single and Double Precision 11-16

11.4.3 Using TUFLOW with Flood Modeller / XP-SWMM, or from SMS 11-17

11.4.4 Customising TUFLOW using TUFLOW_USER_DEFINED.dll 11-18

11.5 Running Simulations 11-19

11.5.1 Dongle Types and Setup 11-19

11.5.1.1 Protocols for Accessing Dongles 11-20

11.5.1.2 "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File 11-20

11.5.1.3 Dongle Failure during a Simulation 11-21

11.5.2 Starting a Simulation 11-21

11.5.2.1 Batch File Example and Run Options (Switches) 11-21

11.5.2.2 Advanced Batch Files 11-26

11.5.3 Windows Priority Levels 11-27

11.1 Introduction

This chapter of the Manual provides guidance on model naming conventions and how the multiple Events and Scenarios functionality may be used to simulate multiple scenarios from a single set of control files.

The latter part of this chapter discusses the four types of TUFLOW.exe files that are available as part of each TUFLOW build, as well as different TUFLOW dongle types and how to start a simulation.

11.2 File Naming

Each hydraulic modelling study can easily generate hundreds of model input files in addition to a large number of check and result files. Devising a sound naming convention as part of the model build process is key to creating a model that is easily understood logically, easier to error check, and provides traceability for quality control purposes.

The examples below are presented as guidance only. They demonstrate the progression of a simple model naming convention to a more complex version that incorporates different flood events and scenarios. The examples will focus on the name of the .tcf control file as this determines the prefix assigned to both the 1D and 2D check and result files.

Some general guidance is to avoid long filenames and use acronyms where possible, for a project of the Brisbane River the acronym BR is preferable to the full name, BR_Exg_100yr_12hr_001.tcf instead of Brisbane_River_Existing_100year_12hour_001.tcf.

Example 1

```
MODEL_001.tcf
```

In its simplest form, the names of the majority of TUFLOW models consist of a few characters denoting the study name and a version number. The characters denoting the study name are typically included in all input files to specify that the files are unique or created for this study. The numbering is used to denote different versions of the model, where each time a change is made and the model re-simulated, the version number is incremented. Use of a model version numbering system ensures it is clear which model input files generated which model output files. This is particularly important when troubleshooting the model.

Example 2

```
MODEL_0100F_001.tcf
MODEL_1000F_001.tcf
```

Most hydraulic modelling studies require the simulation of more than one flood event. In these cases, it is preferable to include the name of the flood event within the model name rather than simply incrementing the model version number. Note that the same number of characters is retained for the 100 year and 1000 year events and the use of ‘F’ to denote a fluvial flood event. Retaining the same number of characters will ensure that when the files are viewed in Explorer, they are presented in ascending order. The model version number is the same in this example and tells the user that the same version of the model has been simulated for two different flood events.

Refer to Section 11.3.1 which presents a method to model numerous flood events using a single .tcf control file.

Example 3

```
MODEL_0100F_EXG_001.tcf
MODEL_0100F_PRP_001.tcf
```

Many hydraulic modelling studies require the simulation of multiple scenarios, such as pre- and post-development scenarios or sensitivity testing where one or more model parameters are varied. Incorporating the name of the modelled scenario into the .tcf easily differentiates the output files associated with each scenario. The characters used to denote the scenario are typically also used for model input files specific to the scenario. For example, the post-development scenario 'PRP' may involve the raising of defences, hence the GIS layer used might be named '2d_zsh_MODEL_PRP_defence_001.MIF'. The presence of this layer in a model simulation of the pre-development scenario 'EXG', will immediately highlight to the user that a mistake has been made.

Refer to Section 11.3.2 which presents a method to model one or more scenarios using a single .tcf control file.

11.3 Simulation Management

TUFLOW incorporates 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.

The following section of the manual discusses the available options.

11.3.1 Events

Most hydraulic modelling studies require the simulation of different probability events. For example, a flood study may have to consider the 2, 5, 10, 20, 50, 100, 200 and 500 year Average Recurrence Interval events, and one or two extreme floods. Each or most of these may have to be simulated using a range of 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. The ability to simulate multiple events using a single set of control files not only makes management of the model easier, it also ensures consistency between the simulations and better quality control.

Multiple events are set up through the creation of 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).

The .tef file is referenced in the .tcf file using the command Event File. 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. Unlimited variations for each type is possible. To automatically insert the name of the event(s) into the output filenames so as to make the output names unique, place "~e~" or "~e<X>~" where <X> is from 1 to 9 into the .tcf. For example,

```
MODEL_~e1~_~e2~_EXG_001.tcf
```

Can be configured to run any number of e1 and e2 scenarios.

- e1=0100F and e2=12h: `MODEL_0100F_12h_EXG_001.tcf`
- e1=0100F and e2=24h: `MODEL_0100F_24h_EXG_001.tcf`
- e1=1000F and e2=12h: `MODEL_1000F_12h_EXG_001.tcf`
- e1=1000F and e2=24h: `MODEL_1000F_24h_EXG_001.tcf`

Two options are available to run the multiple events from the same .tcf file:

1. Using a batch file (.bat): Specifying the new –e or –eX options (see Table 11-2) to run the multiple events using the same .tcf file – refer to the examples below.

2. Using Model Events: Change the events manually before each simulation within the .tcf file. Note that if using the –e option, a .bat file –e events will override any events defined using Model Events which is defined within the .tcf.

The examples below show two approaches for running simulations for a 20 and 50 year flood and 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. 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 in the examples below.

Note: The location of Event File within the .tcf file and the location of the commands within the .tef file are important. If, for example 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. In the examples below, the .ecf Output Folder command is used to set different folders for the 1D results depending on the event.

Events Example 1:

Within the 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 name, for example "Nile_~e~.tcf", and add the following line:

```
Event File == Nile_Events.tef
```

The "Nile_Events.tef" file contains 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
```

Multiple events can be simulated from the one .tcf file using a batch file. Refer to Section 11.5.2 for details how to create a batch file.

```
set TUFLOWEXE=C:\Program Files\TUFLOW\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
```

Events Example 2:

Within the 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 "~e1~" and "~e2" in the .tcf file, for example "Nile_~e1~_~e2~.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 a .bat file may be used (see Section 11.5.2.1):

```
set TUFLOWEXE=C:\Program Files\TUFLOW\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
```

If Event logic blocks are also permitted in the same manner as the If Scenario command. Refer to 11.3.2 for further information.

Events are also automatically defined as variables and can be assigned as the output folder name. See Section 11.3.3 for information on variables.

11.3.2 Scenarios

The If Scenario feature 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.

In Example 1 below, an If Scenario logic block has been inserted into a .tgc file:

Scenarios Example 1:

```
!_____________________________________________________________________________
! 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 in a batch file, 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 "~s~" or "~s1~" 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.
   (ii) If opA was not specified as a scenario, this layer would be ignored. For example, as would be required if modelling the existing scenario.

Example 1 can be extended to include an Option B, where Option B is a modified version of Option A as shown in Example 2a below:

Scenarios Example 2a:

```
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".

However, if Option B was not a modification of Option A, but purely a change to the existing case the commands could be written as shown in Example 2b:

Scenarios Example 2b:

```
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
```

Alternatively, 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.

If the Existing, Option A and Option B scenarios all had their own layer of materials for the whole model the commands could be laid out as shown in Example 2c 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.

Scenarios Example 2c:

```
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\TUFLOW_iSP_w64.exe
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b
%RUN% -b -s exg Nile_~s~.tcf
%RUN% -b -s opA Nile_~s~.tcf
%RUN% -b -s opB Nile_~s~.tcf
```

The If Scenario command can be nested up to 10 levels. The extract below from a .tcf file of a complex model is shown below. The model is a combination of models that can be simulated at different resolutions and in different configurations depending on the simulations objective. Out of interest, the same .tcf file is also used to simulate all historical and design flood events.

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

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

If Scenario == CAS
   ! Cell size dependent commands
   If Scenario == 20m  ### 20m grid Domain
      Start 2D domain == CAS_20m
         Geometry Control File == ..\model_CAS\CAS_20m_001.tgc
         Timestep == 7.5
      End 2D Domain
   Else If Scenario == 60m  ### 60m grid Domain
      Start 2D domain == CAS_60m
         Geometry Control File == ..\model_CAS\CAS_60m_001.tgc
         Timestep == 20
      End 2D Domain
   Else
      Pause == Should not be here - invalid CAS cell size scenario specified.
   End If
   ! Cell size independent commands
```

```
   BC Control File == ..\model_CAS\CAS_001.tbc
   Read GIS IWL == ..\model_CAS\2D_iwl\2D_iwl_CAS_001.mif
   ESTRY Control File == ..\model_CAS\CAS_001.ecf
End If
```

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

11.3.3 Variables

The Set Variable command can be used to set different values for the same variable. This command operates at a higher level than If Scenario or If Event logic blocks so that variables can differ within different scenarios/events. For example, in the case below the variable "2D_CELL_SIZE" is used to set both the grid output resolution and the cell size of the model.

In the .tcf file we have:

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

And in the .tgc file we have:

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

The variable "2D_CELL_SIZE" centralises all of the scenario specific commands in the one location. For example, the timestep, log interval and cell size are all dependent on the scenario.

```
If Scenario == 2m
   Set Variable 2D_CELL_SIZE == 2
   Set Variable 2D_TIMESTEP == 1
   Set Variable Log_Interval == 60
Else If Scenario == 5m
   Set Variable 2D_CELL_SIZE == 5
   Set Variable 2D_TIMESTEP == 2.5
   Set Variable Log_Interval == 120
End If
```

In the .tgc we have the cell size command and in the .tcf we may have the following commands:

```
Grid Output Cell Size == <<2D_CELL_SIZE>>
Timestep == <<2D_TIMESTEP>>
Screen/Log Display Interval == <<Log_Interval>>
```

The use of variables allows the scenario specific commands to occur in a single logic block rather than having multiple If…Else If.. End If blocks of commands. This can optionally be moved into a Read File command.

```
Read File == Cell_Size_Commands_001.trd
```

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

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

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

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

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

If Event ~e1~ is set to "Q100", the output will be directed to ..\results\Q100_OpA.

11.4 TUFLOW Executable Download

11.4.1 Overview and Where to Install

The TUFLOW release consists of four different versions of the executable as follows:

- TUFLOW_iSP_w32.exe
- TUFLOW_iSP_w64.exe
- TUFLOW_iDP_w32.exe
- TUFLOW_iDP_w64.exe

The naming convention for each executable is explained in Table 11-1. 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. 2013-12-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 “2013-12-AE-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.

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, precision (SP or DP), or platform (32 or 64 bit). This can be very useful for ensuring a consistent TUFLOW build/release/version is used.

It is recommended that a folder structure such as shown below is used, noting that the w32 and w64 version will need to be in separate folders. 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 batch files, text editors and GIS packages which are used to start TUFLOW simulations will access the latest build, previous versions can still be used if required.

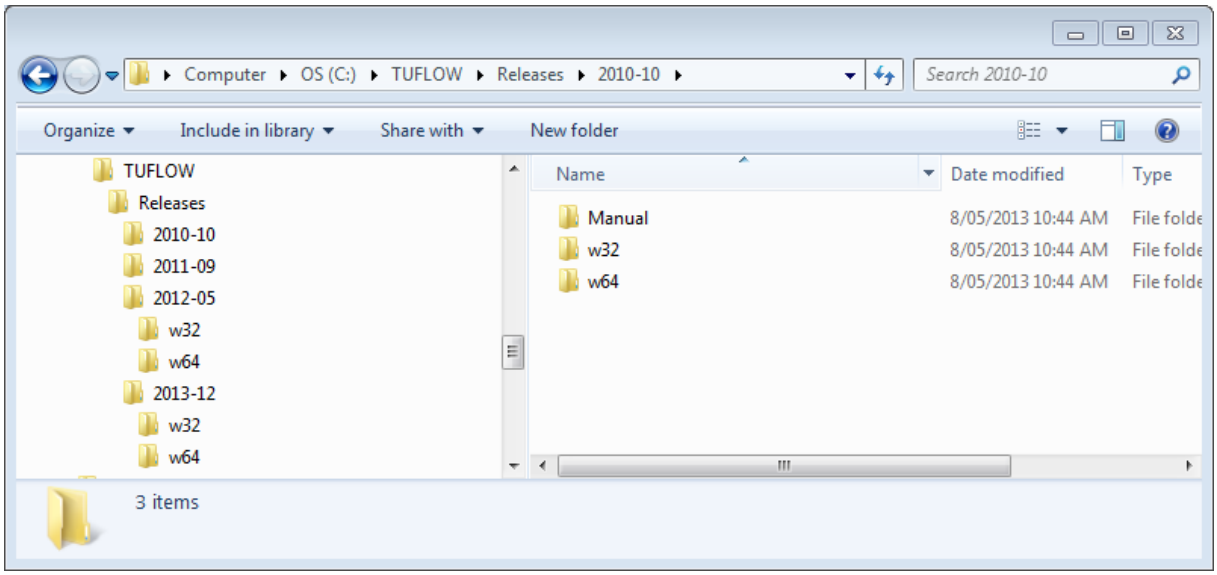


Table 11-1 TUFLOW Versions (iSP, iDP, w32, w64)

Version	Description
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. See Section 11.4.2 for discussion on precision.
Double Precision (iDP)	Compiled using the Intel Fortran Compiler Version 11 using double precision (8-byte) real numbers that typically have around 16 significant figures. See Section 11.4.2 for discussion on precision. 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. For example, a single precision version of TUFLOW will generate an error if used to start the simulation if the command below is included in the .tcf: `Model Precision == Double`
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. 32 bit programs are unable to access as much memory (RAM) as 64 bit versions of TUFLOW. For large models TUFLOW may generate an error that it is unable to allocate enough memory. Typically when the memory requirement is ~1.5GB per simulation the 32 bit version of TUFLOW will not be able to allocate enough RAM.
Windows 64-bit (w64)	Compiled as a 64-bit process. w64 TUFLOW builds can only be run on Windows 64-bit platforms.

Version	Description
	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. **Note that the 64-bit versions of TUFLOW may only be run using a WIBU dongle** (refer to Section 11.5.1).

Running TUFLOW is carried out by initiating the TUFLOW_X.exe file using one of the approaches discussed in Section 11.5.2. The system .dll files are required for the following purposes

- TUFLOW_LINK_X.dll allows other schemes such as Flood Modeller and XP-SWMM to dynamically link with TUFLOW.
- TUFLOW_USER_DEFINED_X.dll allows users to customise TUFLOW to suit their purposes (see Section 11.4.4).
- 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.

11.4.2 Single and Double Precision

When storing floating point values on a computer, a certain number of bytes per value is needed. Single precision numbers use 4 bytes and double precision numbers use 8 bytes. This will yield about 7 digits of precision for single precision and 16 digits for double.

The choice between using the single precision (iSP) and double precision (iDP) versions of TUFLOW therefore depend on the situation that is being modelled. For example, consider a model which represents the breach of a tidal defence. The ground levels are in the order of tens of metres above datum. The predicted water levels might be 5.000001AHD for TUFLOW iSP and 5.000000mAHD for TUFLOW iDP. The additional significant figures offered by TUFLOW iDP has a negligible impact on the model results. On the other hand, if the study area were located on much higher ground (over 100mAHD) the additional significant figures may have a much greater impact on the overall results of the model.

Similarly, the volumes or depths of water that are expected in the model also influence the choice between the iSP and iDP versions of TUFLOW. Direct rainfall models where large amounts of sheet flow are predicted, typically require the additional significant figures offered by TUFLOW iDP. This may become apparent if high mass balance values are experienced when the model is simulated using TUFLOW iSP.

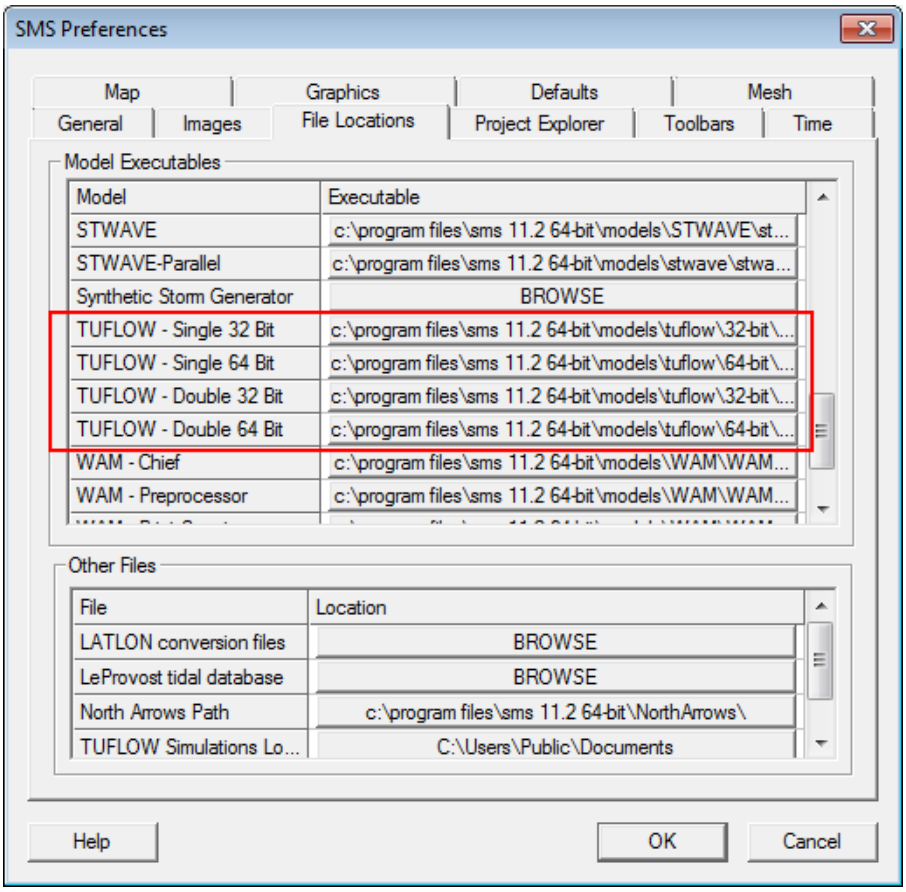
Note that the choice of single or double precision will also impact on simulation times and RAM allocation. iDP versions of TUFLOW will take approximately 25% longer to run and require up to twice as much memory, limiting the ability to run concurrent simulations. Refer to Sections 3.5 and 14.4 for further information. Therefore, if the results of a model run in both iSP and iDP versions of TUFLOW prove to be similar, use the iSP version of TUFLOW is recommended to take advantage of the faster simulation times.

See also the discussion on precision in Section 10.4.1 specifically for models run using TUFLOW GPU.

11.4.3 Using TUFLOW with Flood Modeller / XP-SWMM, or from SMS

Flood Modeller (formerly known as ISIS) 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 Flood Modeller or XP SWMM access them (usually in the same folder as the Flood Modeller and XP-SWMM .exe files). Alternatively, modifying the path and environment variable found within Advanced Settings allows 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.



11.4.4 Customising TUFLOW using TUFLOW_USER_DEFINED.dll

The TUFLOW_USER_DEFINED.dll allows users to customise TUFLOW to suit their needs. The primary use has been for users to customise TUFLOW's flood hazard output. The relevant routines can be requested from support@tuflow.com, modified by the user (requires some minor knowledge of programming using Fortran or similar) and emailed back to support@tuflow.com, upon which TUFLOW_USER_DEFINED.dll will be recompiled and returned to the user. For more information on utilising the user defined routines, contact support@tuflow.com.

11.5 Running Simulations

11.5.1 Dongle Types and Setup

BMT WBM dongles are either:

- A SoftLok (blue) dongle. The issue of these dongles was stopped in August 2010 as the vendor could not provide 64-bit support, but they will continue to be supported by future 32-bit versions of TUFLOW. Provided that the SoftLok dongles is maintained, it can be exchanged for a new WIBU (metal) dongle for a nominal fee (please contact sales@tuflow.com).
- A WIBU Codemeter (metal) dongle. 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);
    - 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 required purchase);
    - 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 – Allows up to a specified limit the number of TUFLOW simulations to be run from the one computer. For example a Local 1 licence will allow a single simulation to be performed on the machine. A local 4 licence allows up to 4 concurrent simulations and requires a quad core CPU to fully utilise. Local 1 to Local 16 licences can be configured.
- Network Licence – allows for multiple TUFLOW processes running at any one time across the organisation's LAN (Local Area Network) up to a specified limit. For example a Network

5 licence allows up to 5 concurrent simulations to be performed, this could be 5 simulations on a single computer, or a single simulation on 5 different computers or anything in between.

Refer to the installation instructions on the TUFLOW Wiki for both types of dongles.

11.5.1.1 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 licence is available, you can optionally set for TUFLOW to continue to try and find an available WIBU dongle licence. This is achieved using the "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" file described below. 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 type / provider. For example a WIBU dongle can't be used to finish a simulation if it is started with a SoftLok dongle which is removed.

11.5.1.2 "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" File

The "C:\BMT_WBM\TUFLOW_Dongle_Settings.dcf" file can be used to set WIBU settings, such as retry time interval and count. It has the same form and notation as a TUFLOW .tcf file.

Note: This file is optional and if not found the settings below are the default.

```
! 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 11-2) and 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.

11.5.1.3 Dongle Failure during a Simulation

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 continues trying until a license is found.

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.

11.5.2 Starting a Simulation

There are a number of ways TUFLOW can be started, in each case the TUFLOW executable is started with the TUFLOW control file (.tcf) as the input argument. There are a number of optional switches that can be specified when starting a simulation; these are outlined in Table 11-2.

Starting a TUFLOW simulation can be carried out in a multitude of methods:

1. From within a text editor;
2. Through a batch file;
3. From within your GIS software:
   (i) MapInfo (through the purchase of the MiTools add-on);
   (ii) ArcMap via the use of the ArcTUFLOW toolbox; or
   (iii) QGIS via the TUFLOW TuPlot QGIS Plugin.
4. From a Console (DOS) Window; or
5. Via the right mouse button in Microsoft Explorer.

This section of the manual only provides detailed instructions on initiating simulations through batch files as it is the most effective method to run several or many simulations. Instructions on all methods have been described on this page on the TUFLOW Wiki.

11.5.2.1 Batch File Example and Run Options (Switches)

A batch file is text file that contains a series of commands or instructions which is read by the Windows Operating System. The batch file requires the extension .bat to be recognised by Windows.

The simplest format of a TUFLOW batch file is to specify a single simulation. This is of the format:

```
<TUFLOW Executable> <TUFLOW Control File>
```

For example:

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe MR_H99_C25_Q100.tcf
```

The .bat file is run or opened by double clicking on it in Windows Explorer. This opens a Console Window and then executes each line of the .bat file. The above batch file will start the TUFLOW executable with the control file "MR_H99_C25_Q100.tcf" as the input file. As the control file argument does not have a file path, the batch file must be located in the same folder as the .tcf file. This could also include a relative or absolute file path, for example in the batch file line below, the batch file could be stored in any folder as the absolute file path is used. As outlined in Section 4.3 the relative file path is typically preferred for TUFLOW modelling inputs.

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe C:\Example\MR_H99_C25_Q100.tcf
```

To start multiple simulations one after the other, these can be listed in succession.

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b MR_H99_C25_Q100.tcf
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b MR_H99_C25_Q050.tcf
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b MR_H99_C25_Q020.tcf
pause
```

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). To carry out the simulations the –t can be removed or replaced with the –x (execute) switch. This switch is optional, but is useful when editing the .bat file to quickly change between –t and –x switches.

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q100.tcf
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q050.tcf
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q020.tcf
pause
```

Comment lines are specified in a .bat file using "REM" (remark) in the first column. Alternatively "::" has a similar effect. See also http://ss64.com/nt/rem.html. For example, if you want to re-run only the first simulation in the examples above, edit the file as follows:

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q100.tcf
```

```
REM C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q050.tcf
:: C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b -x MR_H99_C25_Q020.tcf
pause
```

Table 11-2 below describes other switches that are available. 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).

Any of the options can be prefixed by a:

- "-" (short dash or minus sign);
- "–" (long dash); or
- "/" (forward slash)

For example, to start a model in batch mode, the following all perform the exact same function:

```
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_Q100.tcf
C:\TUFLOW\Releases\2010-10\iSP-w64\TUFLOW_iSP_w64.exe /b MR_H99_C25_Q100.tcf
```

Table 11-2 TUFLOW.exe Options (Switches)

Switch	Description
-b	Batch mode. Used when running two or more simulations in succession from a .bat file (see Section 11.5.2.1). 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 -cap -cncf	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 (for example, only .mif/.mid MapInfo files will be copied, .tab MapInfo files will not). The –ca 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. The –cap option allows the user to specify an alternate path in which to copy all files with the same name. For example, specifying the following, will place a copy of the model into a folder C:\put_model_here: `TUFLOW.exe" -cap "C:\put_model_here" "C:\TUFLOW\runs\M01_5m_003.tcf"` The -cncf option copies the essential input files and excludes all check files. Specifying either –c, -cncf, –ca or –cap on the TUFLOW command line, 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, specifying:

Switch	Description
	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 or –cap option. Note: **a. Use the full path to the .tcf file (this is the default if running from UltraEdit or using the right click approach).** b. Make sure there is sufficient disk space (no checks for sufficient disk space are made). c. Output folders and some output files are 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:”. g. 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. h. 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. i. If –c, -cncf, –ca or –cap is specified, then -t is automatically invoked (ie. the simulation does not commence, only the input data is tested/checked). j. 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 .tef 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 (up to a maximum of nine) events per simulation may be specified by placing a number after –e. –e and –e1 are treated the same (don’t specify both of them otherwise indeterminate results may occur). Also see Section 11.3 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.

Switch	Description
-nwk	Force TUFLOW to search for a network dongle (ie. skip the search for a standalone dongle).
-od<drive>	The Output Drive for a simulation can be specified with the –od command line option. For example –odC will redirect all outputs to the C:\ drive.
-nq	The use of the –nq (no queries) switch prevents any queries from displaying when simulating a TUFLOW model. At this stage, the only query not displayed if this option is used is if you press Ctrl-C to terminate a simulation cleanly. If –nq is specified and Ctrl-C is pressed, the simulation terminates cleanly without a query dialog, so be careful if copying and pasting in another window!
-oz <name>	Map output includes Output Zone <name>. For more information on Output Zones, please refer to Section 9.4.3. For example "-oz ZoneA" would include output for Zone A, similar to the .tcf command: `Model Output Zones == ZoneA`
-pu <id>	Used to select which processing units to direct the simulation towards. At present this only applies to the GPU solver where a simulation is to be directed to a particular GPU card or cards. –pu must be specified once for each device. For example, to direct the simulation to GPU devices 0 and 2, specify –pu0 –pu2. Can be used in place of the .tcf command GPU Device IDs. Note that the numbering starts a 0 for GPU devices. Override files (see Section 4.5.1) can be used to control the device ID's used by a range of computers accessing the same control file.
-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 11.3.2 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	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 which computer is clogging up network licences! See Section 12.4. 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.

Switch	Description
	At present there is a limit of 64 characters for the path. **Note: This feature is only available for SoftLok (blue) dongles.** (See Section 11.5.1.2 for equivalent option for WIBU dongles, and also Simulations Log Folder.)
-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 a .bat file all initialise without error, 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).

11.5.2.2 Advanced Batch Files

Batch files can be setup so that they are more generic and easily customised when moving from one project to another. Within the example below, a variable, TUFLOWEXE, is used to define the path to the TUFLOW executable, 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\2013-12\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 executable changes, or to run a different version of TUFLOW, it is just a simple single line change in the batch file. In the above, note the use of quotes around %TUFLOWEXE% in the definition for the RUN variable – quotes are needed around file path names whenever they contain a space. Also note there is no space between the variable name and the equals sign.

For examples of batch files used for running multiple events and different scenarios from the same .tcf file, see Sections 11.3.1 and 11.3.2.

It is also possible to create control logic such as loops in a batch file. For example this could be used to loop through a large number of events and scenarios.

```
:: This sets the variables as local, another batch file can use the same variables
SetLocal

:: set up variables
set TUFLOWEXE=C:\Program Files\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe
set RUN=start "TUFLOW" /low /wait /min "%TUFLOWEXE%" -b
```

```
set A=Q010 Q020 Q050 Q100 Q200
set B=10min 30min 60min 120min 270min

:: Loop Through
FOR %%a in (%A%) do (
    FOR %%b in (%B%) DO (
        %RUN% -e1 %%a -e2 %%b filename_~e1~_~e2~.tcf
    )
)
pause
```

Further guidance on advanced batch files, including looping examples can be found on this TUFLOW Wiki Page.

11.5.3 Windows 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 able to do other computing tasks until the simulation is finished.

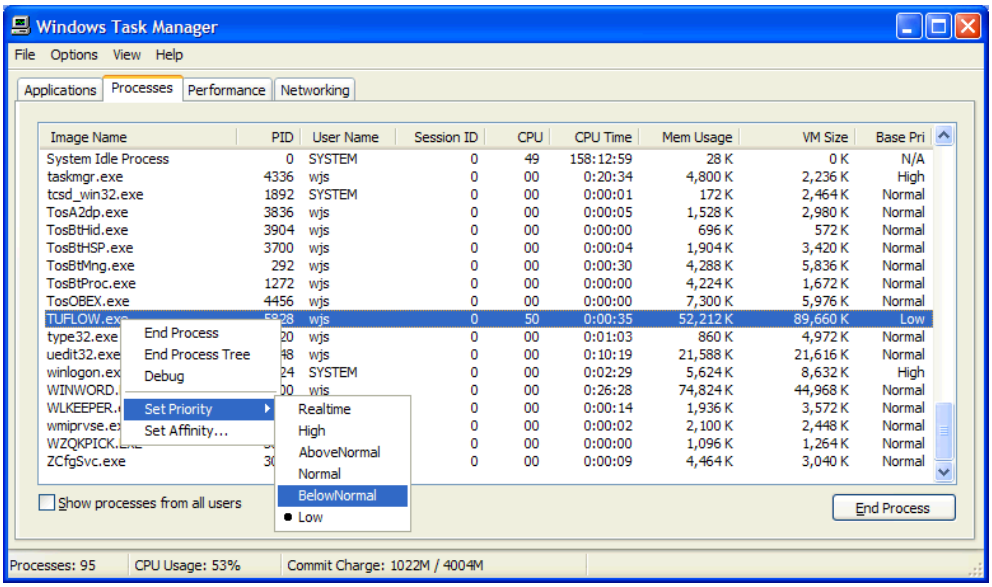


Figure 11-1 Simulation Priority via Windows Task Manager

12 Check and Log Files

Chapter Contents

12 Check and Log Files 12-1

12.1 Introduction 12-2

12.2 Console (DOS) Window Display 12-3

12.2.1 The Console (DOS) Window Does Not Appear! 12-5

12.2.2 Console Window Shortcut Keys 12-6

12.2.3 Customisation of Console Window 12-7

12.3 Message Boxes 12-8

12.4 _TUFLOW Simulations.log Files 12-9

12.4.1 Local .log File 12-9

12.4.2 Global .log File 12-11

12.5 ERROR, WARNING and CHECK Messages 12-12

12.6 Simulation Log Files (.tlf and .tsf files) 12-13

12.7 1D Output File (.eof file) 12-14

12.8 GIS Workspaces (.wor and .qgs files) 12-16

12.9 Check Files and GIS Layers 12-17

12.10 Visualising and Querying Check Layers 12-22

12.11 Mass Balance Output 12-23

12.1 Introduction

This chapter of the Manual describes the check, log and quality control outputs from TUFLOW. This encompasses the following outputs:

- The Console Window (Section 12.2);
- Message boxes (Section 12.3);
- Simulation log files (.log) (Section 12.4);
- GIS Messages layer (Section 12.5);
- TUFLOW log file (.tlf) and summary file (.tsf) (Section12.6);
- 1D Output File (.eof) (Section 12.7);
- GIS Workspace files (.wor and .qgs files) (Section 12.8); and
- Check Files (Section 12.9).

Note: Viewing and processing of results are described in Chapter 13.

12.2 Console (DOS) Window Display

TUFLOW displays a lot of information to the Console (DOS) Window during the data input and simulations stages. However, this typically processes so quick, that it may be difficult to read. The log file (.tlf) (Section 12.6) and check files (Section 12.9) contains this information in a more usable format. However, you can look back through the Window buffer to establish where in the input data process the problem occurs.

After a model has completed initialising its input data and has started its computations, 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 contains a lot of useful information to keep an eye on your simulations as they progress. Whilst we don't expect you to spend your time watching the simulations DOS window, it is useful to glance at occasionally to check that your simulations are "happy".

The Console Window appears as something similar to that shown below:

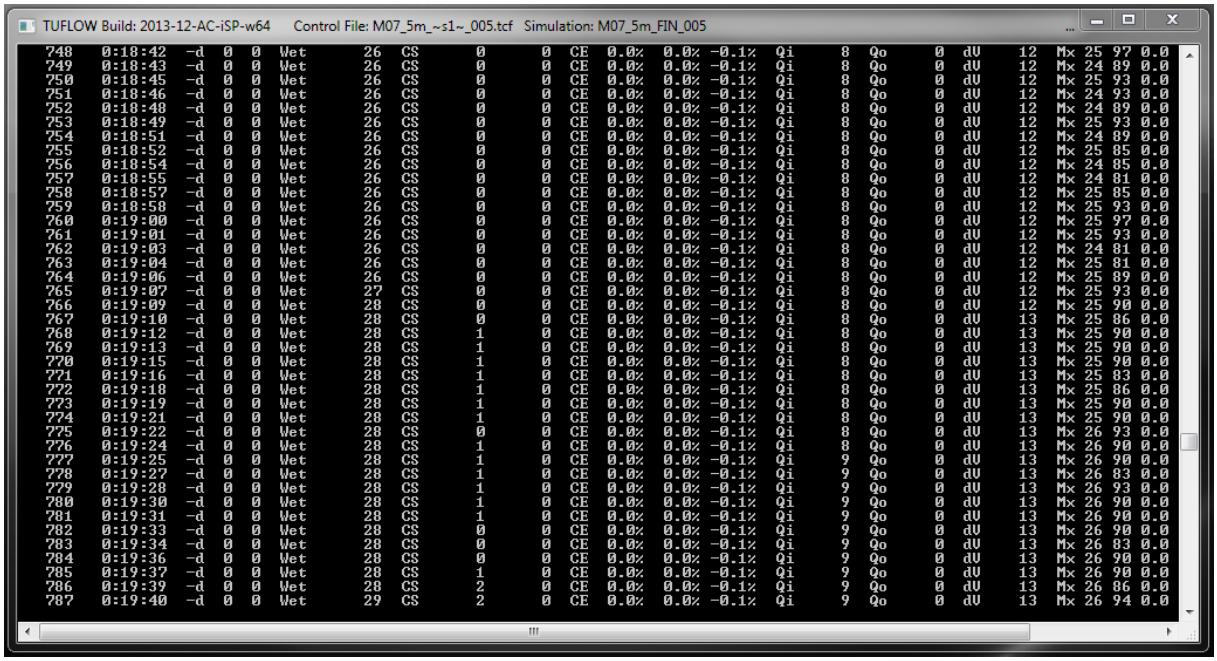


Figure 12-1 Example Console (DOS) Display Window

The following information is shown along each line in order of occurrence. 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 12.5). Negative depths indicate the model is having difficulty in convergence at that location, which may lead to an instability. See also Section 14.3.2.

- "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 m^3/s. If the inflow exceeds 999999m^3/s or falls below -99999m^3/s, the flow is expressed in units of 1,000m^3/s and a single quote symbol is displayed after the number. A double quote symbol indicates the flow is expressed in units of 1,000,000m^3/s.
  - "Qo" followed by the total flow out of the model (all domains) in m^3/s. If the outflow exceeds 999999m^3/s or falls below -99999m^3/s, the flow is expressed in units of 1,000m^3/s and a single quote symbol is displayed after the number. A double quote symbol indicates the flow is expressed in units of 1,000,000m^3/s.
  - "dV" followed by the change in volume in m^3 of the model (all domains) since the last display time. If the change in volume exceeds 999999m^3 or falls below -99999m^3, the amount is expressed in units of 1,000m^3 (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 1,000,000m^3.
- If maximums are being tracked (ie. Maximums and Minimums is set to ON or ON MAXIMUMS ONLY), additional information will be displayed to the Console Window and

.tlf file for each Screen/Log Display Interval. Three numbers are displayed after "Mx" at the end of each line. The first two numbers are the percentage of 1D nodes and percentage of 2D cells that reached a new maximum in the last computational timestep. The third number is the time in decimal hours since no new maximum was recorded anywhere within the model. For example, "Mx 10 21 0.0" indicates that 10% of 1D nodes and 21% of 2D cells recorded a new maximum last timestep, and the time since the last recorded maximum is zero. Once all 1D nodes and 2D cells have reached their maximums the third (time) value will increase above zero.

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 14.1.

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 clock time and CPU time. If the CPU time is significantly lower than the clock time, then either the simulation was paused for a period of time (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).

12.2.1 The Console (DOS) Window Does Not Appear!

Reasons that TUFLOW.exe won't start (ie. no Console Window appears) are:

- When the virtual memory allocation on your computer is congested. Check the virtual memory usage using Windows Task Manager. 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. 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 for modelling computers.
- Later versions of UltraEdit have issues with opening a DOS window, 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 11.5.3, ensure the DOS Box checkbox is not ticked (otherwise two DOS windows appear). This is discussed in the TUFLOW Wiki.

If you continue to have problems please contact support@tuflow.com.

If you are running from a batch file and the window briefly appears, then disappears straight away, you can force the window to stay open so you can diagnose the issue using the following method:

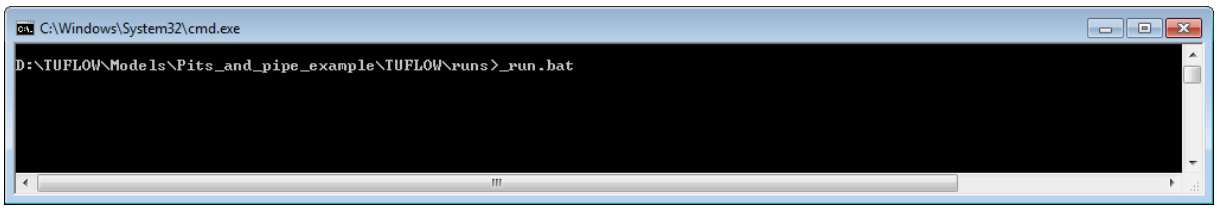
Remove the 'Start "TUFLOW"' commands from the run line, this spawns a new window (which allows multiple simulations to be started at the same time). If this is removed, the simulation will start in the main batch window and give you greater control. Add "Pause" at the end of the batch it will keep the window open. For example, changing:

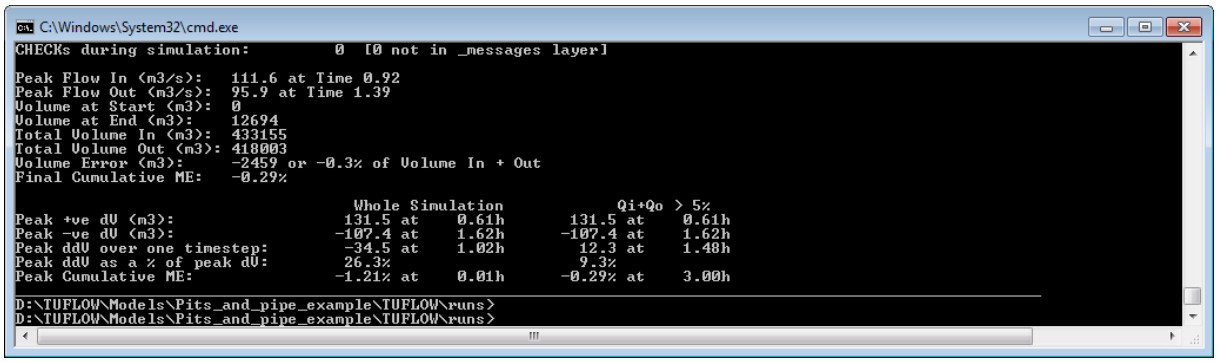
```
Start "TUFLOW" C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b MR_H99_C25_Q100.tcf
```

To:

```
C:\TUFLOW\Releases\2013-12\w64\TUFLOW_iSP_w64.exe -b MR_H99_C25_Q100.tcf
Pause
```

When TUFLOW is started a new console window won't appear, only the batch file. If there is any issue, the pause will ensure the window is kept open. This can also be achieved by opening a command prompt in the TUFLOW\runs\ folder and starting the batch file from the command prompt. The TUFLOW output is then directed to the command prompt, as per the images below.

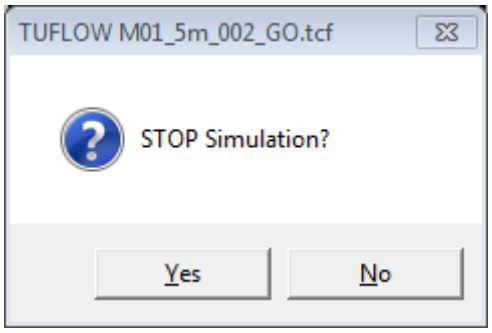




12.2.2 Console Window Shortcut Keys

Useful Console Window shortcut keys 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 and release any network dongle licence. The simulation is logged as being INTERRUPTED in the .tlf and .log files (Refer to section 12.4). Clicking on "No" will continue the simulation.



Ctrl+C is recommended instead of manually cancelling the simulation via clicking the display window close button. TUFLOW will not finalise writing the output result files if the simulation is cancelled via clicking the display window close button. For example, result maximums and minimums will not be written, even if Maximums and Minimums is set to ON.

12.2.3 Customisation of Console Window

The windows and buffer sizes of the Console Window 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.

It is possible to manually set the Console Window buffer as well change the font and colours of the TUFLOW window. The latter may be useful when differentiating different models on a shared computer. For further information, refer to guidance found on the TUFLOW Wiki.

12.3 Message Boxes

Windows message boxes are used to alert the user to an input problem and when a simulation has stopped/finished, or is unable to start. This can be suppressed with the -nmb input switch as described in Table 11-2.

Some example message boxes are shown below.

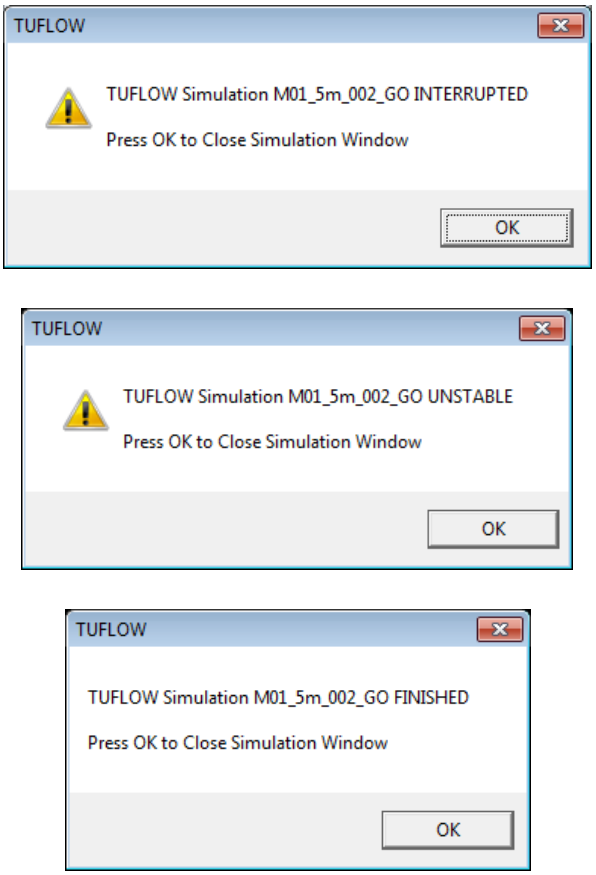


Figure 12-2 Example TUFLOW Message Boxes

12.4 _TUFLOW Simulations.log Files

TUFLOW activity (when simulations are started, finished, interrupted) 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. By default this is written to C:\BMT_WBM\log_ All TUFLOW Simulations.log. This global log file can also be disabled with the .tcf command Simulation Log Folder == DO NOT USE.

12.4.1 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.
    - CH2MHill 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 or demo 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" simulations, indicators of the mass error and stability performance are also provided as follows (see Table 14-1 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 that this file is not deleted or edited as it could provide a valuable trace back to past simulations.

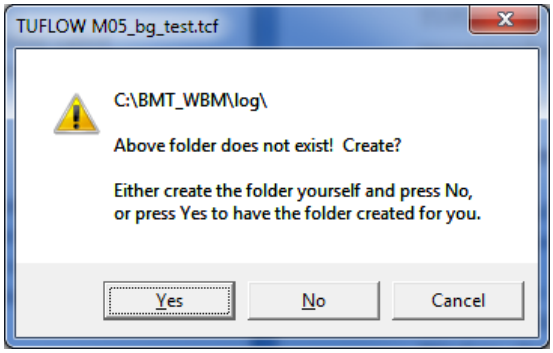
Excerpts from a local "_ TUFLOW Simulations.log" file are shown below:

```
2014-Sep-30 10:55  SL  PC01  Build: 2013-12-AC-iDP-w64  0:00:00 CPU      Started: BRI_Q100_exg_001.tcf
2014-Sep-30 12:14  SL  PC01  Build: 2013-12-AC-iDP-w64  0:04:46 CPU     Finished: BRI_Q100_exg_001.tcf
2014-Sep-30 12:21  SL  PC01  Build: 2013-12-AC-iDP-w64  0:00:00 CPU      Started: BRI_Q100_exg_001.tcf
2014-Sep-30 12:36  SL  PC01  Build: 2013-12-AC-iDP-w64  0:02:03 CPU  Interrupted: BRI_Q100_exg_001.tcf
2014-Sep-30 13:31  SL  PC01  Build: 2013-12-AC-iDP-w64  0:00:00 CPU      Started: BRI_Q100_exg_001.tcf
2014-Sep-30 13:33  SL  PC01  Build: 2013-12-AC-iDP-w64  0:00:00 CPU      Started: BRI_Q100_exg_001.tcf
2014-Sep-30 13:33  SL  PC01  Build: 2013-12-AC-iDP-w64  0:05:37 CPU     Finished: BRI_Q100_exg_001.tcf
2014-Sep-30 13:37  SL  PC01  Build: 2013-12-AC-iDP-w64  0:00:00 CPU      Started: BRI_Q100_exg_001.tcf
```

```
2014-Sep-30 13:38  SL  PC01  Build: 2013-12-AC-iDP-w64  0:03:51 CPU     UNSTABLE: BRI_Q100_exg_001.tcf
```

12.4.2 Global .log File

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 11.5.1.2).
- For the SoftLok (Blue) dongles, you can also use the -slp TUFLOW.exe option (see Table 11-2).
- 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.

The entries to the global .log file are as described for the local .log file above.

12.5 ERROR, WARNING and CHECK Messages

Error, warning, check and other useful messages that are output to the Console Window and log file are also output to a .mif or .shp file provided the message can be geographically located within the model domains. The messages and other information are written to a file called <.tcf filename>_messages.mif or <.tcf filename>_messages_P.shp which is located in the same folder as the .tcf file or in the Log Folder.

The three levels of messages generated in increasing order of severity are:

- Check;
- Warning; and
- Error (the simulation is unable to continue).

A check indicates that something unusual occurred, for example a breakline failed to modify any cell elevations (Check 2079).

A warning is more severe than a check message, but the simulation will still progress. For example a Manning's n value of 0.001 is outside the typical range (Warning 2218).

An error indicates that the simulation is unable to continue. For example if no active cell is found within source area boundary an error (Error 2014) will occur and the simulation will terminate.

This feature allows rapid identification of the error location within the GIS environment of data input errors and simulation instabilities and potential problems. Use of this feature can save days when setting up and checking new models.

Some messages are 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 in MapInfo, 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 in MapInfo. Information on the adjustment is available in the attribute data for the cell.

A message numbering system has been adopted, prefixing all warnings with a four digit number. These numbers may be used to cross-reference the warning with a message database that is stored on the TUFLOW Wiki. The database contains detailed information on the CHECK, WARNING or ERROR to help check/resolve the issue. Each message falls into one of the following message categories:

- 0xxx warning messages refer to errors that occur in neither the 1D or 2D domains.
- 1xxx warning messages refer to errors that occur in the 1D domains.
- 2xxx warning messages refer to errors that occur in the 2D domains.

12.6 Simulation Log Files (.tlf and .tsf files)

TUFLOW produces a log file (.tlf 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 or .shp file (see Section 12.5). Opening the .mif or .shp file in the GIS often provides a far more rapid location of the message within the model domain(s) than via other ways.

A TUFLOW Summary File (.tsf file) is output to the same location as the .tlf file providing a more concise summary of the simulation. This file can also be regularly updated during a simulation. Features are:

- The command TSF Update Interval can be used to control the interval in seconds to update the .tsf file while a simulation is running.
- The .tsf file has a TUFLOW control file style syntax and contains information on the primary simulation inputs, the simulation status, mass balance outputs, etc. Additional information will be added based on user feedback (please email support@tuflow.com if you have a suggestion).

12.7 1D Output File (.eof file)

The .eof file contains a summary of the 1D domain input datasets. 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 relates to the second table. Similarly, 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. Therefore, the .eof file contains the final dataset that is used by the simulation.

By default the .eof file also contains a result summary 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 12-1, are useful for interrogating the hydraulic regime at nodes and channels. The writing of the outputs in the .eof file can be supressed by the Output Files Exclude == EOF command in the .tcf file.

Table 12-1 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/.shp 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.
	S	Upstream controlled friction flow occurred in a Steep (S) channel with a Froude Number greater than one (1).
	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.

Flag (Space 1)	Flag (Space 2)	Description
	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 5-4. Culvert channels only.
	E	The channel or node is empty or dry (ie. the head or water level is at the bottom of the node). 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.

12.8 GIS Workspaces (.wor and .qgs files)

A MapInfo workspace (.wor) and QGIS workspace (.qgs) is automatically created for every simulation. They are named <tcf_filename>.wor / .qgs and are written to the same folder as the .tlf 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, or those with multiple events or scenarios.

The .wor file when opened in MapInfo simply opens the .tab layers. No Map or Browser windows are automatically opened. If the simulated model contains any .shp files, these are not opened however the .shp file layers used by TUFLOW can be viewed when opening the .wor file in a text editor.

Opening the .qgs file in QGIS will open all GIS input and output check layers (.mif and/or .shp). Note that the visibilities of the output layers are unchecked so that the display time is quick.

For ArcMap users, the .mxd files are not directly written by TUFLOW. It's the format is proprietary and can't be directly written. However, the ArcTUFLOW toolbox can be used to load the simulation input files into ArcMap.

12.9 Check Files and GIS Layers

Table 12-2 lists the various 2D and 1D check files that are output when the command Write Check Files is specified in the .tcf file and .ecf file respectively. 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. Note that a number of the check files listed below are only output when the corresponding feature has been used in the simulated model. The attributes of some check files (like that of the _2d_grd_check) may vary depending on the feature used within the model. **A complete list of attributes for each check GIS file is provided on the wiki. Each check file type has a link to the wiki page in the first column.**

Many of the GIS .mif/.mid and .shp files can also be used for creating new, pre-formatted, GIS layers.

Table 12-2 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.**
_bcc_check.mif _bcc_check_R.shp	GIS .mif/.mid or .shp files showing the cell location of 2D boundary conditions.
_DEM_M.flt _DEM_M.asc	A DEM of the final material ID values, similar to the DEM_Z check grid described below. The .tcf command Grid Format can be used output this check file in ASCII format rather than the default FLT format.
_DEM_Z.flt _DEM_Z.asc	A DEM of the final ground/bathymetry elevations, including those from any 1D WLL mesh. The file is given a DEM_Z extension, and can be readily opened by most GIS and other GUIs. The default size of the grid cells is half the smallest 2D cell size. This can be changed using the Grid Output Cell Size command. Grid Format can be used to control the format of the file. To exclude writing this file, include "DEM_Z" in the Write Check Files EXCLUDE list.
_dom_check.mif _dom_check_R.shp	Contains a rectangle for each 2D domain showing the location, orientation and size of the domain.
_fc_check.mif _fc_check_R.shp	GIS .mif/.mid or .shp 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 _fcsh_uvpt_check_P.shp	Contains information on adjustments to the ZU/ZV cell sides as modified by Read GIS FC Shape commands.

Filename or Prefix/Suffix	Description
_glo_check.mif	GIS .mif/.mid or .shp files of any gauge level output (GLO) location.
_grd_check.mif _grd_check_R.shp	GIS .mif/.mid or .shp 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). Use the _uvpt_check.mif/.shp file to view these. Can also be written at different stages within a .tgc file (see Write GIS Grid). The file contains all modifications to the 2D grid at the point in the .tgc file that it is written. Note that a number of additional attributes are appended to the _grd_check layer when some features of TUFLOW have been used.
_input_layers.mif	GIS layer contains full file paths to all input layers used to compile the model. This layer is only written in .mif/.mid format and can be viewed in a text editor.
_lfcsh_uvpt_check.mif _lfcsh_uvpt_check_P.shp	Contains information on adjustments to the ZU/ZV cell sides as modified by Read GIS Layered FC Shape commands (refer to Section 6.12.2.2).
_lp_check.mif _lp_check_L.shp	GIS .mif/.mid or .shp files of any 2D longitudinal profile(s).
_po_check.mif _po_check_L.shp _po_check_P.shp	GIS .mif/.mid or .shp 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 _sac_check_R.shp	Shows the lowest cell selected for Read GIS SA inputs, and cells selected if using Read GIS SA PITS.
_sh_obj_check.mif _sh_obj_check_R.shp	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 _uvpt_check_P.shp	GIS .mif/.mid or .shp files containing the initial velocities, roughness value, FLC, WrF, 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.
_vzsh_zpt_check.mif _vzsh_zpt_check_L.shp _vzsh_zpt_check_P.shp	Contains information on Zpts that have been modified by Read GIS Variable Z Shape commands.

Filename or Prefix/Suffix	Description
_zln_zpt_check.mif _zln_zpt_check_P.shp	GIS .mif/.mid or .shp 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. When written in .mif/.mid format, the points are given different symbology according to whether they have been raised or lowered (up or down triangles) or remain unchanged (a cross).
_zpt_check.mif _zpt_check_P.shp	GIS .mif/.mid or .shp 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 GIS 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.
_zsh_zpt_check.mif _zsh_zpt_check_P.shp	Contains Zpts that have been modified by Read GIS Z Shape commands. When written in .mif/.mid format, the points are given different symbology according to whether they have been raised or lowered (up or down triangles) or remain unchanged (a cross).
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.
_pit_inlet_tables_check.csv	Similar to the _1d_bc_tables_check.csv. It contains tabular data as read from the pit inlet database.
_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. Flood Modeller XZ processed, and MIKE 11 processed cross-section data included. Refer also to the _xsl_check layer.
_bc_check.mif _bc_check_P.shp	GIS .mif/.mid or .shp files of the final 1D boundary conditions (BC). If no boundary conditions were specified, empty .mif/.mid or .shp files are written that can be used to set up a new layer.
_hydprop_check.mif _hydprop_check_L.shp	Contains the hydraulic properties 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.
_inverts_check.mif _inverts_check_P.shp	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 channel, or to nodes.

Filename or Prefix/Suffix	Description
_iwl_check.mif _iwl_check_P.shp	GIS .mif/.mid or .shp files of the initial water levels at the 1D model nodes.
_mhc_check.mif _mhc_check_P.shp	Manhole check layer including any automatically created manholes.
_nwk_C_check.mif _nwk_C_check_L.shp	GIS .mif/.mid or .shp files of the final 1D model network. This check layer contains the channels of the 1D domain only. The _nwk_N_check layer contains the nodes. The layer's lines are coloured based on the channel type (available for the .mid/.mif format only). 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.
_nwk_N_check.mif _nwk_N_check_P.shp	GIS .mif/.mid or .shp files of the final 1D model network. This check layer contains the nodes of the 1D domain only. The _nwk_C_check layer contains the channels. The node symbology is displayed as a red circle in MapInfo for nodes connected to two or more channels, a magenta circle for nodes connected to one channel and a yellow square for nodes not connected to a channel (available for the .mid/.mif format only). **This is very useful for checking for channel ends or nodes that are not snapped.** The top and bottom elevations of the NA table at nodes is output to the US_Invert and DS_Invert attributes.
_WLLo_check.mif _WLLo_check_L.shp _xWLLo_check.mif _xWLLo_check_L.shp	GIS layer of the WLL objects. The attributes provide information on which nodes the WLL are associated with. The _WLLo_check layers are written for ESTRY 1D domains, whereas the _xWLLo_check layers are written for external 1D domains, such as Flood Modeller or XP-SWMM.
_WLLp_check.mif _WLLp_check_P.shp _xWLLp_check.mif _xWLLp_check_P.shp	GIS layer of the WLL points that were generated along the WLLs. These points can then be used for Read GIS WLL Points (see Section 9.5.3). Contains the following attributes. The _WLLp_check layers are written for ESTRY 1D domains, whereas the _xWLLp_check layers are written for external 1D domains, such as Flood Modeller or XP-SWMM.
_xsl_check.mif _xsl_check_L.shp	GIS layer containing tabular data as read from 1d_xs input layers (see Table 5-14). Contains the XS ID and other useful information on the cross-section properties. Refer also to _ta_tables_check.csv.
_x1d_chans_check.mif _x1d_chans_check_L.shp	GIS layer containing the location of 1D channels from an external 1D domain.

Filename or Prefix/Suffix	Description
_x1d_nodes_check.mif _x1d_nodes_check_P.shp	GIS layer containing the location of 1D nodes from an external 1D domain.
2D/1D Models	
_1d_to_2d_check.mif _1d_to_2d_check_R.shp	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 using the .mif/.mid format to allow for easy visualisation of whether the right connections have been made. Additional information is supplied through the attributes. In the .mif layer, all SX cells connected to the same 1D node are grouped together as one object, and the Lowest_ZC_2D value is the lowest 2D cell ZC value of all the cells connected to the 1D node. For the .shp layer, it's not possible to group the cells as one object, so each cell is separate, and therefore, when clicking on the SX cells, note that the value is still the lowest ZC of all 2D cells, not of the individual cell. For 2D HX links, the value is the ZC value of the individual cell.
2D/2D Models	
_2d_to_2d_check.mif _2d_to_2d_check_R.shp	Displays the 2D cells used to link two 2D domains together via a 2d_bc type "2D" boundary. Similar to the _1d_to_2d_check layer, the cells connected to the same hidden 1D node are given the same colour in .mif/.mid format. Use the command Reveal 1D Nodes to view the locations of the hidden 1D nodes.

12.10 Visualising and Querying Check Layers

Many of the GIS check layers output by TUFLOW may be mapped within GIS to provide a visual representation of how the input data has been interpreted by the model. This often makes it easier and quicker to review a certain aspect of the model rather than individually viewing the attributes of each GIS check layer.

For example, using the styling functionality of you GIS software the user could:

- View the spatial distribution of the Soil Infiltration layer (attribute in _grd check file).
- View the conveyance of 1D channel layers (attribute in _hydroprop check).

Examples of visualising check files can be accessed from the TUFLOW Check Files Wiki Page. Any feedback or requests for examples is welcome and can be sent to support@tuflow.com. Examples of viewing these check files for different GIS packages are provided via the links below to the TUFLOW Wiki:

- MapInfo Thematic Mapping
- ArcMap Thematic Mapping
- QGIS Thematic Mapping

12.11 Mass Balance Output

Mass balance information is generated by setting Mass Balance Output to ON (the default). If Mass Balance Output is ON the following mass balance output is available:

- As discussed in Section 12.1 the cumulative mass error percentage appears as three numbers after the letters "CE" in the Console Window. This data is also output to the .tlf file (Section 12.6). 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 12-3,
- Table 12-4 and
- Table 12-5. Monitoring these numbers is important so as to establish the "health" of the model, as discussed in Section 14.1. Ideally all these percentages should be within ±1%. Much higher numbers may occur at the start of a simulation, especially if there are 2D domain(s) rapidly wetting. 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) (see
    - Table 12-3). It is written to the .tcf Output Folder .
    - The **_MB1D.csv** file contains mass balance reporting for all the 1D domains (see
    - Table 12-4). It is written to the .ecf Output Folder.
    - The **_MB2D.csv** file contains mass balance reporting for all 2D domains together and for each individual 2D domain (see
    - Table 12-5). It is written to the .tcf Output Folder.

  These files also report inflows and outflows across HX and SX connections for 1D/2D links between TUFLOW's 1D scheme (ESTRY) and other external 1D schemes (eg. XP). The 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. Both MB1 and MB2 outputs are a 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. The MB1 output is accumulated since the previous output time and the MB2 output is accumulated over the entire simulation.

Time based 1D mass error is output as a GIS layer to a _TSMB.mif/_TSMB_P.shp layer (see

- Table 12-6 for a description of the attributes and also Section 13.2.3). 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 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 Section 13.2.3 and
- Table 12-7 for a description of the attributes. Using GIS thematic mapping of the ME_Avg_Abs attribute is a powerful way of identifying any problematic 2D HX links.

It is possible to specify different time intervals for the display on the screen and the _MB.csv output files. The .tcf command Mass Balance Output Interval is used for setting the interval in the _MB.csv files.

A summary of key model performance indicators is also reported at the end of the simulation in the Console Window and .tlf file (see Section 14.2.1).

Healthy models fall within ±1% cumulative mass error (see Section 14.1 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 double precision versions of TUFLOW (see Section 11.4). This scenario may also require the default wet/dry depth (Cell Wet/Dry Depth) to be reduced to minimise mass errors that can arise from cells frequently wetting and drying with larger wet/dry depths.
- Poorly configured 1D/2D or 2D boundaries which are causing oscillations to occur.
- High frequency of repeated wetting and drying.
- Models located at high elevations above sea level, especially if the inflows are relatively small or direct rainfall is applied. These models should be run using double precision versions of TUFLOW (see Section 11.4). 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).
- 1D nodes that are frequently drying (undershooting), or are being limited if Head Rate Limit is being used (not recommended) can result in mass errors. The _TSMB.mif and _TSMB1d2d.mif are useful for reviewing 1D mass error. These files report the mass error values 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 $1m^3/s$ at a node where $1,000m^3/s$ is passing though is not an issue, while it would be if only $2m^3/s$ was passing through the node).
- 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 14.6).

Table 12-3 _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 entering the model since the previous time in m^3.
Tot Vol Out	The total volume of water leaving the model since the previous time in m^3.
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 since the previous time in m^3.
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 if there are 2D cells rapidly wetting and the flow through the model ("Vol I+O") is relatively small. This is a characteristic of 2D domains, particularly when using the direct rainfall approach. 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 the model in m^3.
Cum Vol I+O	The cumulative volume of water entering and leaving the model in m^3 (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 (%)	(“Cum Vol Err”/max(“Tot Vol” and “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 cells 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. 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 (“Tot Vol” and “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 (%)	(“Cum Vol Err”/“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 cells 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 “Cum Vol I+O” is less than $1m^3$, “Cum Q ME (%)” is set to zero to avoid divide by zero calculations.

Table 12-4 _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.
Q2D V In	The volume of water in m^3 flowing into hidden 1D nodes from 2D QT flow boundaries, since the previous time.

Column	Description
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"/(ΣV In + ΣV Out))*100 (ie. the percentage mass error based on the volume of water flowing through the 1D domains since the previous time). If (ΣV In + ΣV 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 (ΣV In + ΣV 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 1 m^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 1 m^3, "Cum Q ME (%)" is set to zero to avoid divide by zero calculations.

Table 12-5 _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 m3 since the previous time.
V Err	"dVol" minus "V In-Out" (ie. the volume error or amount of water in m3 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. 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. If "Cum V In+Out" is less than 1m^3, "Cum Q ME (%)" is set to zero to avoid divide by zero calculations.

Table 12-6 _TSMB GIS Layer 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 12-7 _TSMB1d2d GIS Layer 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 differentiate the object display based on the 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.

13 Viewing and Post-Processing Output

Chapter Contents

13 Viewing and Post-Processing Output 13-1

13.1 Introduction 13-2

13.2 Plot Output Files and Formats 13-3

13.2.1 Overview 13-3

13.2.2 .csv File Outputs 13-3

13.2.3 _TS GIS Layers 13-5

13.3 Plot Output Viewers and Tools 13-7

13.3.1 GIS Plot Viewers (TuPlot and miTools) 13-7

13.3.2 Other Plotting Viewers (eg. Excel) and Scripts 13-8

13.4 Map Output 13-9

13.4.1 Overview 13-9

13.4.2 GIS Layers (.MIF and .SHP) 13-9

13.4.2.1 Maximum and Minimum Output 13-9

13.4.2.2 _ccA GIS Layer 13-10

13.5 Map Output Viewers and Animators 13-11

13.6 Conversion to GIS Formats 13-12

13.7 Impact Analysis Mapping 13-13

13.1 Introduction

This chapter provides guidance on viewing and processing TUFLOW output.

Note: For details on the numerous options for customising TUFLOW output please see Chapter 9.

Note: We have transferred, and are continuing to transfer, content of relevance to this Chapter to the TUFLOW Wiki, therefore this Chapter is relatively short! Links to the various TUFLOW Wiki pages are provided where available. As more Wiki pages are populated, the manual will be updated accordingly. It is also recommended to Google "TUFLOW Wiki" then the software/topic of interest or please contact support@tuflow.com if the information being sought cannot be found.

13.2 Plot Output Files and Formats

13.2.1 Overview

The following sections describe plot or time-series outputs that are generated by a TUFLOW simulation.

Note: Plot outputs that need to be configured prior to the simulation are described in Section 9.3. Also see Table 9-1 for a list of commands that effect plot output.

13.2.2 .csv File Outputs

A range of model time-series results from the 1D and 2D domains are output in .csv file format. These files are typically used in spreadsheet software for graphing and time-series analysis. The commands Start Time Series Output and Time Series Output Interval are used to control the period and frequency of output.

The 1D water levels at nodes (_1d_H.csv), and flows (_1d_Q.csv) and velocities (_1d_V.csv) in channels are output in separate .csv files. This information is also provided in the _TS GIS layer (see Section 13.2.3) and within the .eof file (see Section 12.7). Maximum and minimum values are not output to the .csv files however are contained within both the _TS GIS layer and the .eof file as well as the 1d_mmH, 1d_mmQ and 1d_mmV GIS layers (refer to Section 13.4.2.1).

2D time-series data from plot output (PO) or longitudinal profile (LP) data at locations defined using 2d_po and 2d_lp layers (see Section 9.3.3) are output as _PO.csv and _LP< name>.csv files.

The 2016-03 release also includes the time-series and maximum (summary) output in .csv files for Reporting Locations (see Section 9.3.1) and for Structure Groups (see Section 9.3.2). These options have the ability to combine 1D and 2D domain outputs. The .csv files that are produced using the 2016-03 release are listed in Table 13-1.

The 2016-03 release, by default, uses a new plotting output folder structure. For the 2016 approach a "plot" folder is created under the Output Folder. The plot folder contains the subfolders and files as listed in Table 13-1. The previous approach is still supported for legacy models and can be set by using Output Approach == Pre 2016, noting that new features such as Reporting Locations are not supported if using this approach.

A python library and the QGIS TuPlot Plugin are available to provide powerful scripts and a GIS viewing platform to view and post-process these data. This .csv plotting data can also be accessed by standard spreadsheet software such as Microsoft Excel.

For models that contain 1D operational structures (refer to Section 5.8), an additional file (_1d_O.csv) will be output containing information on the operation of each structure over time. This file serves as a valuable check for the commands defined within the .toc Operating Control File.

Table 13-1 plot Folder File Descriptions (2016-03 Release)

Folder	Filename	Description
\plot\ Folder		
plot\	<simulation_id>.tpc	TUFLOW Plot Control file. This is a simple text file that contains information and links to the data available for the simulation. This file is used by TuPlot to load up complete plotting data sets and quickly access data.
\plot\csv\ Folder		
plot\csv\	<simulation_id>_1d_Chan.csv	Contains information on the channel connectivity and properties.
plot\csv\	<simulation_id>_1d_Cmx.csv	Contains the channel maximums data: flows and velocities as well as time of the maximums.
plot\csv\	<simulation_id>_1d_Nmx.csv	Contains the node maximums data: water levels and energy levels.
plot\csv\	<simulation_id>_1d_Node.csv	Contains information on the 1D nodes.
plot\csv\	<simulation_id>_1d_<ot>.csv	Time series output for output type <ot> at the 1D nodes or channels. For nodes, energy (E) and water level (H) are available. For channels, flow area (A), flow (Q) and velocity (V) are output. The output is controlled by the command Output Data Types, which by default includes "H", "Q" and "V".
plot\csv\	<simulation_id>_1d_O.csv	Operational structures time series output including information on the operational status and time varying values of variables.
plot\csv\	<simulation_id>_2d_<ot>.csv	Time series output for each 2d_po output type <ot> as triggered by Read GIS PO commands.
plot\csv\	<simulation_id>_RLP_H.csv <simulation_id>_RLP_Hmx.csv	Reporting Location water levels (_H) and maximums plus other information at the peak water level (_Hmx). See Section 9.3.1.
plot\csv\	<simulation_id>_RLP_Q.csv <simulation_id>_RLP_Qmx.csv	Reporting Location flows (_Q) and maximums plus other information at the peak flow (_Qmx). See Section 9.3.1.
plot\csv\	<simulation_id>_SHmx.csv <simulation_id>_SQ.csv	Maximums and other information (_SHmx) and time-series total structure flow (_SQ) for all 1D structures and grouped structure output (see Section 9.3.2). This output is controlled by the "S" option in Output Data Types, which by default includes "S".

Folder	Filename	Description
\plot\gis\ Folder		
plot\gis\	<simulation_id>_PLOT.csv	Summary .csv file containing information on the GIS objects and plot types available.
plot\gis\	<simulation_id>_PLOT_L	GIS layer in shapefile or MapInfo file format containing all plot line objects (eg. 1D channels and flow Reporting Locations).
plot\gis\	<simulation_id>_PLOT_P	GIS layer in shapefile or MapInfo file format containing all plot point objects (eg. 1D nodes and water level Reporting Locations).

13.2.3 _TS GIS Layers

The _TS GIS layers contain time-series output with each layer presenting different information as described below in Table 13-2:

Each layer contains several attributes at the start summarising the time-series data. These attributes are:

- The maximum and minimum values;
- The time in hours of the maximum and minimum values; and
- The average and the average of the absolute mass error values (for the TSMB and TSMB1d2d files only).

Note that the output frequency of the time output in _TS GIS layers is automatically adjusted so that the limit of 245 output times is not exceeded (this limit is the maximum number of attributes allowed in some GIS software). For example, if based on the Time Series Output Interval setting there are 400 output times, then every second time will be written to the _TS GIS layer giving a total of 200 output times, but at least the full hydrograph is displayed!

Table 13-2 _TS GIS Layer Descriptions

GIS Layer Name	Description
_TS.mif _TS_L.shp _TS_P.shp	Contains all 1D channel (velocities and flows), 1D nodes (water levels) and 2D PO (plot output locations from 2d_po layers).
_TSF.mif _TSF_P.shp	Contains the flow regime flags for culverts (see Figure 5-1 and Figure 5-2), and other types of channels (see Table 12-1).
_TSL.mif _TSL_P.shp	Contains the culvert and bridge loss coefficients after any adjustments if Structure Losses == ADJUST or the "A" flag has been specified. If there is a manhole connected to one/both ends of a culvert, the loss coefficients will be affected by any manhole energy losses as discussed in Sections 5.12.5.5. 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 _TSMB_P.shp	Contains the mass errors at 1D nodes (refer to Section 12.11 and Table 12-6)
_TSMB1d2d.mif _TSMB1d2d_P.shp	Contains the mass errors across 1D/2D interface linkages, ie. HX links (refer to Section 12.11 and Table 12-7)

13.3 Plot Output Viewers and Tools

13.3.1 GIS Plot Viewers (TuPlot and miTools)

GIS based utilities are available to assist in the viewing of 1D and 2D time series results. Information about these utilities is provided on the following TUFLOW Wiki pages:

- QGIS TuPlot Plugin
- MapInfo Plotting Functions using miTools

The 2016-03 TUFLOW release will be required to best utilise these visualisation options. The release includes a new Output Approach, which produces additional information to aid the data processing that is used by these utilities.

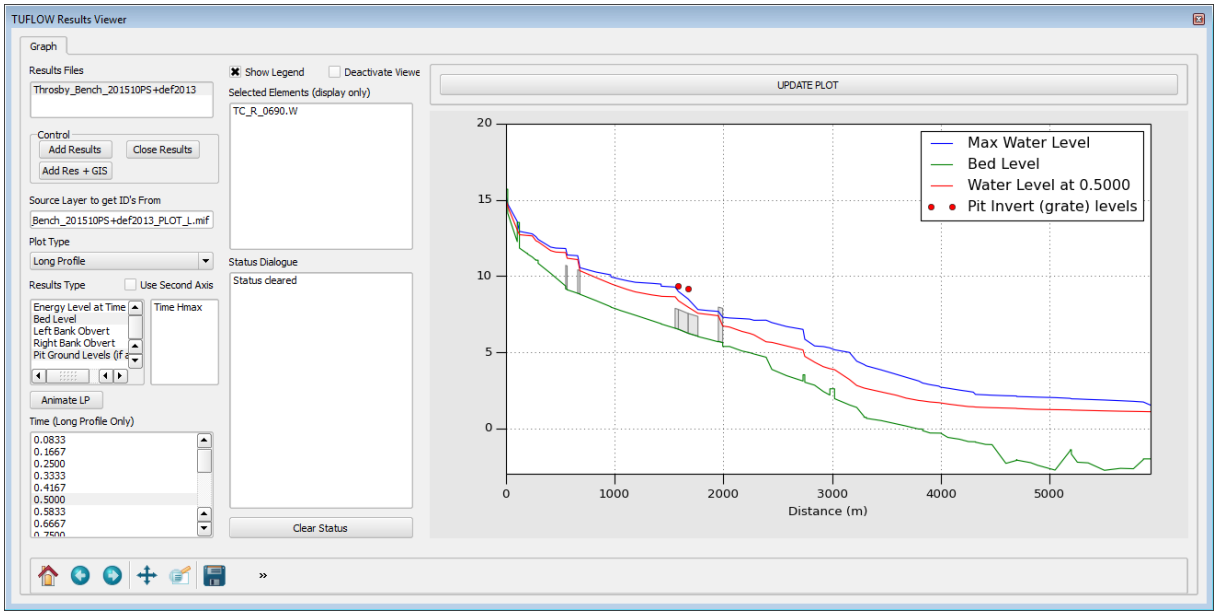


Figure 13-1 Example of the QGIS TuPlot Plugin Profile Plot

13.3.2 Other Plotting Viewers (eg. Excel) and Scripts

Excel is commonly used for charting and setting up report quality templates for plotting TUFLOW time-series output. All the .csv output is designed to be readily imported into Excel and used within the standard spreadsheet environment.

TUFLOW GUIs developed by other software vendors all include varying degrees of functionality for plotting and viewing time-series charts and profiles. Please refer to the user manuals and support services for these software.

Where repeated plotting of results occurs, or there are a large number of charts to produce, Python scripts offer a powerful option in terms of high quality and workflow efficiency. For some initial guidance on using Python scripts please see this TUFLOW Wiki Page.

13.4 Map Output

13.4.1 Overview

The following sections describe map outputs that are generated by a TUFLOW simulation.

Note: Map outputs that need to be configured prior to the simulation are described in the following Sections:

- **Section** 9.4**:** Customising Map Output
- **Section** 9.5**:** Including 1D Results in Map Output
- **Section** 9.6**:** Map Output Formats
- **Section** 9.7**:** Map Output Data Types
- **Section** 9.8**:** Specialised Outputs

Also see Table 9-1 **for a list of commands that effect map output.**

13.4.2 GIS Layers (.MIF and .SHP)

13.4.2.1 Maximum and Minimum Output

Maximum/minimum values for water levels at nodes, and flows and velocities in channels are output to GIS layers with the extensions1d_mmH, 1d_mmQ and 1d_mmV as well as to the end of the .eof file (see Section 12.7).

The GIS layers 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 absolute maximum (positive or negative) is provided. This is particularly useful for tidal reaches or where a channel has significant flows in both directions.

The _mmQ GIS layer, also contains the difference in maximum water level drop and the slope as a percentage of the water surface along the channel. This is useful for quickly identifying any troublesome behaviour along 1D networks by viewing/searching for any negative (adverse) slopes or large unexpected changes in flood level.

The _mmH GIS layer also contains a useful attribute, dH. This contains the largest water level drop across the channels that end at that node. 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.

13.4.2.2 _ccA GIS Layer

The _ccA GIS 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 (available for the .mid/.mif format only). The thinner and paler the line the smaller the pFull_Max value. The pFull_Time attribute is useful for thematically mapping in GIS to identify culverts that are performing well, and others that are not. The example shown in Figure 13-2 illustrates the _ccA GIS layer output. The thicker and darker the line, the better the pipe performed in terms of reaching full flow capacity.

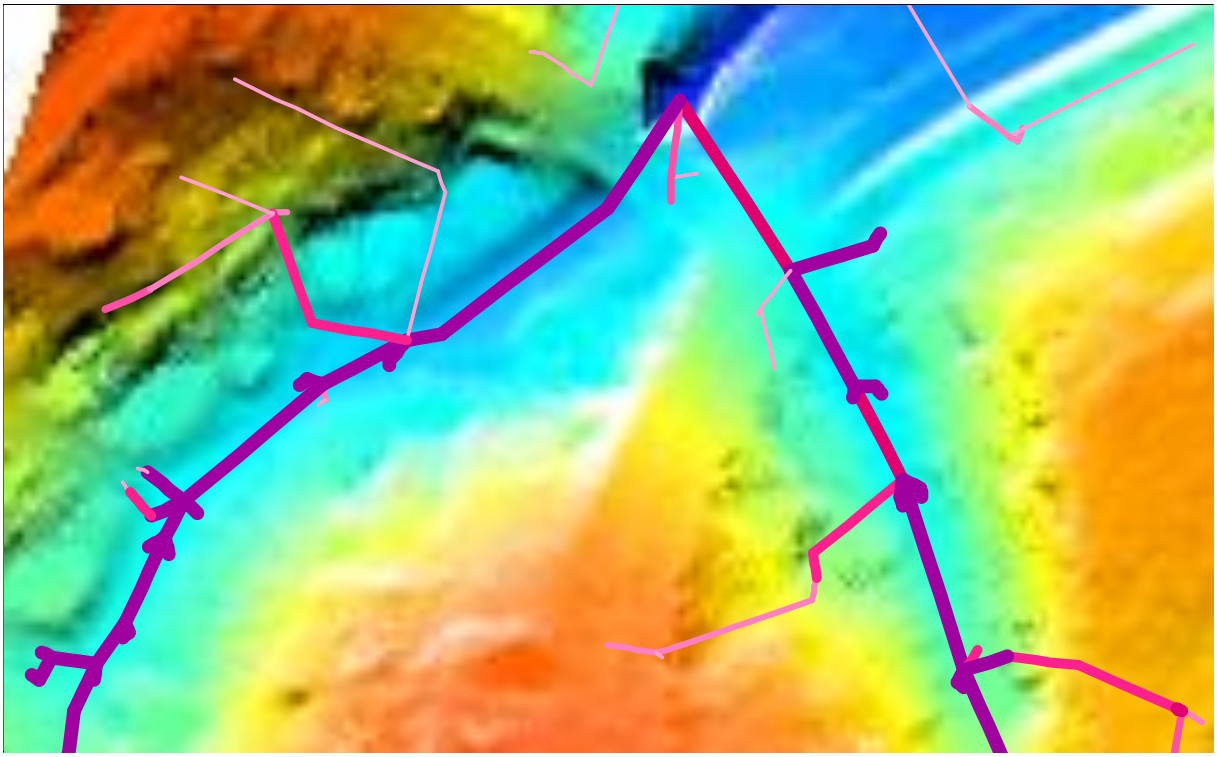


Figure 13-2 Example of the _ccA GIS Layer Showing Culvert Performance

13.5 Map Output Viewers and Animators

Viewing and animating TUFLOW map output is accomplished through a wide range of GIS and GUI software. Information, guidance and tips and tricks for using these software are provided via the links provided in Table 13-3. It is intended to continually add content to the TUFLOW Wiki to provide users with guidance, tips and tricks on how to view and animate TUFLOW map output, so in the meantime if there is no guidance for your GIS/GUI, please contact support@tuflow.com and we will help you out.

Table 13-3 TUFLOW Map Output GIS and GUI Software Links

GIS/GUI	Link
GIS Software	
ArcGIS	TUFLOW Wiki ArcGIS Tips ArcGIS Information
MapInfo / Vertical Mapper	TUFLOW Wiki MapInfo Tips MapInfo Information
QGIS / Crayfish	TUFLOW Wiki QGIS Tips TUFLOW Wiki Crayfish Guidance QGIS Information
GUI Software	
12D Model	TUFLOW Wiki 12D Model Tips 12D Model Information
Blue Kenue	Please refer to Blue Kenue Software Site TUFLOW Wiki guidance is planned to be provided in the future
Ensight	TUFLOW Wiki Ensight Tips
FEWS	Please refer to Delft-FEWS Wiki TUFLOW Wiki guidance is planned to be provided in the future
Flood Modeller	Flood Modeller Information TUFLOW Wiki guidance is planned to be provided in the future
SMS	TUFLOW Wiki SMS Tips SMS Information
WaterRIDE	WaterRIDE Information TUFLOW Wiki guidance is planned to be provided in the future
XP2D	XP2D Information XP2D Homepage

13.6 Conversion to GIS Formats

The TUFLOW Utilities 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. Some of the more common functions include:

- Creating a .mif/.mid or .shp file of velocity vectors;
- Creating a grid of the maximum values from two or more input grids; and
- Calculating the affluxes between two grids

Refer to Chapter 15 and the TUFLOW Utilities Wiki Page for further information on the TUFLOW Utilities.

Note that as of the 2016-03 release Map Output Format may be used to output directly to GIS, ASC and/or FLT grid formats, eliminating the need to convert the map output results to GIS and grid formats at the completion of the model simulation. Refer to Section 9.6.4.1 for further information.

13.7 Impact Analysis Mapping

It is often necessary to carry out a comparison of results between two or more model simulations. For example, to quantify the impact of a proposed development on the predicted flood extent and flood depths or assess the sensitivity of the model due to changes in one of the parameters. A number of the TUFLOW utilities (refer to Section 15) may be used to carry out this comparison. For example, the ASC_to_ASC.exe utility may be used to compare ASC or FLT formatted grids. This utility outputs two grids:

- A difference grid subtracting the second input grid from the first; and
- A second grid with a "_wd" suffix, denoting locations where the model was previously wet and now dry and vice versa.

Alternatively, the RES_to_RES.exe may be used. Rather than comparing ASC or FLT formatted grids, the RES_to_RES.exe compares SMS result files (either in .dat or .xmdf format) for a single timestep or all timesteps in the model simulation. This utility unlike the ASC_to_ASC.exe, is limited to comparing model results with the same .2dm mesh file. A number of alternate methods for calculating flood afflux are outlined in Module 5 of the TUFLOW tutorial model.

Further information on ASC_to_ASC.exe, RES_to_RES.exe and the other TUFLOW utilities are discussed in Chapter 15, with more detail and examples via the TUFLOW Wiki Utilities page.

14 Quality Control

Chapter Contents

14 Quality Control 14-1

14.1 Introduction 14-2

14.2 Model Health 14-3

14.2.1 Healthy Model Indicators 14-3

14.2.2 Timestep 14-6

14.3 Troubleshooting 14-7

14.3.1 General Comments 14-7

14.3.2 Identifying the Start of an Instability 14-7

14.3.2.1 Tips for 2D Domains 14-8

14.3.2.2 Tips for 1D Domains 14-9

14.3.2.3 Tips for 1D/2D Links 14-10

14.3.3 Suggestions and Recommendations 14-11

14.4 Large Models (exceeding RAM) 14-13

14.4.1 Influence of TUFLOW Version 14-13

14.4.2 Influence of Model Design 14-14

14.4.3 Memory Usage Reporting 14-16

14.4.4 Temporary Memory Usage 14-17

14.4.5 GPU models and RAM requirements 14-17

14.5 Past Release Version Backward Compatibility 14-18

14.6 Check List 14-19

14.1 Introduction

Proficient and effective 1D and 2D hydrodynamic modelling is a skill that takes time 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, etc.). While in most cases the reasons for problems are due to the quality of 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 corrected 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.

14.2 Model Health

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 (> ±1 %) as indicated by the "CE" percentages displayed to the Console Window (see Section 12.1), and output to the various mass balance files as described in Section 12.11.
- "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 12.1.
- 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 the cause 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 GIS file, resolve the problems in order of occurrence (ie. in the order the messages appear in a MapInfo Browser Window, top to bottom).

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 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 can be an issue.

The underlying message is invest in good quality input data and experience! As the saying goes, **"garbage in, garbage out"**

14.2.1 Healthy Model Indicators

A summary of healthy model indicators is provided 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 12.4. The indicators are discussed in Table 14-1 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 14-1 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 location and time. 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 GIS 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 Section 14.3.2 for further discussion. Negative depths are often an indicator to the start of an instability.
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, 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. This behaviour is indicative of a boundary line that is not well aligned (perpendicular to flow) and possibly should be changed.
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. The time of peak water level is also a good indicator of this, if the time of peak water level is the same as the end time of the simulation, the simulation has likely not run long enough for the flood waters to peak.
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.

Column	Description
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: (Total Volume In - Total Volume Out) – (Volume at End – 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 significant problems with the model.
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 may have been unsteady at some point. 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 significant problems within the model.
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.**

14.2.2 Timestep

There is a tendency for hydraulic modellers to "solve" an instability by reducing the model 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 $^1/_2$ to $^1/_5$ 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 (>50m) or models with significant areas of deep water (>5m), the above rule-of-thumb may not apply with the timestep often being smaller. This is due to the Courant condition described in Section 3.4.

Using too small a timestep can mask fundamental problems in the input data, and hide mistakes in the construction of the model. For example, if the user accidently applies a topography modifier with an elevation -99m below the surrounding cells, a small timestep may "remove" the instability but does not resolve the issues in the input data.

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.

14.3 Troubleshooting

14.3.1 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 for the last timestep. Very large velocity vectors and/or excessively high or low water levels occur in the vicinity of the instability.

The instability can also be located using the **_messages.mif / .shp files** (see Section 12.5).

Always search the .tlf files for "UNSTABLE", "WARNING", "CHECK" and "ERROR". ERRORs stop the simulation, while WARNINGs and CHECKs do not. TUFLOW attempts 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 warnings, therefore, search through the .tlf files, or start at the beginning of the attributes within the _messages.mif / .shp file resolve the earlier errors first, rather than working backwards from the final error which resulted in the instability.

The TUFLOW Wiki contains a comprehensive database of TUFLOW check, warning, and error messages. A separate Wiki page has been set up for each message including a description and suggestion how to address the message.

14.3.2 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 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.

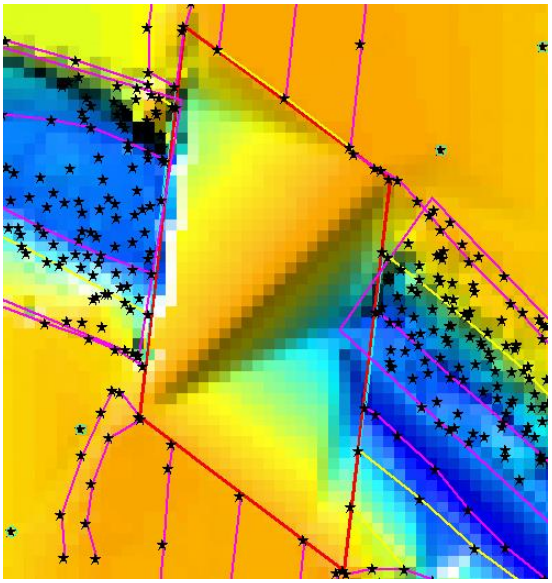
The WARNINGs are sent to the _messages.mif / .shp file (see Section 12.5). Import or open these into the GIS and they point directly at the location of the negative depth. If the number of these warnings are substantial (eg. if a model remains stable but with minor instabilities), select some of the first negative depth warnings in the attribute data 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.

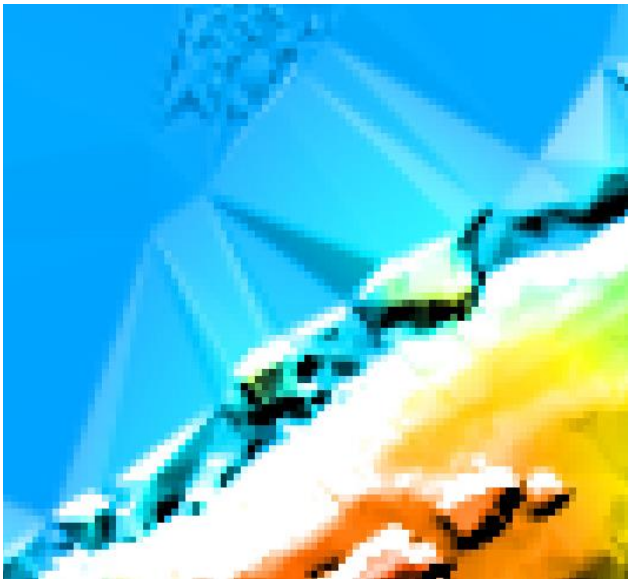
Also see the discussions on rectifying problematic models below as well as the commands Depth Limit Factor (1D domains) and Instability Water Level (2D domains).

14.3.2.1 Tips for 2D Domains

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 and MB2 outputs (see Map Output Data Types and Table 9-10) 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 identify and resolve the problem.

1. Set the 2D Timestep to somewhere between $^1/_2$ to $^1/_5$ of the cell size (in metres).
2. Run the model until it becomes unstable or has generated the WARNING 2991 messages.
3. Import or open the _messages GIS layer.
4. Within your chosen GIS package, view the _messages GIS layer in a tabular format 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. Select a few of these WARNINGs and locate these geographically in map window.
6. Using the various _check GIS layers, carry out some fundamental checks:

(i). 2d_zpt_check or _DEM_Z.flt: 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 the Zpts it is recommended to create a new 2d_zpt or 2d_zsh layer rather than editing the original data source for quality control purposes.

(ii). 2d_uvpt_check or 2d_grd_check: check the Manning's n values are as expected, if not identify why.

Other points to note for 2D domains are:

- 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.
- 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 TUFLOW (see Section 11.4). In addition, the default Cell Wet/Dry Depth may need to be lowered from 0.002m to 0.0002m (when using Units == US Customary the equivalent would be lowering from the default of 0.007 to 0.0007ft) due to the substantial amount of sheet flow.

14.3.2.2 Tips for 1D Domains

As advised in Section 14.3.2.1 for 2D domains, the location of the instability should be determined by investigating where the WARNING 1991 messages occur. 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 1991 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:

   (i) 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.

   (ii) One or more incorrect upstream and downstream bed levels. Import the 1d_inverts_check GIS file, label the Invert attribute, and cross-check that the inverts are correct.

   (iii) A very steep channel entering a gently sloping channel. If this is the real situation, then additional 1D channels giving a higher computational resolution may be needed, or inserting a structure at the transition may help.

   (iv) If there is a sudden change in 1D flow area, then a more appropriate and more stable approach would be to insert a structure representative of the situation to model the energy losses associated with the contraction and/or expansion of water causing.

   (v) 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.

   (vi) A common solution is to add additional storage to the node. This can be done by using the 1d_nwk Length_or_ANA attributeTable 5-11, 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 are 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 effect on the hydraulic results).

Besides investigating where the WARNING 1991 messages occur, the most effective approach is to thematically map or categorise the _TSMB GIS layer using the ME_Avg_Abs attribute. A MapInfo example of this is provided in the TUFLOW Wiki.

14.3.2.3 Tips for 1D/2D Links

Some common configuration mistakes 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. In these situations the 1D and 2D mass error reporting can appear satisfactory, however the overall model mass error is poor.

   The _TSMB1d2d GIS 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.

   HX connections are recommended along river banks for nested 1D channels.

2. Unrealistic flow patterns occurring across the HX lines, sometimes causing strange flow patterns and circulations in the 2D domain. This may be due to one of the following reasons:

(i). 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 the 2D domain along the HX line will not accurately reflect the real situation. This can create recirculation in the 2D and oscillations in the 1D. 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. Typically, areas where there are significant changes in longitudinal water surface gradient will require finer 1D resolution. Additional 1D nodes may be incorporated quickly and easily in the model through the use of automatically interpolated cross-sections (refer to Section 5.10.7).

(ii). Conversely, having too fine a spatial 1D resolution may cause stability problems due to insufficient nodal storage. As a rule-of-thumb, the node surface area should be similar to the width of the 2D/1D interface multiplied by 3 to 10 cell widths.

(iii). 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.

(iv). Similarly, at 1D structures where there is a significant drop in water level (for example, across a weir), 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.

3. Instabilities can occur across HX boundaries that are located in areas of ponding water due to the frequent transfer of water back and forth between the 1D and 2D domains. Use the 1D_2D_check layer to view the ZC elevations of the boundary, ensuring the elevations are appropriate and the boundaries have been digitised on the top of banks. In some situations the HX boundaries may need to be relocated and the schematisation of the model revisited to resolve the issue.

4. Bumpy topography in the approach to SX boundaries may lead to instabilities. Smooth out the Zpt elevations where appropriate, ensuring there is a 2D flow path leading to / departing from the boundary.

5. An inappropriate number of SX boundary cells in relation to the 1D structure width may cause stability problems. The number of cells selected should generally always correlate to the 1D structure width. For example, a 5m wide culvert should be connected to 2-3 SX boundary cells when the cell size is 2m wide. If the 1D structure width is less than or equal to a single cell width, two rather than one boundary cell may need to be selected to achieve stability.

14.3.3 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 12.2.1).

2. If the Console Window disappears for no apparent reason, first check the following:

   (i). 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 file is written by default).

   (ii). Your computer network is/was not down.

3. If you are experiencing instability water levels. Follow the advice given in Section 14.3.2.1 and 14.3.2.2. Otherwise, check the water level that is used for detecting instabilities (Instability Water Level). If every Z point elevation has not been allocated, the default elevation of 99999 will be assigned. Provided the default settings for the .tcf command Zpt Range Check have not been altered, TUFLOW will report an ERROR message. If there are 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. 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/.shp 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 support@tuflow.com.

4. If TUFLOW indicates that GIS objects are not snapped, not connected, could not be found or are outside the model domain, check that:

   (i) The most resent updates have been saved or exported for use by TUFLOW

(ii) The relevant GIS layers are in the correct projection and that the objects are snapped to each other. Some GIS programs can handle layers of different projections, however, TUFLOW requires that all layers be in the same projection. This projection must be a Cartesian projection (not degrees latitude/longitude) and is specified using MI Projection (.tcf file) and/or SHP Projection. The default setting is that all input layers are of the same projection otherwise an ERROR occurs (see GIS Projection Check).

(iii) Different GIS software may use a different precision in the coordinates, this may result in objects being snapped within the software but not in TUFLOW. The distance which objects are considered snapped in TUFLOW can be set with the Snap Tolerance command in TUFLOW.

5. If you are having stability problems, check that the computational timestep is appropriate (see Section 3.4 and Section 14.3.2).

6. Discontinuous initial water levels, particularly at 2D/1D interfaces are a common source of instability. If the model is going unstable near a 2D/ID interface shortly after starting, check that the initial water levels in the 1D and 2D domains are similar.

7. 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 is output.

8. Deep river bends with “bumpy” topography may cause instabilities in 2D models. Smoothing the topography, rather than reducing the timestep is recommended.

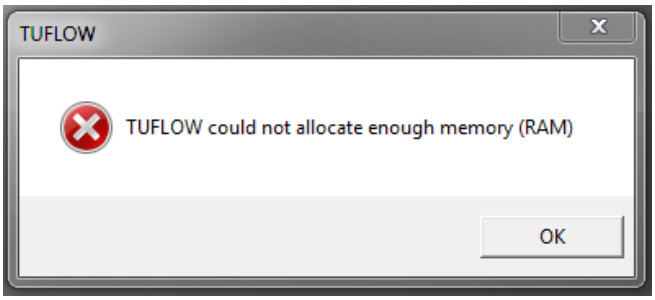
9. Under-sized 1D node storage (NA tables) connected to 2D HX boundaries may cause instabilities near the 2D/1D interface. Over-sized storage attenuates the inflow hydrograph. As a rule-of-thumb, the node surface area should be similar to the width of the 2D/1D interface multiplied by 3 to 10 cell widths.

10. Irregular topography just inside a 2D boundary may cause instabilities. If problems occur, smooth the rough topography.

11. It is preferable to use a timestep that divides neatly into 60 or 3600, ie. 1, 2, 3, 4, 5, 6, 10, 12, 15, 20, 30, 45, 60, etc in that the time-series and map output coincides with the timestep increments.

14.4 Large Models (exceeding RAM)

An issue that is encountered by TUFLOW modellers using computers with limited RAM (by today's standards!) is a warning stating that the software is unable to allocate enough RAM.



The error may be caused by a number of contributing factors, generally falling into one or more of the following categories:

- The available RAM or hardware specifications of the computer;
- If a 32 bit or 64 bit version of TUFLOW is used;
- The precision of TUFLOW used to simulate the model; and
- The features used or the design of the model.

The first check that should be made is the amount of available RAM on the computer. What is the computers specifications and what is already being used by other processes? This can be done in Windows Task Manager. Third party software used to build TUFLOW models and view their results (such as GIS packages and SMS) can consume large amounts of RAM leaving little remaining for the TUFLOW simulation.

14.4.1 Influence of TUFLOW Version

The choice of TUFLOW version can influence the amount of RAM needed to simulate the model. The double precision (DP) versions of TUFLOW utilise more RAM (up to twice as much) than the single (SP) precision versions as all real numbers are 8-byte reals rather than 4-byte reals. Refer to Section 11.4.2 for further information on the difference between single and double precision versions of TUFLOW.

Most 32bit versions of the Windows Operating System are limited to a total physical memory of 4GB of RAM. This does vary depending on the version on Microsoft Windows. For example Windows 7 Starter Edition is limited to 2GB, see the Microsoft website for more details. Windows also limits the maximum amount of memory a single process can utilise, for a 32 bit System this is 2GB

We have found that RAM issues may occur using the 32 bit version of TUFLOW on a 32 bit version of Windows when the RAM requirement exceeds 1.5GB. An identical 32 bit version of TUFLOW running on a 64 bit version of Microsoft Windows is typically able to access approximately 2.0GB of memory (as reported in the .tlf file). A 64 bit version of TUFLOW (which requires a 64 bit version of Windows) is limited by the amount of RAM available, of which the theoretical limit should by 8TB (note this is

RAM and not hard drive space). We are aware of TUFLOW simulations that have required greater than 40GB of RAM per simulation and have run successfully using the 64 bit version of TUFLOW.

TUFLOW has been compiled with the /LARGEADDRESSAWARE linker option, which means that on a 32 bit system if the /3GB boot switch has been enabled it can access more memory. Instructions for enabling this can be found on the TUFLOW forum. It is noted that not all Windows updates and drivers work well with this enabled and if issues are experienced this should be disabled (instructions in forum post above). If memory issues are encountered on a 32 bit version of TUFLOW moving to the 64 bit version is the preferred approach.

Note that the 64 bit versions of TUFLOW may only be run using a WIBU dongle and a computer with a 64 bit operating system. The SoftLok dongles are only able to run 32 bit versions of TUFLOW (refer to Section 11.5.1).

14.4.2 Influence of Model Design

The key factors that influence memory allocation are:

- The size of the redundant area around the perimeter of the model;
- The number of 2D cells, hence the domain extent and the cell size; and
- The model features utilised.

The redundant area is defined as the difference between the 2D domain size and the active model area. As discussed in Section 6.7, TUFLOW automatically strips any redundant rows/columns around the active area of the model to reduce simulation times this however does not reduce the amount of RAM consumed by the model. Refer to Sections 6.5 and 6.7 for further information on defining the 2D domain and the active/inactive areas of a model.

Figure 14-1 presents two different versions of the TUFLOW Tutorial model, where the only difference is the size of the 2D domain. The 2d_dom check file of both models has been overlaid on top of the active area of the model defined by a magenta polygon with CD=1 in a 2d_code GIS layer.

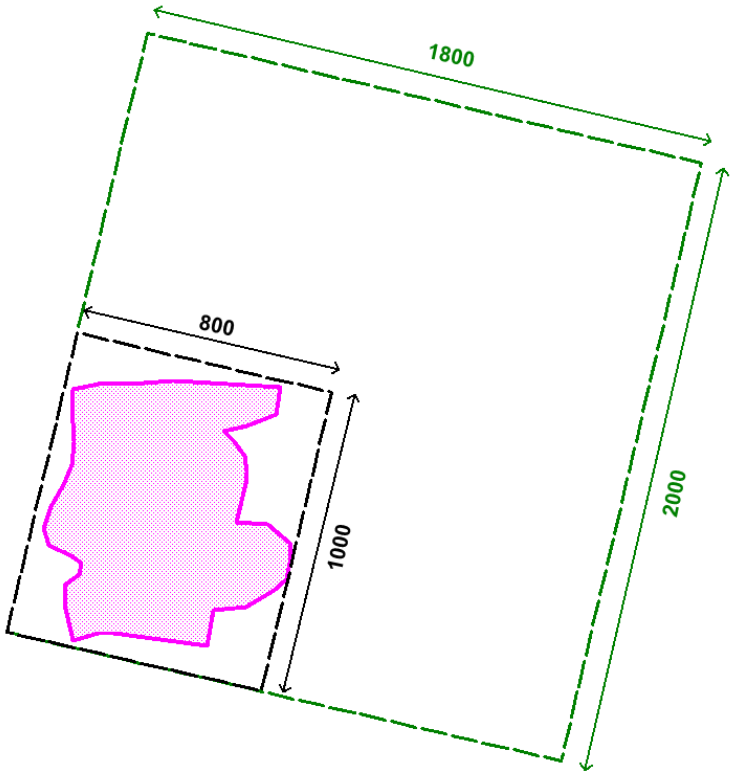


Figure 14-1 Influence of 2D Domain Size on RAM Allocation

The green coloured 2d_dom_check in Figure 14-1 has been defined using the .tgc command Grid Size (X,Y) == 1800, 2000. The 2D domain extent is shown to be significantly larger than the active area of the model (the magenta polygon).

The size of the redundant area may be determined by searching within the .tlf file for "redundant". The tlf file for this example provides the following information on the redundant area.

```
Isolating redundant perimeter sections of 2D domain Domain_001...
...Reduced computational grid by 79%.  Now extends from [5,7] to [196,169].
...Reduced output grid by 79%.  Now extends from [5,7] to [196,169].
```

It indicates TUFLOW has reduced the computation grid by 79%, meaning the active area is only 21% of the size of the 2D domain is required for the simulation. A search for "memory" in the .tlf file shows the memory allocated to the model:

```
Memory requested for 2D domain Domain_001 = 101 Mb
Total Memory requested thus far = 104 Mb
```

In the second example, the size of the 2D domain extent has been reduced by specifying Grid Size (X,Y) == 800, 1000. The black coloured 2d_dom_check in Figure 14-1 shows this is a better match in size to the active area of the model. This is reflected in the information reported in the tlf:

```
Isolating redundant perimeter sections of 2D domain Domain_001...
...Reduced computational grid by 8%.  Now extends from [5,7] to [196,169].
```

```
...Reduced output grid by 8%.  Now extends from [5,7] to [196,169].
```

The smaller 2D domain also has a noticeable impact on the memory required to run the model, in this case, reducing it by a factor of 5.

```
Memory requested for 2D domain Domain_001 = 23 Mb
Total Memory requested thus far = 27 Mb
```

The 2D cell size, dictates the total number of cells within a given study area, keeping in mind that halving the cell size quadruples the number of cells. The number of cells has a direct correlation to the amount of RAM required to run the model. For each cell, TUFLOW stores a number of variables including water level and velocity components. The greater the number of cells, the greater the amount of information stored and therefore the greater the amount of memory needed to simulate the model. Simulating the model with a variety of cell sizes during the initial development stage, will provide a better understanding of the impact the cell size has on simulation time, memory allocation and model results. Refer also to Sections 3.3.1 and 3.5.

A number of optional TUFLOW features may also increase the memory requirements. Common features include Model Output Zones (Section 9.4.3) and the ASC or FLT grid Map Output Format (Section 9.6.4.1).

14.4.3 Memory Usage Reporting

The TUFLOW log file (.tlf) provides a summary of the model's memory usage, as per below. The memory usage information can be used to identify which items of a model require the most RAM.

```
1D QX and HX BCs automatically detected
1D Geometry data memory ........          18 Mb
1D Boundary data memory ........           0 Mb
1D yQ Curves memory ............           0 Mb
1D Control structures memory ...           0 Mb
1D Mesh data memory ............           3 Mb
1D Results storage memory ......          24 Mb
1D Temporary data memory .......           8 Mb
1D Computational arrays memory .           0 Mb
Total 1D domain memory (RAM) requested ............        54 Mb
Total 1D domain character memory (RAM) requested ..         7 Mb

Domain_001  : Allocating Memory Pointers:
Domain_001  : Grid data memory ...............         47 Mb
Domain_001  : Variable Z and Layered FC memory          0 Mb
Domain_001  : Sub-domain linking memory ......          4 Mb
Domain_001  : Flow constrictions memory ......         16 Mb
Domain_001  : Weirs and viscosity factor memory        14 Mb
Domain_001  : Pressure, wind and waves memory           0 Mb
Domain_001  : General Memory .................         14 Mb
Domain_001  : Boundary conditions memory .....          7 Mb
Domain_001  : Wind & waves boundary memory ...          0 Mb
Domain_001  : Work arrays memory .............        115 Mb
Domain_001  : SMS High Res format memory .....          0 Mb
```

14.4.4 Temporary Memory Usage

A number of input steps may require additional memory for processing during model initialisation. For example, when a large DEM is input (Read Grid Zpts) this is read into memory and then processed. This allows for a very rapid processing of the file, though may consume significant additional memory. The DEM can be tiled to reduce memory usage at start-up. Other files that consume additional memory are (Read GIS Z Shape) as these track the modified elevation and the change in elevation for all Zpts.

The Maximum Vertices and Maximum Points commands can be used to increase or decrease the size of a single GIS object and will change the memory required.

14.4.5 GPU models and RAM requirements

When running the TUFLOW GPU Solver, the TUFLOW.exe carries out all the pre and post-processing of the data (setting up the model and storing and tracking of all output data). These tasks consume significantly more memory than that required to carry out the hydraulic computations, therefore, although the hydrodynamic calculations are carried out on the GPU cards, significant amounts of CPU RAM are also required. The simulation also consumes 100% of a CPU core as the CPU and GPU are in regular communication.

The 2016-03 release has optimised the CPU RAM requirements for GPU Solver simulations by deactivating memory not used by the GPU Solver, however, significantly more CPU RAM is required than GPU memory mostly due to the CPU carrying out all of the pre-processing to set the model up and for storing and outputting results as mentioned above. The optimisation of the CPU RAM can be switched off using GPU RAM Optimisation. Testing between the 2013-12 and 2016-03 releases indicates the 2016-03 release uses 33% less CPU RAM than the 2013-12 release, while the GPU memory usage has increased by around 10% for the 2016-03 release due to the addition of new GPU features. As a guide, for the 2016-03 release the CPU RAM requirement is around 4 to 5 times that of the GPU memory (with grid based output active and set to the defaults).

Of note is that a reasonable amount of CPU RAM is consumed if using the grid output of results (usually a requirement for GPU Solver models due to the size of the models, as mesh based output is unmanageable in viewing platforms). This is because grid output can be on a different resolution and orientation than the model's 2D computational grid, and the default resolution is half the 2D cell size. Therefore, the memory requirement for this output can be significantly reduced by increasing the Grid Output Cell Size.

An option that directly outputs on the same resolution as the 2D grid is being considered for future releases. If this is the only map output option specified, this would negate the need to have a triangular mesh to interpolate results from and also remove the need to store in memory grid interpolation factors.

14.5 Past Release Version Backward Compatibility

New and improved methods may inadvertently cause slight differences in results. Alternatively, complications in the code when making improvements and even vagaries of the source code compiler can also cause result differences.

During the life of TUFLOW, since 1989, every effort has been made to provide full backward compatibility. Models developed in 1989 can still be simulated to achieve the same results. Old file formats are still typically recognised or can be translated into more recent formats. There have, however, been numerous enhancements, improvements and the inevitable bug fixes.

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 .tlf log files so that it is clear what version of the software was used to simulate the model. Chapter 16 provides links to all release notes listing changes between different TUFLOW builds.

The TUFLOW Version Backward Compatibility Page documents those changes that may cause a change in model results going back to the 2006-06 TUFLOW release. If a legacy or old model is being upgraded to the latest TUFLOW release, it is recommended that you familiarise yourself with possible changes to the default settings that may change results and make the necessary changes as appropriate. In nearly all cases a backward compatibility switch (see Defaults) is provided so that new builds can reproduce past builds results although this is not always possible.

14.6 Check List

Table 14-2 presents a generalised list to help guide reviewers and modellers when carrying out quality control checks on a model. This list is not exhaustive, and experienced modellers who know what to look for must carry out the reviews at all times.

Table 14-2 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; • Observations from simulations; • Key modelling assumptions; and • A list of data sources used. 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 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 3.3.1), 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.	

Item	Description	Checked
	Regarding the latter, this is effectively carried out by reviewing the _DEM_Z.flt check file (see Table 12-2). Note: Reviewing the elevations in the .2dm file is not appropriate as only the ZH Zpt is represented in the .2dm file (the 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). Are hydraulic controls such as levees, roads and embankments represented in the model?	
Bed Resistance Values	Bed resistance values are to be reviewed by an experienced modeller. The review should focus on checking: • The DEM_M check file; • The Manning_n attribute values in the uvpt_check file layer created by Write Check Files; • the material values in the .2dm (SMS 2D mesh) file; • The Material attribute values in the grd_check file 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 12.11), it is good practice to carry out independent mass checks. Standard practice is to place 2d_po flow lines (see Section 10.7.1) in several locations throughout 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	

Item	Description	Checked
	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 leaving the model.	
Hydraulic Structures	Head losses through a structure need to be validated through: • Calibration to recorded information (if available). • 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 what is expected (see Section 6.12). 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 3.6).	

15 Utilities

Chapter Contents

15 Utilities 15-1

15.1 Introduction 15-2

15.2 Console Utilities 15-3

15.2.1 TUFLOW_to_GIS 15-3

15.2.2 ASC_to_ASC 15-4

15.2.3 RES_to_RES 15-5

15.2.4 Convert_to_TS1 15-5

15.2.5 TIN_to_TIN 15-6

15.2.6 xsGenerator 15-7

15.2.7 12da_to_from_gis 15-7

15.3 Excel Utilities 15-9

15.4 GIS Based Utilities 15-10

15.4.1 MiTools 15-10

15.4.2 TuPlot QGIS Plugin 15-10

15.4.3 ArcGIS Toolbox 15-11

15.1 Introduction

The TUFLOW Utilities are a set of tools that can be used to convert input data for use in a TUFLOW model, or to process / manipulate the raw 2D result files produced from a model simulation. Many of the utilities are DOS executables similar to the TUFLOW engine, however some are macros for use in Excel or python scripts. GIS based utilities have also been developed for MapInfo, QGIS, and ArcGIS.

This chapter of the manual describes each of the utilities and in the case of the TUFLOW DOS Utilities, provides some examples of their most common uses. The TUFLOW Wiki provides a comprehensive list of available options for each DOS utility along with further examples.

15.2 Console Utilities

The TUFLOW Console Utilities are like the TUFLOW engine in that they are command window executables with no user interface. They are available for download at no cost from the TUFLOW website.

They can be run using either of the following methods:

- Customising the right mouse button: This method is limited to those operations that only require one file as an input. Instructions for setting this up are provided on this TUFLOW Wiki Page.
- Creating a batch file: This method provides the opportunity to carry out bulk processing of data and for operations that refer to more than one input file.

Double-clicking on each TUFLOW Utility will open up a DOS Console Window displaying instructions for its use, available options and syntax.

15.2.1 TUFLOW_to_GIS

The TUFLOW_to_GIS.exe utility provides a variety of functions that converts TUFLOW output into GIS and other formats. The primary ones are:

- Convert TUFLOW .dat or .xmdf files to:
  - 3D surface files in ASCII Grid or FLT binary formats. 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 or .shp files of the .dat or .xmdf 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 or .xmdf files (eg. velocity and unit flow) in .mif or .shp file format.
- Interrogate the TUFLOW results to add modelled data to a calibration point GIS data layer in .mif or .shp 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.

TUFLOW_to_GIS.exe examples have been provided below. The full list of available options and further examples are provided in this TUFLOW Wiki Page.

Example 1

```
TUFLOW_to_GIS.exe -asc -max my_model_h.dat
TUFLOW_to_GIS.exe -asc -max -typeh my_model.xmdf
```

Creates an ASC grid of the maximum water levels in my_model_h.dat / my_model.xmdf. For ASC or FLT output, the default resolution of the grid will be half the 2D cell size. This may be manually specified by the user by including the –grid<dist> field in the above. For example to specify a resolution

of 2m, add –grid2. Note that TUFLOW is able to directly output results in gridded formats (.asc and .flt), avoiding the need for a separate conversion step. Refer to Section 9.6.4.1.

Example 2

```
TUFLOW_to_GIS.exe -mif -vector -sf0.5 -t2 my_model_V.dat
TUFLOW_to_GIS.exe -shp -vector -sf0.5 -t2 -typeV my_model.xmdf
```

Creates a.mid/.mif or .shp vector file for output time 2. The vectors are stored as polygon objects and output at the cell corners. The 'sf' (scale factor) switch scales the size of the vectors in relation to the 2D cell size. A sf of 0.5 scales the length of vectors of 1m/s for _V.dat files or $1m^3/s$ for _q.dat files to half the 2D cell size. The default is a sf of 1.

15.2.2 ASC_to_ASC

The ASC_to_ASC.exe utility performs a range of functions on gridded files of either ASC or FLT formats or a combination of both. Unlike the RES_to_RES.exe utility (see Section 15.2.3), this utility can be used to compare or process grids from different .2dm meshes. The primary functions of this utility are:

- Produce a difference grid between two input files;
- Convert an ASC file to an FLT file and vice versa;
- Produce a grid determining the maximum value of all input grids; and/or
- Extract elevations from a DEM for breaklines in the 2d_zsh format.

Some ASC_to_ASC.exe examples are presented below. The full list of available switches and further examples are provided at this TUFLOW Wiki Page.

Example 1

```
asc_to_asc.exe -dif Q100_dev_impact_h.asc Q100_dev_h.asc Q100_exg_h.asc
```

Outputs an ASC file called Q100_dev_impact_h.asc that contains the difference of Q100_dev_h minus Q100_exg_h.

Example 2

```
asc_to_asc.exe -max Q100_30min_h.flt Q100_45min_h.flt Q100_60min_h.flt
```

Outputs two .flt grids: a numerical grid containing the maximum value of all input grids, as well as a classified grid with the name of the input grid in which the maximum value was obtained.

Example 3

```
asc_to_asc.exe -brkline 2d_zsh_breaklines_L.shp DEM.asc
```

Extracts elevations from DEM.asc and populates the 'Z' attribute of polylines within 2d_zsh_breaklines_L.shp.

15.2.3 RES_to_RES

The RES_to_RES.exe utility has a range of options to post-process TUFLOW result files in .dat or .xmdf format. Its primary functions are:

- Calculate the difference between two datasets;
- Calculate the maximum from a number of input datasets;
- Convert a .dat file to .xmdf format and vice versa; and/or
- Calculate the time taken for the results to increase by a specified cut-off depth.

Some examples of using RES_to_RES.exe are presented below. The full list of available switches and further examples are provided in this TUFLOW Wiki Page.

Example 1

```
res_to_res.exe -dif -typeH results_developed.xmdf results_existing.xmdf
```

Compares two input datasets and outputs a third grid containing the difference (the first results file minus the second). Optional flags include –out to specify the name of the output file, –t to specify the time at which to carry out the difference (rather than at all times), and –nowetdry to supress a wet/dry check using the water level results.

Example 2

```
res_to_res.exe -max -t99999 results_1_h.dat results_2_h.dat results_3_h.dat
```

Extracts the maximum water level from the three input .dat files. If a maximum folder exists in an .xmdf file for the type specified, this dataset will be used, for dat files use –t99999 to call the maximum data in the .dat file.

Example 3

```
res_to_res.exe -toi0.2 -typeD results1.xmdf
```

Calculates the time taken for the results to increase by the cut-off. A value of 0.2 is specified in the above example, and will output results containing the time required for the depth to increase by 0.2m. If there are depth values greater than 0.2 initially (due to initial water level conditions), the output is the time for the depth to increase by 0.2m. For dry areas it is the time for the depth to reach 0.2m.

15.2.4 Convert_to_TS1

The Convert_to_TS1.exe utility 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. 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" reports 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.

Some Convert_to_TS1.exe examples are presented below. The full list of available options and further examples are provided in this TUFLOW Wiki Page.

Example 1

```
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.

Example 2

```
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.

Example 3

```
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.

Example 4

```
convert_to_ts1.exe -rafts -csv Q100.tot Q050.tot Q100.loc Q050.loc
```

Outputs .csv files for the four files specified.

15.2.5 TIN_to_TIN

The TIN_to_TIN.exe utility 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.

Some TIN_to_TIN.exe examples are presented below. The full list of available options and further examples are documented in this TUFLOW Wiki Page.

Example 1

```
tin_to_tin.exe -12d "My DTM.tin"
```

Converts the SMS TIN "My DTM.tin" to a 12D .12da TIN. The created .12da file will be named "My DTM.tin.12da".

In addition to the -sms, -12d or -tri flag, specify -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.

Note, the extension of the input TIN file determines the format.

15.2.6 xsGenerator

The xsGenerator.exe utility 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 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).

xsGenerator can also be used to generate a cross-section database using survey data in the Flood Modeller format (refer to the -isis option). The first file specified must be a .mif file in the correct projection, with subsequent files being the survey files. Wildcards can also be used to input multiple files (eg. *.xyz).

Some examples of using xsGenerator.exe are presented below. The full list of available options and further examples are documented in this TUFLOW Wiki Page.

Example 1

```
xsGenerator.exe -M XS_survey.mif
```

Creates a 1d_xs layer and .csv files for each cross-section.

Example 2

```
xsGenerator.exe -M -isis Projection.mif  *.xyz
```

Creates a 1d_xs layer and .csv files based on a group of survey files in Flood Modeller format.

15.2.7 12da_to_from_gis

This TUFLOW utility converts .12da files from the 12D software (www.12d.com) to and from .mif and .shp files.

The utility is particularly useful for creating TUFLOW compatible inputs from 12D files. For example:

- By default, when converting from a .12da file to a .mif or .shp file without any additional options, 12da_to_from_gis.exe automatically creates a .mif or .shp file suitable for use with the command Read GIS Z Line. This is useful for importing 3D breaklines (eg. of a road design) directly into TUFLOW.

- The -xs option can be used to generate a TUFLOW 1D cross-section database from a 12D DTM. This approach is preferable to extracting cross-sections manually or 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.

Some examples of using 12da_to_from_GIS.exe are presented below. The full list of available options and further examples are documented in this TUFLOW Wiki Page.

Example 1

```
12da_to_from_gis.exe -mif 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.

Example 2

```
12da_to_from_gis.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

Example 3

```
12da_to_from_gis.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 specifying the following in the .tgc file:

```
Read GIS Z Line THICK == mi\2d_hx_lines_draped.12da.mif
```

15.3 Excel Utilities

A Microsoft Excel macro has been developed and is available to download at no cost from the TUFLOW Utilities downloads page. Instructions for installation and further information is provided in this TUFLOW Wiki Page. The macro performs the following functions:

- Plots 1D time-series results in .csv format (refer to Section 13.2.2);
- Plots 2D PO (refer to Section 9.3.3) results from a _PO.csv file;
- Exports the active chart to .jpg format;
- Exports the active worksheet to .csv format; and
- Exports the entire workbook to .csv format.

15.4 GIS Based Utilities

Utilities for MapInfo, QGIS, and ArcGIS have been created and are described in the following section of the manual. All GIS based Utilities provide tools to streamline the model build process. In addition, the MiTools and TuPlot QGIS Plugin, contain tools to visual 1D results.

15.4.1 MiTools

The MapInfo and TUFLOW Productivity Utilities (miTools) have been developed specifically to improve the efficiency of setting up and reviewing TUFLOW models, as well as improving the day to day ease of using MapInfo Professional.

There are many available functions with the MiTools Utilities. Some of these are listed below:

- Import and export of single or multiple MIF files;
- Increment a selected layer by creating a copy, assigning a new revision number, and closing the original layer;
- Automation of digitising cross-section (1d_xs), water level lines (1d_WLL), and connection lines (2d_bc type CN) based on the 1d_nwk layer;
- Graphing of selected cross-sectional data from a .csv file with or without the maximum water level from a _1d_mmH result file; and
- Graphing of time-series results for one or more objects within the _TS layer.

MiTools may be downloaded from tuflow.com, however note that use of the tools requires a license in addition to the one purchased to simulate TUFLOW. Contact sales@tuflow.com for further information. For further information or to arrange for a free 30 day evaluation license, please contact mitools@tuflow.com.

15.4.2 TuPlot QGIS Plugin

The TUFLOW TuPlot QGIS Plugin provides tools to improve the efficiency of setting up, running and viewing the results of TUFLOW models. There is no cost associated with using TuPlot. Some of the available functions are described below:

- Automated methods to create the TUFLOW folder directory and generate empty files;
- Incrementing the active layer by creating a copy, assigning a new revision number, and closing the original layer;
- Plot 1D time-series results and view long-section profiles; and
- Start a TUFLOW simulation from within QGIS.

The TuPlot QGIS Plugin is available as a free download from the TUFLOW Wiki. Installation instructions and further details on the tools available are also described.

15.4.3 ArcGIS Toolbox

The ArcGIS Toolbox is available for ArcMap version 10.1 and newer. The toolbox helps with streamlining the process of creating and editing a TUFLOW model in ArcMap. Some of the available functions are described below:

- Automated method to generate TUFLOW empty files;
- Import TUFLOW empty files and create a new .shp file;
- Start a TUFLOW simulation from within ArcMap; and
- Load all input .shp files from a model simulation into a common ArcGIS Map Window.

The ArcGIS Toolbox is available as a free download from tuflow.com.

16 New (and Past) Features and Changes

New features, enhancements, bug fixes and other changes to each new TUFLOW Build that is released via www.tuflow.com are provided in release notes issued for each new release. The release notes contain a description of the changes for each build issued for that TUFLOW release.

Releases are denoted by a year and month, for example, 2016-03, that represents the date when it was first issued. Builds for each release are denoted by two letters starting at AA, therefore the first build for the 2016-03 release is called Build 2016-03-AA. Subsequent builds are issued should there be any new minor features / enhancements and/or bug fixes. These will be denoted AB, AC, …. The notes for TUFLOW releases since 2005 can be accessed via the links below.

TUFLOW 2016-03 Release Notes

TUFLOW 2013-12 Release Notes

TUFLOW 2011-09 and 2012-05 Release Notes

TUFLOW 2010-10 Release Notes

TUFLOW 2009-07 Release Notes

TUFLOW 2008-08 Release Notes

TUFLOW 2007-07 Release Notes

TUFLOW 2006-06 Release Notes

TUFLOW 2005-05 Release Notes

17 References

Aust Roads. (1994) *Waterway Design - A Guide to the Hydraulic Design of Bridges, Culverts and Floodways*

Australian Emergency Management Institute (2014) *Australian Emergency Management Handbook 7: Managing the Floodplain Best Practice in Flood Risk Management in Australia*

Babister, M. and Barton, C.L. (2012) *Two Dimensional Modelling in Urban and Rural Floodplains Stage 1 & 2 Report.* Engineers Australia Australian Rainfall and Runoff Revision Projects Project 15: P15/S1/009 November 2012

Barton, C.L. (2001) *Flow Through an Abrupt Constriction – 2D Hydrodynamic Model Performance and Influence of Spatial Resolution.* Cathie Louise Barton, M.Eng.Sc Master Thesis, University of Queensland

Benham, S. and Rogencamp, G. (2003) *Application of 2D Flood Models with 1D Drainage Elements* NSW FMA Conference, Forbes, 2003

Bos, M.G. (1976) *Discharge Measurement Structures* Publication No. 161, Delft Hydraulic Laboratory, Delft, The Netherlands

Bos, M.G. (1989) *Discharge Measurement Structures* 3rd revised edition. International Institute for Land Reclamation and Improvement, Wageningen, the Netherlands. ILRI Publication 20

Boyte, C. (2014) *The Application of Direct Rainfall Models as Hydrologic Models Incorporating Hydraulic Resistance at Shallow Depths* BEng Thesis, The University of Queensland

Bradley J.N. (1978) *Hydraulics of Bridge Waterways* HDS 1, FHWA, Bridge Division

Cox, R.J., Shard, T.D., and Blacka, M. J. (2010) *Australian Rainfall and Runoff Revision Project 10: Appropriate Safety Criteria for People* Australian Institute of Engineers

CSIRO (2000) *Floodplain Management in Australia Best Practice Principles and Guidelines* SCARM Report 73." CSIRO Publishing, Commonwealth of Australia

DHI (2009) *MOUSE PIPE FLOW – Reference Manual* Danish Hydraulic Institute, 2009

Environment Agency (2010) *The Fluvial Design Guide* United Kingdom Environment Agency.

GHD (2011) *MBRC Storm Tide Management Study: Stage 1 – Scoping Report* Prepared for Moreton Bay Regional Council, 18 February 2011

Henderson, F.M. (1966) *Open Channel Flow* Macmillan, 1966

Huxley, C.D. (2004) *TUFLOW testing and Validation* Christopher Dylan Huxley, B.ENG. Thesis, Griffith University

Miller, D.S. (1994) *Discharge Characteristics* IAHR Hydraulic Structures Design Manual No.8, Hydraulic Design Considerations, Balkema Publ., Rotterdam, The Netherlands, 249 pages

Morrison, W.R.B, Smith, P.A. (1978) *A Practical Application of a Network Model Numerical Simulation of Fluid Motion* North Holland Publishing Company, Amsterdam, 1978

Neelz, S. and Pender, G. (2013) *Benchmarking the Latest Generation of 2D Hydraulic Modelling Packages* United Kingdom Environment Agency, SC120002/R

NSW Government (2005) *Floodplain Management Manual* NSW Department of Land and Water Conservation (NSW Government)

Rawls, W, J, Brakesiek & Miller, N, (1983) *Green-Ampt Infiltration Parameters from Soils Data* Journal of Hydraulic Engineering, vol 109, 62-71

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., 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. (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. (2001a) *Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme* The Institution of Engineers, Australia Conference on Hydraulics in Civil Engineering Hobart 28 –30 November 2001

Syme, W.J. (2001b) *TUFLOW – Two & One-dimensional Unsteady FLOW Software for Rivers, Estuaries and Coastal Waters* The Institution of Engineers, Australia 2D Seminar Sydney February 2001

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

Tullis, B. and Robinson, S. (2008) *Quantifying Culvert Exit Loss* Journal of Irrigation and Drainage Engineering , April 2008, Vol. 134, No. 2 : pp. 263-266

USACE (1987) *Hydraulic Design Criteria – Sheets 111-1 to 111-14* US Army Corp of Engineers

USBR (1987) *Design of Small Dams* U.S. Dept. of the Interior, Bureau of Reclamation, 1987 - Technology & Engineering

Appendix A .tcf File Commands

Adjust Head at ESTRY Interface
Apply Wave Radiation Stresses
Apply Wind Stresses

BC Control File
BC Database
BC Event Name
BC Event Source
BC Event Text
BC Wet/Dry Method
BC Zero Flow
Bed Resistance Cell Sides
Bed Resistance Depth Interpolation
Bed Resistance Values
Boundary Viscosity Factor
BSS Cutoff Depth

Calibration Points MI File
Cell Wet/Dry Depth
Cell Size
Change Zero Material Values to One
Check Inside Grid
Check MI Save Date
Check MI Save Ext
Command Line Processing
CSV Header Line
CSV Maximum Number Columns
CSV Time

Defaults
Define Event
Define Output Zone
Demo Model
Density of Air
Density of Water

Depth/Ripple Height Factor Limit
Display Water Level
Distribute HX Flows

End 1D Domain
End 2D Domain
End After Maximum
End Define
End Map Output
End Time
ESTRY Control File
Event File
Excel Start Date
Extrapolate Heads at Flow Boundaries

First Sweep Direction
Force File IO Display
Free Overfall
Free Overfall Factor
Froude Check
Froude Depth Adjustment

GA Convergence Value
GA Maximum Iterations
Geometry Control File
GIS Format
GIS Grid Vector Type
GIS Grid Vector Direction
GIS Grid Vector SF
GIS Grid Vector TTF
GIS Projection Check
GIS Supported Object Ignored
GIS Unsupported Object
Global FC Ch Factor
Global Weir Factor
GPU Device IDs
GPU DP Check
GPU RAM Optimisation

GPU Solver
GPU Temporal Scheme
Grid Format
Grid Output Cell Size
Grid Output Origin

HQ Boundary Approach
HX Force Weir Equation
HX ZC Check

If Event
If Scenario
Input Drive
Inside Region
Instability Water Level

Latitude
Layered FLC Default Approach
Line Cell Selection
Link 2D2D Adjust Velocity Head Factor
Link 2D2D Approach
Link 2D2D Distribute Flow
Link 2D2D Global Stability Factor
Log Folder

Map Cutoff Depth
Map Output Corner Interpolation
Map Output Data Types
Map Output Entire Model
Map Output Format
Map Output Interval
Mass Balance Output
Mass Balance Output Interval
Maximum 1D Channels
Maximum Courant Number
Maximum Points

Maximum Velocity Cutoff Depth
Maximum Vertices
Maximums and Minimums
Maximums and Minimums Only for Grids
Maximums and Minimums Time Series
Maximums Approach
Meshparts
MI Projection
Model Events
Model Output Zones
Model Platform
Model Precision
Model Scenarios
Model TUFLOW Build
Model TUFLOW Release

Negative Depth Approach
NetCDF Output Compression
NetCDF Output Start Date
NetCDF Output Time Unit
NetCDF Output Direction
NetCDF Output Format
Null Cell Checks
Number Iterations
Number 2D2D Link Iterations

Output Approach
Output Drive
Output Files
Output Folder

PO Approach
Pause

Rainfall Boundaries
Rainfall Gauges
Recalculate Chezy Interval
Read File
Read GIS Cyclone/Hurricane
Read GIS FC
Read GIS GLO
Read GIS IWL
Read GIS LP
Read GIS Output Zone
Read GIS PO
Read GIS Reporting Location
Read GIS X1D Network
Read GIS X1D Nodes
Read GIS X1D WLL
Read GIS X1D WLL Points
Read Grid IWL
Read Grid RF
Read Materials File
Read Restart File
Read RowCol IWL
Read Soils File
Reveal 1D Nodes

SA Minimum Depth
SA Proportion to Depth
Screen/Log Display Interval
Set IWL
Set Route Cut Off Type
Set Route Cut Off Values
Set Variable
Snap Tolerance
SHP Projection
Simulations Log Folder
Start 1D Domain
Start 2D Domain
Start Map Output
Start Time
Start Time Series Output
Supercritical
SX Flow Distribution Cutoff Depth
SX Head Adjustment
SX ZC Check

Time Series Output Format
Time Series Output Interval
Time Output Cutoff
Timestep
Timestep Maximum Increase
TSF Update Interval
Tutorial Model

UK Hazard Debris Factor
UK Hazard Formula
UK Hazard Land Use
Units
Unused HX and SX Connections
Use Forward Slash

Verbose
Viscosity Coefficient
Viscosity Formulation
VG Z Adjustment

Water Level Checks
Wetting and Drying
Wind/Wave Shallow Depths
Write Check Files
Write Empty GIS Files
Write PO Online
Write Restart File at Time
Write Restart File Interval
Write Restart File Version
Write Restart Filename

Write X1D Check Files

XF Files
XF Files Include in Filename
XMDF Output Compression

Zero Negative Depths
Zero Rainfall Check
ZP Hazard Cutoff Depth
Zpt Range Check

Adjust Head at ESTRY Interface == [ON | ON VARIABLE | {OFF}]

(Optional)

This command's main use is to provide backward compatibility for older models (pre Build 2006-03-AB) 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 7-5). 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 flow path due to the dynamic head). Based on numerous and wide ranging application of HX lines, it is recommended 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.

Apply Wave Radiation Stresses == [ON | {OFF}]

(Optional)

Unsupported feature – yet to be set up and tested on PC version.

Apply Wind Stresses == [ON | ON VARIABLE | {OFF}]

(Optional)

Unsupported feature – yet to be set up and tested on PC version.

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 Appendix D). There can only be one .tbc file per 2D domain.

BC Database == <.csv_file>

(Mandatory)

Sets the active BC Database file as described in Section 7.5. 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 7.5.2 for a description of how the BC event commands operate and refer to the command BC Event Source.

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 11.3.1, 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 7.5.2 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 Wet/Dry Method == [PRE 2005-11-AF]

(Optional)

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.

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.

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.

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.

The AVERAGE M option, 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.

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). The default value is SPLINE n which uses a curved fit so that the n values transition gradually. LINEAR M and LINEAR n both use a linear interpolation of the M (1/n) and n value respectively.

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.

Bed Resistance Values == [{MANNING N} | MANNING M | CHEZY]

(Optional)

Sets the bed resistance formula to use. The default value is MANNING N.

Boundary Viscosity Factor == [{1.0} | <factor>]

(Optional)

Multiplies the eddy viscosity coefficient by <factor> along all open (external) boundaries, and 2D and HX links.

For releases prior to 2016-03 the eddy viscosity coefficient was previously set to zero for the boundary cells (this was because land boundaries required this and open boundaries were treated in the same manner). For the 2013-12 release for Link 2D2D Approach == METHOD D the default is set to 1.0 to improve the performance in flow patterns along the 2D link lines. Changing this value in the range of 0.0 to 5.0 (possibly higher) usually has little effect on results, however, increasing the factor may help "stabilise" unrealistic circulations along a boundary or 2D / HX link line without adversely affecting results. Sensitivity test prior to adopting larger factors.

BSS Cutoff Depth == [{0.1} | <BSS_cutoff_depth>]

(Optional)

Defines the depth threshold (m) below which the Bed Shear Stress (BSS) and Stream Power (SP) Map Output Data Types will linearly reduce to zero.

Calibration Points MI File == <mif_layer>

(Optional)

Assigns the peak water level calculated during the simulation as an extra attribute to the .mif/.mid layer. 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.

The GIS layer at present must be in the .mif/.mid format and is opened and closed during the start-up 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 switches (refer to Table 11-2).

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) or 0.007ft if using Units == US Customary. 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 (0.16ft) 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 or 0.0007ft) may be required due to the substantial amount of shallow sheet flow. A reduced wet/dry depth of 0.0002m (0.0007ft) is particularly recommended for direct rainfall models, noting that the cell wet/dry depth cannot be set to below 0.0002m (0.2mm) or . 0.0007ft.

For multiple 2D domains, this command is domain dependent.

Cell Side Wet/Dry Depth == [{0.001} | <depth_in_m>]

(Optional)

Note: Legacy command. No longer required or recommended for use subsequent to implementation of Wetting and Drying == ON METHOD B. See TUFLOW 2010 manual for details.

Cell Size == <value_in_metres>

(Mandatory if not specified in .tgc file)

Sets the grid cell size in metres. Rarely used; normally specified in the .tgc file (see Section 3.3.1).

Change Zero Material Values to One == [ON | {OFF}]

(Optional)

The default material value is zero which means that every cell must be assigned a material value (ie. use Set Mat as the first materials command in the .tgc file). For backward compatibility set to ON.

Check Inside Grid == [{ERROR} | WARNING | OFF]

(Optional)

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.

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.

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.

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

(Optional)

Relaxes new rules in the 2016-03 release that will force an ERROR if a "==" (double =) is not specified for a command. This is to help prevent issues associated with specifying a single = and the command not being processed correctly (as could occur prior to the 2016-03 release).

CSV Header Line == [{} | SINGLE]

(2D only. Optional)

Note: This is a legacy command that applies to formats prior to the default .csv output for the 2016-03 release.

Specifying SINGLE will only output a single header line in _PO.csv files, which makes it much easier to use the file for graphing in Excel. The simulation name is also included in the label so that it's easy to distinguish between simulations when graphing comparisons. This is not the default setting, so this command needs to be specified to activate the feature.

CSV Maximum Number Columns == <max_col>

(1D & 2D/1D. Optional)

Note: This is a legacy command that applies to formats prior to the default .csv output for the 2016-03 release.

Used to specify a maximum number of columns for 1D .csv output files. The default setting is limitless. Refer also to the .ecf command CSV Maximum Number Columns.

CSV Time == [DAYS | {HOURS} | MINUTES | SECONDS]

(1D & 2D/1D. Optional)

Specifies the time values of .csv outputs. The default is HOURS.

Will apply to 1D and 2D .csv output.

```
Defaults == [ PRE 2016 | PRE 2013 | PRE 2012
| PRE 2010-10 | PRE 2008-08 | PRE 2007-07 | PRE 2006-06 ]
(Optional)
```

PRE 2016 sets backwards compatibility to the 2013-12 release. This resets the default settings back to:

- Bridge Zero Coefficients == OFF
- Weir Flow == METHOD B
- Output Data Types == H V Q
- Boundary Viscosity Factor == 0.0 for single 2D domain models.
- Layered FLC Default Approach == CUMULATE
- SX Flow Distribution Cutoff Depth == 0.0
- Zero Rainfall Check == WARNING
- GIS Unsupported Object == WARNING | 0
- End After Maximum == <eam> | 0.0 (ie. a tolerance of zero applies)
- XMDF Output Compression == OFF
- Activates the previous method for selecting the primary upstream and downstream channels (there is no special command available for this change, although one can be provided upon request – please email support@tuflow.com). The previous method did not correctly take into account the channel's bed slope, which on occasions would not select the primary upstream and/or primary downstream channel to be that with the closest bed level. This may change results where the primary upstream and downstream channels have an effect, for example, in determining the approach and departure velocities at a structure. Changes in results, if any, are usually minor.

PRE 2013 sets backwards compatibility to the 2012-05 release. This resets the default settings back to:

- Structure Routines == ORIGINAL
- Structure Flow Levels == WATER
- Weir Flow == METHOD A
- PO Approach == METHOD A
- Time Series Output Format == PRE 2013
- Map Output Corner Interpolation == METHOD A
- Maximum Courant Number == 0.3 (only applies if GPU Solver == ON)
- Maximums and Minimums == OFF
- Write Restart Filename == OVERWRITE
- Link 2D2D Approach == METHOD B

PRE 2012 sets backwards compatibility to the 2011-09 release. This resets the default settings back to:

- Manhole Approach == METHOD A
- M Channel Approach == METHOD A
- SA Proportion to Depth == OFF
- Wetting and Drying == ON
  (2012-05 default is Wetting and Drying == ON METHOD B)
- Viscosity Coefficient == 0.2, 0.1
- Maximums and Minimums Time Series == OFF
- HQ Boundary Approach == METHOD A

For **PRE 2010-10** and earlier please refer to the TUFLOW 2010 manual.

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.

Ideally placed within a .tef file (TUFLOW Event File) – see Section 11.3.1, but can be used one or more times in a .tcf file.

Define Output Zone == <oz_name>

(Optional)

Starts a block of .tcf commands for an Output Zone named <oz_name>. Only one block can be specified for each unique < oz_name >.

Use End Define to terminate the block. An ERROR occurs if End Define is not specified. Refer also to Model Output Zones.

Demo Model == [ON | {OFF}]

(Optional)

When set to ON, allows simulation of the Demo Models developed for the 2012 Flood Managers Association (FMA) Conference without the need for a TUFLOW license. Note that writing to the _All_TUFLOW_Simulations.log file (see Section 12.4.2) is switched off when this command is set to ON. For further information on the Demo Models refer to Section 2.4.2 and the TUFLOW Wiki.

Density of Air == [{1.25} | <value>]

(Optional)

Sets the density of air in kg/m^3. 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/m^3.

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.

Display Water Level == <X>, <Y>

(Optional)

Displays the water level on the screen for the cell located at X,Y where X and Y are the geographic coordinates in metres.

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 Flood Modeller 1D link as incorrect results presently occur. It has not been tested with the XP-SWMM 1D link.

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.

End After Maximum == <eam> | [{0.001} | <h_tol>]

(Optional)

Terminates the simulation <eam> hours after the last time a new maximum was recorded anywhere in the model. End Time should also still be specified as an upper limit to finish the simulation.

<h_tol> is an optional second argument that sets a height tolerance for detecting whether a 1D node or 2D cell has reached its maximum. For example, "End After Maximum == 0.25 | 0.01" will terminate the simulation once either "End Time ==" has been reached, or there are no 1D nodes or 2D cells that have increased by 1cm (0.01m) in the last 15mins (0.25h). The % of 1D nodes and 2D cells that have reached their maximum are displayed after the "Mx" on the console window and in the .tlf file.

The default for <h_tol> is 0.001m (or 0.001ft), unless "Defaults == PRE 2016" in which case the default is 0.0. Note that is using a 0.0 tolerance this command may not work as expected due to numerical precision issues.

End Define

(Optional)

Ends a Define Event or Define Output Zone block of .tcf commands. The End Define command must be specified if either of the two preceding commands occur within the .tcf.

<format> End Map Output == <time_in_hours>

(Optional)

The simulation time in hours when map output terminates. If the command is omitted, the simulation end time is used.

This command can be defined for different output formats by including the output format extension on the left of the command. For example, to set the end time for XMDF output to 10hrs use:

```
XMDF End Map Output == 10
```

This functionality is also available for the commands Start Map Output, Map Output Interval and Map Output Data Types. Refer to Section 9.4.2.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

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.

ESTRY Control File [{} | AUTO] == <.ecf_file>

(Mandatory if linking to an ESTRY 1D model)

Specifies the ESTRY control, .ecf, file (see Section 4.6). There can only be one .ecf file.

The AUTO option automatically sets the .ecf filename to the same as the .tcf file (except for the extension).

Note 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).

Event File == <.tef_file>

(Optional)

Sets the active event file. See Section 11.3.1.

Whilst this command may be repeated to change the active event file it is recommended that only one event file be created for all simulations.

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>.

Note: Only applies if CSV Time == DAYS.

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.

Force File IO Display == [ON | {OFF}]

(Optional)

If set to ON, forces all file opening and closing information to be displayed to the screen and .tlf file. At present, nearly all file opening and closing is displayed, but some isn't and more may be removed in future releases (eg. Write PO Online == ON displays a lot of file opening and closing information as the simulation proceeds, so should you wish to activate this for checking purposes set this command to ON).

Note: This command would mainly be used for debugging a file opening and closing issue.

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 6.3.1.

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).

The default is 0.6. 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 (See Section 6.3.1). 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).

GA Convergence Value == [{0.001} | <value_in_metres>]

(Optional)

Sets the Green Ampt iteration infiltration convergence test in metres. The default value is 0.001m.

GA Iterate == [{ON} | OFF]

(Optional)

Sets the Green Ampt infiltration to iterate until convergence is reached. See also GA Convergence Value and GA Maximum Iterations commands. The default approach is ON.

GA Maximum Iterations == [{10} | <value>]

(Optional)

Sets the limit on the number of iterations for the Green Ampt solution. The default value is 10. If the number of iterations exceeds this value a WARNING 2302 is issued.

Geometry Control File == <.tgc_file>

(Mandatory)

Specifies the geometry control, .tgc, file (see Section 4.7). There can only be one .tgc file per 2D domain.

GIS Format == [{MIF} | SHP]

(Optional)

Specifies the output format for GIS check layers and GIS outputs such as the _TS layers. If the command GIS Format is not specified, the GIS format used for check layers and other GIS outputs is based on

whether MI Projection or SHP Projection has been specified. If neither or both of these commands have been specified, and GIS Format has not been specified, the default of using .mif files is adopted.

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

GIS Grid Vector Type == [{Region} | Point]

(Optional)

Specifies whether the vector information (output over a regular grid) should be as points or region objects. Region objects refer to the arrows that TUFLOW_to_GIS produces, scaled according to the vector magnitude. The magnitude and direction are output as attributes to the layer. The default is regions (as per TUFLOW_to_GIS).

To apply a different setting for different vector data types, inset the data type before the command. See Section 9.6.6.

GIS Grid Vector Direction == [{ANGLE} | BEARING | VERBOSE]

(Optional)

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

To apply a different setting for different vector data types, inset the data type before the command. See Section 9.6.6.

GIS Grid Vector SF == [{1} | <scale factor>]

(Optional)

Scale factor for the scaling of region objects for GIS output of vector data. The default value is 1. A value of 1 means a velocity of 1 m/s is one 2D cell long, therefore, with a scale factor of 2, a vector of magnitude 1 m/s would be two 2D cells long. A negative value outputs vectors of fixed length equal to <scale_factor> in metres or feet.

To apply a different setting for different vector data types, inset the data type before the command. See Section 9.6.6.

GIS Grid Vector TTF == [{0} | <tail_thickness_factor>]

(Optional)

Tail thickness Factor, scales the thickness of the arrow tails (default = 0). The thickness is the <tail_thickness_factor> times the arrow length. For some GIS software such as ArcGIS, a zero tail thickness value may cause issues as the region shape wraps onto itself. To apply a different setting for different vector data types, inset the data type before the command. See Section 9.6.6.

GIS Projection Check == [{ERROR} | WARNING]

(Optional)

Checks that the MI Projection and/or the SHP 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 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.

The default setting is ERROR and will prevent TUFLOW from starting the model simulation. Changing to WARNING output a message to the messages.mif/.shp GIS layer and will allow the model to continue to compile.

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

(Optional)

Controls the response to GIS object ignored messages (for example, see Message 2073). If set to ERROR the simulation is terminated, while is set to WARNING (the default) the message is issued and the simulation continues. The default is for a WARNING to be issued as per releases prior to 2016-03.

Note that various TUFLOW inputs expect different GIS object types, so the behaviour of this reporting varies. For example a 1D boundary (1d_bc) layer can contain points snapped to the 1D nodes and/or region objects used to apply flow boundaries to nodes that fall within the region. So whilst a line or polyline is a generally supported object (see GIS Unsupported Object) any line objects in a 1d_bc layer are not used and TUFLOW issues Message 1099.

GIS Unsupported Object ==
[{ERROR} | WARNING] | [{1} | <level>]

(Optional)

Controls the approach for issuing messages in relation to unsupported GIS objects. As a background to the changes GIS software typically store vector data in three broad geometries:

- Points
- Lines

- Regions

Within these geometry types, different GIS packages may offer options for digitising objects. For example, when drawing a line object in MapInfo the user has the option for a line, a polyline and an arc. From left to right the editing buttons to digitise a line, polyline and arc in MapInfo are:

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

```
Line 340604.21 5782377 340612.07 5782369.48
    Pen (1,2,0)
Pline 3
340609.55 5782359.99
340614.35 5782361.73
340623.19 5782363.59
    Pen (1,2,0)
Arc 340630.84 5782358.24 340640.43 5782382.01 180 270
    Pen (1,2,0)
```

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

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

This command, introduced for the 2016-03 release, controls TUFLOW's response for geometries that are not supported by TUFLOW. The ERROR option stops the simulation with a Message 0323, while WARNING issues Message 0323 and continues the simulation. There is an optional severity level component that can be specified as a second argument (separated by a vertical bar) with the options for <level> being:

The severity levels are:

- Level 0 – No checks on unsupported geometries (i.e. previous behaviour)
- Level 1 – Check for ellipses, rectangles, rounded rectangles and arcs (curved arcs)
- Level 2 – All the level 1 checks as above plus checks for null and text objects.

The default for unsupported objects is ERROR | 1, that is arcs, ellipses, rectangle and rounded rectangles will cause the simulation to stop, but any None and Text objects will be ignored.

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.

Global Weir Factor == [{1.0} | <value>]

(Optional)

Factor that adjusts the broad-crested weir formula (see Section 6.3.1). 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).

GPU Device IDs == <list_of_device_ids>

(Optional)

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

```
GPU Device IDs == 0, 2
```

Note that the GPU device numbering starts at 0 rather than at 1.

A GPU licence will be needed for each device ID.

Also refer to the batch switch –pu<id> described in Table 11-2 which has the same function.

GPU DP Check == [{ERROR} | OFF]

(Optional)

The default setting of ERROR, causes TUFLOW to stop with ERROR 2420 advising that it is recommended to use the single precision version of the GPU Solver. Due its explicit formulation and being depth based, TUFLOW GPU does not usually require to be run in double precision (DP) mode. Refer to Section 10.4.1 for further information. There can also be substantial speed gains using single precision (SP) on some GPU cards, and there is a significantly less memory footprint. If DP is desired or required for the GPU Module specify GPU DP Check == OFF in the .tcf file, and run using a DP TUFLOW exe.

GPU RAM Optimisation == [{ON} | OFF]

(Optional)

Set to ON to not optimise CPU RAM allocation for GPU Solver simulations. This will cause the CPU RAM requirements to use around 50% more memory. Should only be necessary to set to OFF to identify whether the memory allocation is the cause of any issues when running the GPU Solver.

GPU Solver == [ON | {OFF}]

(Mandatory for TUFLOW GPU models)

Must be set to ON to simulate the model using the TUFLOW GPU Engine. If the command is not specified or set to OFF, the standard TUFLOW Classic CPU Engine will be used to simulate the model.

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

(Optional)

This command sets the order of the temporal solution for TUFLOW GPU simulations. The default is the recommended 4th order temporal solution therefore this command is usually not specified. Available options are:

1 – First order out of place

2 – Second Order

4 – Fourth Order

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

Grid Format == [ASC | {FLT}]

(Optional)

Sets the format that TUFLOW uses to write output grids. The default is the FLT format. For ASCII format, Grid Format == ASC must be specified.

Grid Output Cell Size == <grid_cell_size>

(Optional)

Sets the cell resolution in metres of all grid outputs with the default being half the smallest 2D cell size. At present only one output grid resolution is possible, so the last occurrence of the command will prevail. It is planned to allow different resolutions to be specified for different map output formats and different Output Zones in a future release. Also note that a DEM of the final Zpts is now automatically written

using this resolution if writing check files unless it is excluded using Write Check Files Exclude == DEM_Z.

Increasing the grid output cell size will reduce the RAM required and the size of the output files such as the ASC, FLT, NC (NetCDF) and WRR formats. Therefore, for very large models (eg. GPU Solver models), consider increasing this value if there are memory (RAM) allocation issues.

Grid Output Origin == [{AUTOMATIC} | MODEL ORIGIN]

(Optional)

AUTOMATIC, the default, adjusts the origin for the output grids by rounding to the Grid Output Cell Size so that all grids produced from different simulations using different model extents, and between different Output Zones, are all aligned.

If Grid Output Origin == MODEL ORIGIN is specified, this sets grid output (eg. ASC or FLT) to have its origin exactly at the model's lowest left coordinates for map output. Note that output grids of different origins (due to a change in the model's schematisation or addition of 1D WLLs), or from different Output Zones, may not be aligned.

HQ Boundary Approach == [METHOD A | METHOD B | {METHOD C}]

(Optional)

Sets the approach to be used for automatically generated 2D HQ boundaries (refer to Table 7-4).

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 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 (refer to Table 7-5) can be used to specify the max depth to be used for generating the curve (if less than 1m it is set to 1m).

METHOD C (the default) automatically removes consecutive flow values in automatic 2D HQ boundary tables. For existing models that run with an automatic 2D HQ boundary, if the change in consecutive flow values is less than 0.00001 the results may be very slightly affected by this change. Use of Method B provides backward compatibility if this is an issue.

HX Force Weir Equation == [ON | {OFF} | <value>]

(Optional)

When set to ON, forces the weir flow equation to be applied across all active HX cell sides when the flow is upstream controlled (the default is OFF). This command lowers the HX cell centre elevation by 0.002m below the lowest active HX cell side elevation to force an adverse slope. If a value is specified

the feature is turned ON and the value is the amount by which to set the HX cell centre elevation below the lowest active cell side, for example, HX Force Weir Equation == 0.1 would use 0.1 instead of 0.002.

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

```
HX ZC Check == [ {ON} | OFF ]
```

(Optional)

If ON (the default), checks whether the minimum ZC elevation at or along a HX object (see Table 7-4 and Table 7-5) 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 7-5), 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. Turning this check off is not recommended and may results in mass balance issues.

```
If Event == <e1> | <e2> | <e3>...
[ Else If Event == <e1> | <e2> | <e3>...  ]
[ Else ]
End If
```

(Optional)

Controls which commands to process for different events as specified using the –e run (see Section 11.3.1 and Table 11-2) or Model Events.

The "If Event" block must be terminated by "End If" otherwise an ERROR occurs.

Optional "Else If Event ==" and "Else" blocks can be embedded between "If Event" and "End If". The first block encountered that refers to a specified scenario is processed, and all remaining blocks within the "If Event" 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 Event" blocks) and its commands are only processed if no previous blocks have been processed.

```
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 Section 11.3.2 and Table 11-2) 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.

```
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.

```
Inside Region == [ METHOD A | {METHOD B} ]
```

(Optional)

Specifies the method used to assign 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.

Method B offers much faster processing than Method A. To appreciate the increase in start-up time this feature offers, testing on two large models reduced the start-up time from 20 minutes to 3 minutes for one model, and from 40 minutes to 5 minutes for the other. The faster start-up time occurs for any polygon layers being accessed from the .tgc and .tbc files, particularly those containing large number of vertices.

Instability Water Level == [{see_below} | <value_in_metres>]

(Optional)

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).

Alternatively, this command is used to set the instability water level manually.

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.

Layered FLC Default Approach == [CUMULATE | {PORTION}]

(Optional)

New command for the TUFLOW 2016-03 release to specify the settings for layered flow constrictions (see Section 6.12.2.2). The default setting for the 2016-03 release of TUFLOW is PORTION. For releases prior to 2016-03 the CUMULATE approach was used.

It is possible to individually specify the method to be used for each structure. Specify either CUMULATE or PORTION in the Shape_Options attribute (refer to Table 6-21). If neither CUMULATE or PORTION occur, the setting as specified in the command Layered FLC Default Approach is used.

Line Cell Selection == [METHOD A | METHOD C | {METHOD D}]

(Optional)

Sets the method for selecting 2D cells along lines in GIS layers.

Method A (used prior to Build 2006-06-AA) is the original method and is provided for backward compatibility.

Method C (the default up until Build 2007-04-AC), 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.

Method D (the default) selects cells using the same approach as for Method C, however 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.

Method C and Method D 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.

Link 2D2D Adjust Velocity Head Factor == [{0} | < value >]

(Optional)

Sets the proportion of the velocity head by which to adjust the water levels along the 2D link line. A value of zero applies no adjustment whilst a value of 1.0 adjusts (downwards) the level by $V^2/2g$. The default is 0.0 (no adjustment). Some tidal models demonstrated improved performance by setting to 1.0, however, where there is rapid changes in velocities along a 2D link line, for example in an urban situation where velocities vary considerably from road to garden to between buildings, this feature may not improve the 2D link performance, therefore sensitivity test before adopting.

Link 2D2D Approach == [METHOD A | METHOD B | METHOD C | {Method D}]

(Optional)

Command used to determine the approach used to link multiple 2D domains (refer to Section 8.3.1)

Method A is the original 2D2D link approach introduced for the 2009-07 release. This command and Method B was also incorporated into the 2009-07 release.

Method B (also introduced for the 2009-07 release) was an improvement over Method A which was prone to generating flows where the 2D2D link was located along dry boundaries with "bumpy" topography. Method B is preferred over Method A.

Method C (introduced for the 2012-05 release) is the same as Method B, but allocates the storage of the 2D link cells (these cells do not contribute to the model's storage so the storage is assigned to the hidden nodes) more evenly between the hidden nodes and ensures the nodes' bed elevation is at or below the lowest connected 2D link cell.

Method D (the default) includes significant enhancements to the 2D/2D linking of different 2D domains. It is recommended that models utilising 2D/2D linking upgrade to Build 2013-12-AC or later where possible to make use of these enhancements. For backwards compatibility, specify Link 2D2D Approach == Method B.

Link 2D2D Distribute Flow == [{ON} | OFF]

(Optional)

Improves the distribution of flow between 2D cells along 2d_bc 2D link lines. The default is ON. For 2D/2D linking methods prior to Method D OFF applies.

Link 2D2D Global Stability Factor == [{1.0} | <value>]

(Optional)

Will globally factor all 2D link hidden 1D node storages to easily carry out a quick sensitivity test on the effect of increasing the storage of the hidden 1D nodes or to globally improve stability along 2D link lines if needed. The default value is 1.0. Usually, increasing the storage of the hidden 1D nodes by up to around a factor of 5 has only a slight effect on results. Note that any factor specified using the 2d_bc "a" attribute is also applied on top of this factor. Also note that the default value for the 2d_bc "a" attribute using Method D is 1.0 (prior methods use 2.0).

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.

Map Cutoff Depth == [{0.0} | <value_in_m>]

(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 very shallow sheet flow and flooding.

Map Output Corner Interpolation == [METHOD A | METHOD B | {METHOD C}]

(Optional)

This command determines the approach taken to extrapolate model results to the cell corners at the wet/dry interface.

Method A (the approach adopted prior to the 2013-12 release) was the original routine where some issues were found with tracking the maximum depth related outputs, particularly on steep slopes, at the wet/dry interface where the depth extrapolated to the cell corners was exaggerated as it was extrapolated horizontally.

Method B and Method C resolve the issues found in Method A. Method B will provide slightly better output along thin breaklines where they are dry or free-overfalling occurs, however, Method C provides the most consistent extrapolation to cell corners in terms of tracking maximum values at the wet/dry interface on steep slopes and is the recommended approach and default setting.

<format> Map Output Data Types == [{h V} | <data_types>]

(Optional)

Defines the data types to be output. Refer to Table 9-10 and Table 9-11 for a description of available options. The data types may be specified in any order or combination and are not case sensitive. Spaces between data types are optional, however are strongly recommended to ensure there is no misunderstanding.

For example to output water levels, velocities and unit flow enter the following line:

```
Map Output Data Types == h V q
```

The default is:

```
Map Output Data Types == h V
```

The format(s) of the map output is controlled by the .tcf command Map Output Format (refer to Section 9.7 for further information). Note that not all Map Output Data Types are available for all formats due to limitations or constraints of the format.

This command may also be defined for different output formats by including the output format extension on the left of the command (refer to Section 9.4.2). For example, to write the XMDF Map Output Data Types H and D use:

```
XMDF Map Output Data Types == h d
```

This output format specific functionality is also available for commands Start Map Output, End Map Output and Map Output Interval.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

Map Output Entire Model == [{ON} | OFF]

(Optional)

The default is to produce map output for the whole model irrespective of the number of Output Zones (refer to Section 9.4.3) being written. Map output commands that occur outside Output Zone definitions apply to the entire model (ie. as is the case in previous releases). If map output for the entire model is not required specify Map Output Entire Model == OFF.

Map Output Format
== [{XMDF} | <formats>]
[{SMS} | SMS HIGH RES | SMS HIGH RES CORNERS ONLY |
SMS TRIANGLES]

(Optional)

Sets the format(s) for TUFLOW map output. As of the 2016-03 release there are now a wide range of output formats to choose from as documented in Section 9.6 and listed in Table 9-9.

The default setting for the 2016-03 release is the XMDF format, whilst prior to 2016-03 the default was the DAT file format. The available output formats are described below with further detail discussed in Section 9.6. For example, to output formats in XMDF and FLT formats specify:

```
Map Output Format == xmdf flt  ! Produce map output in XMDF and FLT formats
```

This command can be used within a Define Output Zone block to change the setting from that for the entire model map output to be different for the output zone.

There is no limit (other than disk space!) on the number of formats specified per model simulation. Some further examples are:

Example 1

```
Map Output Format == XMDF ASC
```

Outputs results in both XMDF and ASC file format

Example 2

```
Map Output Format == XMDF SMS TRIANGLES
```

Outputs results in in XMDF using the SMS TRIANGLES approach to include the cell centre values.

Example 3

```
Map Output Format == DAT SMS
```

Outputs results in DAT format using the standard SMS approach. This is the default if the Map Output Format command is omitted.

Output format specific commands (Map Output Interval, Start Map Output, End Map Output and Map Output Data Types) can then be used to customise each format type as per the examples below. Refer to Section 9.4.2.

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

```
XMDF Map Output Interval == 360
```

Or to only output depths (d) and Bed Shear Stress (BSS) in ASC format specify:

```
ASC Map Output Data Types == d BSS
```

<format> Map Output Interval == <time_in_seconds>

(Mandatory)

The output interval in seconds for map based output. If the command is omitted, an ERROR 0045 is output.

If set to zero (0) no time based map output is produced, and only the maximums are written (note tracking of maximums must be switched on – see command Maximums and Minimums).

This command can be defined for different output formats by including the output format extension on the left of the command. For example, to set the Map Output Interval for XMDF output to 6 minutes use:

```
XMDF Map Output Interval == 360
```

If only the maximums in ASC format are required, use:

```
ASC Map Output Interval == 0
```

This output format specific functionality is also available for commands Start Map Output, End Map Output and Map Output Data Types. Refer to Section 9.4.2.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

Mass Balance Corrector == [ON | {OFF}]

(Optional)

Note: Legacy command. No longer required or recommended for use subsequent to implementation of Wetting and Drying == ON METHOD B. See TUFLOW 2010 manual for details of command.

Mass Balance Output == [{ON} | OFF]

(Optional)

If set to ON (the default), 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.

It is possible to use different timesteps for the mass balance outputs and those shown on the Console Window. The command Mass Balance Output Interval is used for setting the interval in the mass balance outputs whilst Screen/Log Display Interval sets the interval for the Console Window. A summary is included at the end of the simulation on the display console and .tlf file of key model performance indicators (see Section 14.2.1).

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 set to 0 (zero), the output interval used is the largest 2D computational timestep set by the command Timestep.

If the command is omitted (or set to -1), output is at the lesser of Map Output Interval and Time Series Output Interval.

Maximum 1D Channels == [<mchan> | {100000}]

(Optional)

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

Maximum Courant Number == <Cr_max>

(Optional for 2D Models only)

TUFLOW Classic:

An **under-development** feature that switches on adaptive timestepping. The command is only available for 2D only models at present, but may be extended to 1D/2D models in future. A Cr_max value of 0 or less switches adaptive timestepping off (the default for TUFLOW Classic).

The Timestep command is only used to set the initial timestep if <Cr_max> is greater than zero. It is possible to control the maximum rate at which a timestep can increase by using the command Timestep Maximum Increase. There is no limit to how quickly the timestep can decrease.

See Section 3.4.4 for more information.

TUFLOW GPU

This option when greater than zero switches on the adaptive timestepping. For the 4th order time solver a value of 1.0 (the default) is recommended. If set to 0 adaptive timestepping is turned off and the fixed timestep (set using command Timestep) is used. The recommendation is not to specify this command and use the default settings.

For GPU Solver simulations, the Timestep command is only used to help set the initial timestep if <Cr_max> is greater than zero. The approach taken is to use an initial timestep equal to the Timestep value divided by 10, where the Timestep value entered should be the same as that used for a TUFLOW Classic simulation. TUFLOW Classic, being implicit, uses much greater timesteps than the explicit solver used in TUFLOW GPU. This approach was adopted so that if switching between Classic and GPU runs there is no need to change the Timestep value.

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 Velocity Cutoff Depth == [<y> | {0.1}]

(Optional)

This 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) 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.

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.

Maximums and Minimums == [ON | OFF | {ON MAXIMUMS ONLY}]

(Optional)

Sets whether maximums and/or minimums are to be tracked and written as map output. The default setting for this command is ON MAXIMUMS ONLY. To suppress tracking and outputting of maximums for map output set to OFF.

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

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

Maximums and Minimums Only For Grids == [ON | {OFF}]

(Optional)

Writes only the maximum and minimum grids for specified Map Output Data Types at the end of a simulation. This command only applies if the "Grid", "ASC" and/or "FLT" option is specified for the command Map Output Format.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

Maximums and Minimums Time Series == [{ON} | OFF]

(Optional)

If set to ON, allows for tracking of maximums and minimums at every timestep for PO and LP outputs.

Maximums Approach == [Method A | {Method B}]

(Optional)

This command specifies the approach taken in tracking the maximums for map outputs.

Method A is the original approach and only provided for backward compatibility for legacy models.

Method B (the default) introduced for Build 2012-05-AC, fixes a bug that would not correctly track the maximums if both cell cornered map output (.dat and .xmdf formats) and the GRID (ASCII grid) output were both specified. As a consequence of this bug, the approach for tracking cell-centred output

maximum water levels is now used by default for both cell-centred and cell corner output formats. This may cause a very slight lowering of maximum water levels (usually a mm or less) in isolated locations.

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 15.2.1) 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.

MI Projection ==
[<.mif file> | Projection_line_from_MIF_file>]

(Optional but recommended)

Sets the geographic projection for all GIS input and output in MID/MIF format. 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 MID/MIF format. If no "Header.mif" file is found, non-earth coordinates are assumed.

Alternatively, a projection line extracted from a mif file may be entered (although the previously described approach of specifying a MIF file is the recommended approach). 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 (metres) 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 MID/MIF GIS layers read by TUFLOW MUST USE this projection. The projection must be a Cartesian based projection and in metres, not a spherical projection such as Latitude/Longitude.

Further examples of creating a projection file using various software packages can be found on the TUFLOW Wiki.

See also the corresponding command SHP Projection, for GIS input and output files in .shp format. If a model has a mixture of .mif and .shp files as input, then both MI Projection and SHP Projection should be specified.

Model Events == <e1> | <e2> | <e3>...

(Optional)

Alternative to using the –e option when running TUFLOW (see Table 11-2). 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.

Model Output Zones == <oz_name_1> | <oz_name_2> | …..

(Optional)

Controls which outputs zones are to be used. If this command is omitted, no output from Output Zones will be written. Separate multiple Output Zones using a "|". For example, to output from zones ZoneA and ZoneC specify:

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

See Section 9.4.3 and the .tcf command Define Output Zone for further information.

Model Platform == [w32 | w64]

(Optional)

Forces the .tcf file to only be run using either a 32-bit (w32) or 64-bit (w64) version. If the command is not specified, the model may be run in either version. If the w32 version of TUFLOW is specified,

and the w64 version is used, the simulation stops with an error. Similarly, if w64 is specified, the simulation stops if the w32 version is used. For further discussion see Section 11.4.

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 11.4.

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 Scenarios == <s1> | <s2> | <s3>...

(Optional)

Alternative to using the –s option when running TUFLOW (see Table 11-2). 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.

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. The simulation stops with an error if the specified build is different to that used to run the model.

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 | 2011-09 | 2012-05 | 2013-12]

(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.

Negative Depth Approach == [Method A | {Method B}]

(Optional)

Method A represents the case of no special treatment of negative depths.

Method B (the default) 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.

NetCDF Output Compression == [OFF | {ON} | <level>]

(Optional)

Sets the compression for the output NetCDF file. If set to ON (the default) a compression level of 1 is used. The compression level can also be set via a number from 1 – 9. Higher levels result in smaller files but are slower to write/access, with the greater the number the greater the compression and slower the write/read time. If set to OFF a NetCDF "classic" format is used without compression. If set to ON a NetCDF 4 file is used with compression. The compressed version works well in ArcMap and Matlab but not in QGIS at the time of writing.

NetCDF Output Start Date == [{2000-01-01 00:00} | OFF | <date_in_isodate_format>]

(Optional)

Sets the output units for the NetCDF time variable. If set to OFF or NONE, the "units" attribute of the .nc file is simply the simulation time unit (eg. units = 'hours'). If a date is provided the NetCDF time units will be in the format "<unit> since <date>". For example, unit = 'hours since 2000-01-01 00:00' or 'days since 2000-01-01 00:00'. The default setting is '2000-01-01 00:00' as not having a date appears to cause issues in ArcMap. If a date is specified, it is strongly recommended that this is in isodate

format. TUFLOW **does not** check the date is valid, it is simply written to the NetCDF time variable as entered in this command.

See also NetCDF Output Time Unit command below.

NetCDF Output Time Unit = [DAY | {HOUR} | MINUTE]

(Optional)

Set the output time unit for .nc outputs. The default is hours (as per other TUFLOW outputs). See also NetCDF Output Start Date command above.

NetCDF Output Direction == [{ANGLE} | BEARING]

(Optional)

When output in NetCDF raster format any vector outputs (eg. flow or velocity) are output as two datasets: magnitude and direction, for example, magnitude_of_velocity and direction_of_velocity. This command specifies whether the direction_of NetCDF attribute is in arithmetic (0 degrees east) or geographic (0 degrees north). The default is arithmetic.

NetCDF Output Format == [{Generic} | FEWS]

(Optional)

Sets the output format for the NetCDF outputs, the outputs are similar but the FEWS format option outputs the maximum, time of peak, duration outputs with a time stamp at the beginning of the simulation to ensure that it loads correctly into FEWS. For the generic output format no timestep is output with these static datasets. For more information on the format please see the TUFLOW Wiki page TUFLOW NetCDF Raster Format for more details.

Null Cell Checks == [ON | {OFF}]

(Optional)

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.

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 dam break 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.

Output Approach == [{DEFAULT 2016} | PRE 2016]

(Optional)

Sets the output approach. For the TUFLOW 2016-03 release a new approach for time-series and summary .csv files was implemented that combines the 1D and 2D plot (time-series) data into a single folder, allowing utilities such as TuPlot to provide access to all plot output under the one folder. For further information see Section 13.2.

It is intended that further enhancement of the default settings for map and other outputs will be added to future releases.

For output in formats prior to the 2016-03 release use PRE 2016.

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.

Output Files [{ALL} | NONE | INCLUDE | EXCLUDE] == <output type list>

(Optional)

Can be used to supress or include certain time series and 1D output. Whilst this command currently only applies to 1D output it is assigned to the .tcf commands as is intended to extend to other output files in future releases that may contain 2D output.

The EXCLUDE or INCLUDE options allow for a space delimited list of file extensions or prefixes to be specified to exclude or include output files and GIS layers from being written. The notation must be the same as those used by TUFLOW. For example, "eof" would apply to the .eof file. To exclude/include more than one layer ensure there are spaces between the prefixes. If EXCLUDE or INCLUDE occurs more than once, the latter occurrence prevails.

Note that for the .eof file, this only affects the writing of results output to this file. The header information and input data written to the .eof file will still be output.

Valid options to include or exclude are: EOF, 1d_mmH, 1d_mmQ, 1d_mmV, 1d_CCA, TS, TSF, TSL, TSMB, TSMB1D2D.

Examples:

```
Output Files EXCLUDE == EOF TSMB TSMB1D2D
Output Files INCLUDE == TS, 1d_CCA
Output Files ALL
Output Files NONE
```

The ALL and NONE options require no file type list. All output files will be written, specification of the ALL or NONE option will nullify any prior occurrence of an EXCLUDE or INCLUDE list; this is useful if you wish to write no or all output files for one particular run – simply add Output Files ALL to the end of the .tcf file.

Output Folder == <folder>

(Optional)

Redirects all TUFLOW output data except the log and summary files to <folder>. Typically used to write output to your local C: or D: drive instead of filling up the network drive, or to keep results separate to the input data. A URL can be used (eg. \\wbmserv\Computer001\tuflow\results), which is 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).

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

PO Approach == [METHOD A | {METHOD B}]

(Optional)

METHOD A is the original 2d_po flow lines approach and only provided as a precaution for legacy models. Method A and Method B should give the same results, but Method B is much faster and can substantially reduce simulation times if numerous 2d_po flow lines exist.

METHOD B (the default) utilises a substantially faster approach for 2d_po flow lines, which will not slow down a simulation if numerous 2d_po flow lines exist, as was occurring with old releases.

The previous approach can be used by specifying PO Approach == Method A however, this should only be necessary in the event of a problem arising in which case please email support@tuflow.com.

Pause == <message>

(Optional)

Causes TUFLOW to stop whenever it encounters it. It can also be used to pause the simulation and display a given message. The user has the option to continue or discontinue the simulation via a dialog window.

Example:

```
Pause == Invalid Scenario specified.
```

Rainfall Boundaries == [{STEPPED} | SMOOTHED | SMOOTHED TIME CENTRED]

(Optional)

The SMOOTHED TIME CENTRED 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 (the default) 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).

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.

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).

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. The command can be used to redirect file(s) up to a maximum of ten levels.

Also available in .tgc and .ecf files.

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 WI10 and AP options for Map Output Data Types.

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.

Read GIS FC == <gis_layer>

(Optional)

Opens a GIS layer in .mif/.mid or .shp file format containing details on flow constrictions to model bridges, box culverts, etc. (see Section 6.12.2). This command may be used any number of times.

It is recommended to use the .tgc command Read GIS FC Shape in preference to this command.

Read GIS GLO == <gis_layer>

(Optional)

Opens a GIS layer in .mif/.mid or .shp file format containing details on gauge level output (2d_glo) locations (refer to Section 9.4.4). 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.

A buffer has been 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.

Read GIS IWL == <gis_layer>

(Optional)

Opens a GIS file in .mid/.mif or .shp format 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 7.6.1.2. Initial water levels may also be read directly from an .asc or .flt grid using the command Read Grid IWL.

Read GIS LP == <gis_layer>

(Optional)

Opens a GIS layer containing details on longitudinal profile output (LP) locations (see Section 9.3.3). This command may be used any number of times.

Read GIS Output Zone == <gis_layer>

(Optional)

Reads a GIS layer containing one or more polygons that define the regions to be output. The attributes of the layer are not used. Refer to Section 9.4.3 for further information on Model Output Zones.

Read GIS PO == <gis_layer>

(Optional)

Opens a GIS layer containing details on plot output (PO) locations (see Section 9.3.3). This command may be used any number of times.

Read GIS Reporting Location == <gis_layer>

(2D/1D. Optional)

Reads GIS points and lines for time series result from 1D and 2D sections of the model. This feature has the ability to combine flows, and track maximums, across 1D and 2D domains. See Section 9.3.1 for further information.

Read GIS X1D Network == <gis_layer>
Read GIS ISIS Network == <gis_layer>
Read GIS XP Network == <gis_layer>
Read GIS 12D Network == <gis_layer>

(Optional)

Reads the location of external 1D scheme nodes and channels. Supported 1D schemes are Flood Modeller (formerly known as ISIS), XP-SWMM or 12D Solution's DDA. 1D Nodes and Channels are referred to as Nodes and Units in Flood Modeller; Nodes and Links in XP-SWMM; and Pits and Pipes in 12D). The nodes in the GIS layer(s) are required for linking the external 1D scheme to TUFLOW. The channels are required only if using Read GIS X1D WLL to integrate 1D and 2D results in the map output. The only attribute required is the ID of the Flood Modeller node/unit, XP-SWMM node/link and 12D pit/pipe. See Section 8.2.3 for details on linking an external 1D scheme to TUFLOW, and Section 9.5 for integrating 1D and 2D results.

For linking with TUFLOW, the linked nodes must occur in this layer or in a Read GIS X1D Nodes layer. The nodes and channels (links) can be placed in separate layers, and split over several layers if desired. If desired, and the WLL feature is not being used, the layer can solely contain linked nodes and no channels.

Note: Flood Modeller IDs are case sensitive (because Flood Modeller is case sensitive), therefore, the IDs in Flood Modeller and the IDs in the 1d_x1d layer(s) must be identical (including case).

```
Read GIS X1D Nodes == <gis_layer>
Read GIS ISIS Nodes == <gis_layer>
Read GIS XP Nodes == <gis_layer>
Read GIS 12D Nodes == <gis_layer>
```

(Optional)

This command is superseded by Read GIS X1D Network, but continues to be supported for legacy models. It is now redundant as the nodes can be placed in a Read GIS X1D Network layer (the Read GIS X1D Network layer can only contain the nodes, and no channels, if so desired).

```
Read GIS X1D WLL == <gis_layer>
Read GIS ISIS WLL == <gis_layer>
Read GIS XP WLL == <gis_layer>
Read GIS 12D WLL == <gis_layer>
```

(Optional)

Reads the location of an external 1D scheme's WLLs. See Section 9.5 for integrating 1D and 2D results. The GIS layer is identical to that used for Read GIS WLL.

```
Read GIS X1D WLL Points == <gis_layer>
Read GIS ISIS WLL Points == <gis_layer>
Read GIS XP WLL Points == <gis_layer>
Read GIS 12D WLL Points == <gis_layer>
```

(Optional)

Reads the location of an external 1D scheme's WLL points for setting elevations and materials at points along WLLs. See Section 9.5 for integrating 1D and 2D results for more details. The GIS layer is identical to that used for Read GIS WLL Points.

```
Read Grid IWL == <grid_file>
```

(Optional)

Reads an ESRI ASCII (.asc) or a binary (.flt) grid defining the water level at the start of the simulation. The format is controlled by the file extension (use .flt for binary grids, otherwise an ASCII grid is assumed regardless of the file's extension).

Note: This command overwrites any IWL values set in the .tgc file for the same 2D cells.

For details see Section 7.6.1.2. Also refer to the command Read GIS IWL.

Read Grid RF == [<.csv_file> | <.nc_file>]

(Optional)

Reads a comma-delimited or NetCDF (.nc) file to define the rainfall applied directly to 2D cells over time. The .csv file is an index file that references the grids in either ESRI ASCII (.asc) or a binary (.flt) format. The .nc file contains all the rainfall grids over time within the one file. Refer to Section 7.4.3.3 for further information.

Note that the Read Grid RF command is located within the .tcf file, unlike the Read GIS RF and Read RowCol RF commands which are located within the .tbc file. This allows for the rainfall to be applied across all domains if multiple 2D domains are being used. The Read Grid RF command is not domain dependent whilst the .tbc file is.

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

(Optional)

Reads a text file containing Manning's n values for different materials (land-use types).

Two file formats (.tmf and .csv) are available and are discussed in detail in Section 6.9. Multiple materials files are also able to be specified.

If a second argument, <n_factor> is provided, this value is used to factor all Manning's n values. For example, to increase all Manning's n values by 10%, enter "Materials File == My_Materials.tmf | 1.1".

Presently only available for Bed Resistance Values == MANNING N based bed resistance values. Can be extended to CHEZY and MANNING M upon request.

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.

Refer to Section 7.6.2 for further information.

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 7.6. Initial water levels may also be read directly from an .asc or .flt grid using the command Read Grid IWL.

Read Soils File == <file.tsoilf>

(Optional)

Reads a text file containing an integer soil ID, infiltration method and soil parameters for different soil types. Refer to Section 6.10.4 for more information.

Reveal 1D Nodes == [ON | {OFF}]

(Optional)

When set to ON, displays the hidden 1D nodes within a 2d_bc 2D link line for models containing multiple 2D domains. Refer to Section 8.3.1 for more information.

SA Minimum Depth == [{0.1} | <depth in m>]

(Optional)

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

SA Proportion to Depth == [{ON} | OFF]

(Optional)

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

Proportioning to depth provides improved performance if using as a side inflow, especially for dam break type analyses, and reduces the effect of any duplicated routing when applying local hydrographs from a hydrology model.

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 the 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.

Set IWL == [<value> | AUTO | {0.0}]

(Optional)

If a <value> is specified, 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, Read RowCol IWL or Read Grid IWL.

If AUTO is specified, sets the IWL in 1D and 2D domains to the value of the water level boundary in the model at the start of the simulation. If the model has more than one water level boundary, and the starting level is different, an ERROR 0037 occurs. This feature only works for HT and HS boundaries. Note that the initial water level is only applied to 1D nodes and 2D cells that have been allocated a zero IWL (this is the default value). This means that Read GIS IWL can still be used to set the IWL in other parts of the model, such as a lake. Note that if Read GIS IWL sets a zero IWL, then this will be overridden by the AUTO value.

Note: This command overwrites any IWL values set in the .tgc file for the same 2D domain.

Set Route Cut Off Type == [{Depth} | Velocity or V | Hazard or VxD or Z0]

(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. Depth, velocity and hazard options are available. 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 9.5.1 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.

Set Route Cut Off Values == <y1, y2, ...>

(Optional)

Sets the cutoff values for the evacuation route feature (see Section 9.5.1) 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.

Set Variable <name> == <value>

(Optional)

In any TUFLOW control file use of this command defines a variable's name and value. Wherever you want to refer to the variable, the variable's name must be bounded by "<<" and ">>" characters.

Note, that when using Set Variable do not use <<…>> to bound the variable's name as is required to reference the variable in the control files.

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

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

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

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

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

If Event ~e1~ is set to "Q100", the output will be directed to ..\results\Q100_OpA.

Variables may be set to any characters and can be referred to as often as needed in any control file and in other files such as the .tmf and .toc files. At present you can't use variables in the bc_dbase.csv files, but future builds may offer this feature.

For example the lines below give some idea of how you could use variables that explains how scenarios and events are automatically set as available variables. The commands below set the model's cell size, timestep and output interval, and sets the folders for outputting check files and results according to Scenario 1 which is the model's grid resolution (cell size), ie. one of "0.5m", "1.0m" or "2.0m".

.tcf file entries:

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

! Set times
Start Time == 0
End Time == 0.5
Timestep == <<2D_Timestep>>
Write Check Files == ..\check\<<~s1~>>\
Output Folder == ..\results\<<~s1~>>\
Screen/Log Display Interval == <<LogInt>>
```

.tgc file entry:

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

Note that If Scenario and If Event operate at a higher level than Set Variable, so that they can be used to set different values for the same variable.

Snap Tolerance == [{0.001} | <snap_distance>]

(Optional)

Sets the search distance in metres or feet for detecting whether two GIS objects are connected (snapped). Note that this command can be repeated to change the tolerance only for a single layer. For example the below could be used set the tolerance and then return it to the default value:

```
Snap Tolerance == 0.01
Read GIS Network == ..\model\gis\1d_nwk_example_L.shp
Snap Tolerance == 0.001
…
```

Shallow Depth Stability Factor == <SF>

(Optional)

Note: Legacy command. No longer required or recommended for use subsequent to implementation of Wetting and Drying == ON METHOD B. See TUFLOW 2010 manual for details.

SHP Projection == < .prj_file >

(Optional)

This command is similar to the MI Projection command that sets the .shp file projection for checking whether input layers are in the same projection, and for setting the projection of all output layers (eg. check layers).

Example:

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

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

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" (refer to Section 12.4.2).

This command takes precedence over the -slp option (see Table 11-2) 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 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 8.3.1. If there is only one 2D domain, this command is optional.

<format> 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.

This command can be defined for different output formats by including the output format extension on the left of the command. For example, to set the start time for XMDF output to 1hr use:

```
XMDF Start Map Output == 1
```

This output format specific functionality is also available for commands End Map Output, Map Output Interval and Map Output Data Types. Refer to Section 9.4.2.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

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.

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.

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 6.3.1 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.

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.

SX Flow Distribution Cutoff Depth == [{0.005} | <depth_m>]

Sets the depth of water below which an SX boundary cell will not receive flows from the connected 1D element.

Prior to the 2016-03 TUFLOW release this was set to 0.0, as soon as the cell was wet, flow was applied. Set to 0.0 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 | <dZ_limit_in_m>]

(Optional)

If ON (the default), checks whether the minimum ZC elevation at or along a SX object (see Table 7-4 and Table 7-5) 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 7-5), 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 OFF, higher ZC elevations are allowed and no automatic lowering of ZC elevations occurs.

A value may be entered to set a maximum permitted change in ZC elevation caused by the use of any "Z" flags for 2D SX links. For example, if SX ZC Check == 0.5 is specified, then if any 2D cell is lowered by more than 0.5m an ERROR 2050 occurs. This allows the modeller to automatically lower 2D cells within the specified limit, but flag an ERROR 2050 if there is a substantial change, which usually indicates there is something inconsistent between the 1D channel bed and the 2D topography. **It is strongly recommended that if using the SX Z flag, that this option is specified.**

Time Series Output Format == [PRE 2013 | {2013}]

(Optional)

The default (as of the 2013-12 release) is to output 1D .csv files to a csv folder in a slightly new format. However, the output of time-series and other data has been superseded by the Output Approach for the 2016-03 release. This command is provided for legacy models or for where the modeller prefers to use this output arrangement.

To output in the formats prior to the 2013-12 release, specify Time Series Output Format == PRE 2013 in the .tcf file.

Time Series Output Interval == <time_in_seconds>

(Mandatory if Output Interval (s) not specified)

The output interval in seconds for time series based output (PO and LP).

Either Output Interval (s) or Time Series Output Interval must be specified otherwise an ERROR 0046 message is output. The default values for these intervals is the computational timestep, and by not specifying values can cause excessive amounts of memory to be allocated, sometimes causing undesirable results!

Time Output Cutoff [Depths | VxD | <Hazard>] == [{OFF} | <v1, v2, ...>] (Optional)

If one or more comma or space delimited values are specified using the above command, the map output will contain additional time output for each value v1, v2, … as follows. The type of the values is controlled by the [Depths | VxD | <Hazard>] setting (only one of these can be specified). Depths is the depth of water, VxD is the velocity times depth product, and <Hazard> must one of the hazard categories as documented in Table 9-11. VxD and Z0 are one in the same.

1. The time that a cell first experiences a value greater than each of <v1, v2, …>, the simulation time of exceedance is retained, and included in the map output. If using the .dat output format, the data is accessed using a simulation time of 100,000 + v in the .dat file, where v is v1, v2…. This output is useful for mapping flood warning times for different depths of inundation or when VxD or hazard categories are exceeded. When output as a grid (.asc or .flt) map output format, the file extension given to this output is _TExc_<cutoff>.
2. The duration of time that a cell is inundated above each of <v1, v2, …> is also retained. For .dat file map output the durations are stored under the time of 200,000 + v, where v is v1, v2…. This output is useful for mapping duration of inundation above v1, v2, …. When output as a grid (.asc or .flt) map output format, the file extension given to this output is _TDur_<cutoff>.

Note: If this command is specified more than once, the last one will prevail.

The command Time Output Cutoff Hazards is also accepted as an alias to Time Output Cutoff VxD.

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 or one hour are recommended. For example, 0.5, 1, 2, 3, 5, 6, 7.5, 10, 12, 15, 20, 30, 45, 60, etc. seconds.

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 Maximum Increase (%) == [{0.5} | <value_as_a_%>]

(Optional)

This command controls the maximum rate at which an adaptive timestep, using the command Maximum Courant Number, can increase. There is no limit to how quickly the timestep can decrease. See Section 3.4.4 for more information.

TSF Update Interval == [{0} | <interval_in_sec>]

(Optional)

Sets the interval in seconds to update the .tsf file while a simulation is running. If set to 0 or less (the default) the .tsf file is only updated at the start and the end of the hydrodynamic calculations.

TUTORIAL MODEL == [ON | {OFF}]

(Optional)

When set to ON, allows simulation of the Tutorial Models without the need for a TUFLOW license. For further information refer to Section 2.4.

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 <= 0)
- 1 = Low Hazard (H <= 0.75)
- 2 = Moderate Hazard (H <= 1.25)
- 3 = Significant Hazard (H <= 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.

If the UK Hazard Land Use is not specified, and the default UK Hazard Formula is used, the debris factors for the CONSERVATIVE land use are assumed.

In the table below, the V > 2m/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 > 2m/s criteria.

Guidance on debris factors (DF) for different flood depths, velocities and dominant land uses

Depths	Pasture/Arable	Woodland	Urban	Conservative
0 to 0.25 m	0	0	0	0.5
0.25 to 0.75 m	0	0.5	1	1
d>0.75 m and/or v>2	0.5	1	1	1

Ref: FD2321/TR1 Table 3.1

Units == [{METRIC} | ENGLISH | IMPERIAL | US Customary]

(Optional)

By default, TUFLOW uses metric units. Full support for US Customary Units (English or Imperial – they are identical) is available by setting Units == US Customary, Units == English or Units == Imperial. The default settings for all inputs are the same as for metric, but converted to their US Customary units' equivalent. Where the manual refers to a default value in metric, TUFLOW will use the equivalent value in US Customary Units. The input and output units are as follows:

- Length: feet
- Area: square feet
- Rainfall, Initial Loss and Continuing Loss(/h): inches
- Catchment Area: square miles
- Constant eddy viscosity value: ft^2/s
- For determination of hazard categories (Z0 and ZUK0) the values are based on using feet and seconds.

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).

Setting to "WARNING" will issue a WARNING message but allow TUFLOW to continue to run. It is not recommended that the WARNING option be used other than for backward compatibility.

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

(Optional)

If set to ON, forward slash (/), rather than backslash (\), is used in the text output files contain file paths (e.g .tlf, .qgis, .tpc, .wor). For LINUX systems forward slash must be used, while for Windows either can be used.

Verbose == [ON | OFF | {LOW}]

(Optional)

If set to ON, outputs to the console window and .tlf file even more information! To see all information, set to ON. LOW outputs information to the screen and the .tlf file at a level of detail between ON and OFF. The default setting for the 2016-03 release is LOW.

Viscosity Coefficients == [{0.5, 0.05} | <smag, cons>]

(Optional)

Sets the eddy viscosity coefficient(s) (see Section 3.6).

If Viscosity Formulation == SMAGORINSKY (the default), two values can now be specified, the first being the Smagorinsky coefficient, and the second a constant eddy viscosity component. The default values are 0.5 and 0.05m^2/s respectively. Note that the 0.5 is dimensionless and is the coefficient for the Smagorinsky equation. The Smagorinsky equation recalculates the eddy viscosity coefficients every timestep at every 2D cell, with the resulting coefficients varying depending on the spatial difference in velocity magnitude and direction (see Section 3.6). The constant component remains unchanged over time and space. If a second value is not specified, the constant component is zero.

If using the Viscosity Formulation == SMAGORINSKY, the coefficient is typically between 0.06 to 1.0.

Prior to the 2012-05 release, the default values were 0.2 and 0.1m^2/s.

Viscosity Formulation == [CONSTANT | {SMAGORINSKY}]

(Optional)

Sets the viscosity formulation (see Section 3.6). Options are:

- "CONSTANT" – the viscosity coefficient remains constant
- "SMAGORINSKY" – applies the Smagorinsky formula

The default is SMAGORINSKY.

VG Z Adjustment == [{MAX ZC} | ZC | ZH]

(Optional)

MAX ZC is the default setting. This forces 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.

The ZC option

The ZH option provides backward compatibility for models using the original VG adjustment of Zpts based on changing the ZH values.

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 or falls below the

negative of the Instability Water Level. Switching this option off reduces the computation time very slightly.

Wetting and Drying ==
[{ON METHOD B} | ON | ON NO SIDE CHECKS | OFF]
(Optional)

Controls the wetting and drying method.

The ON METHOD B approach (default) introduced an enhanced wetting algorithm in the 2012-05 release that provides significant improvement to inflows on steep areas (eg. direct rainfall models), whilst maintaining low mass error, greater stability and often larger timesteps. Use of this approach does not require use of the Shallow Depth Stability Factor that was previously automatically invoked for direct rainfall models. This method makes an estimate of the likely velocity that will occur when a cell side first wets and feeds this information into the solution matrices. Previously, the velocity used was that from the previous timestep which was zero as the cell side was dry. The zero velocity essentially created a frictionless slope and would cause a surge of water, albeit very shallow, when the cell side first wets. This was not usually a major issue, however, with direct rainfall models all cell sides can become wet in one timestep and if the terrain is steep a significant surge and unacceptable mass errors would occur. This method largely overcomes this effect.

The (pre 2012-05 default) ON approach 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.

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.

Write Check Files [{ALL} | NONE | INCLUDE | EXCLUDE] == <prefix_list_or_file_prefix>

(Optional)

Creates GIS check files in .mid/.mif or .shp format and text .csv files for quality control checking of model input data. Refer to Section 12.9 for further details on the check files produced. The options available for this command can be used to control which check layers are written.

The EXCLUDE or INCLUDE options allow for a space delimited list of file prefixes to be specified to exclude or include GIS layers from being written. Prefixes must be the same as those used by TUFLOW. For example, "zpt" would apply to the zpt_check layer. To exclude/include more than one layer ensure there are spaces between the prefixes. If EXCLUDE or INCLUDE occurs more than once, the latter occurrence prevails.

Examples:

```
Write Check Files EXCLUDE == zpt uvpt  ! excludes writing of the zpt_check and uvpt_check layer
Write Check Files INCLUDE == dem_Z  ! will only write the Grid of the model's ground elevations to the same location as the .tcf file.
```

The ALL option requires no prefix list. All check files will be written to the same folder as the .tcf file. Specification of the ALL option will nullify any prior occurrence of an EXCLUDE or INCLUDE list; this is useful if you wish to write no or all check files for one particular run – simply add Write Check Files ALL to the end of the .tcf file.

Alternatively, the Write Check Files command may be used without options to add a prefix to all check files or specify a location in which to write the files. If <file_prefix> is omitted or ends in a "\" to indicate a folder, the .tcf filename (without the .tcf extension) is prefixed to each check file. <prefix_list> can include a folder path that is normally set to the check folder. See the examples below for this subtle difference.

Examples:

```
Write Check Files ALL ! writes all check files with no prefix to the same location as the .tcf file.
Write Check Files == C:\tuflow\check\2d  ! writes all check files to the folder "C:\tuflow\check" and prefixes with "2d"
Write Check Files == C:\tuflow\check\  ! writes all check files to the folder "C:\tuflow\check" and prefixes with the .tcf filename
Write Check Files == C:\tuflow\check  ! writes all check files to the folder "C:\tuflow" and prefixes with "check"
```

The NONE option is similar to the ALL option and requires no prefix list. No check files will be written and specification of the NO option will nullify any prior occurrence of an EXCLUDE or INCLUDE list.

Examples:

```
Write Check Files NONE ! will suppress the writing of all check files
```

Specifying the Write Check Files command in the .tcf file will now automatically also write the 1D check files. There is no need to specify Write Check Files in the .ecf file unless a different folder path for the files is desired.

This command can be used within an Output Zone definition block to change the setting from that for the entire model map output.

Write Empty GIS Files == [{} | <folder> | <file_format>]

(Optional)

Creates empty 1D and 2D GIS files in either mid/.mif or .shp format. Each 1D layer as described in Table 2-4 is produced with the required attribute definitions pre-defined, but containing no geographic objects. Provided the MI Projection or SHP Projection command has been previously specified, each layer has the correct GIS projection.

The layers are prefixed using the prefixes defined in Table 2-4 and are given a suffix of "_empty". If <folder> is specified, the GIS files are located in the folder, which must already exist.

If the second argument is MIF or is omitted (as for the first example below), the empty files are in the .mif/.mid format. Use SHP to output empty files in the .shp format, as shown in the second example below.

After writing the files, TUFLOW stops executing.

Examples:

```
Write Empty GIS Files == ..\model\mi\empty ! Writes empty GIS layers in .mif/.mid
format
Write Empty GIS Files == ..\model\shp\empty | SHP ! Writes empty GIS layers in .shp
format
```

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 GIS 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.

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. An .erf file is also written for the 1D components if there is 2D/1D dynamic linking. 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.

Refer also to the command Write Restart Filename.

Write Restart File Version == 2

(Optional)

Uses a more detailed dump of the 2D domains that will result in a more precise restart.

Write Restart Filename == [{INCLUDE TIME} | OVERWRITE]

(Optional)

Command to control whether restart files are overwritten or are time-stamped.

If INCLUDE TIME (the default) is specified, .trf and .erf filenames are time-stamped and written for each restart output time, and the files are placed in a folder called "trf".

OVERWRITE is the case for all prior versions of TUFLOW where the .trf and .erf files are overwritten each time restart files are written. If Defaults == PRE 2013 is set the default setting is OVERWRITE.

To output restart files at regular intervals use the command Write Restart File Interval.

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 Flood Modeller or XP-SWMM.

Note that Write Check Files must be also be specified (and not set to NONE).

XF Files == [{ON} | OFF]

(Optional)

Sets the global default for whether or not to automatically generate XF files. See Section 4.10.

XF Files Include in Filename == <text>

(Optional)

Appends <text> to the end of XF filenames. This command prevents the reprocessing of the DEM each time a different model utilising the same DEM is run.

XMDF Output Compression == [{ON} | OFF | <level>]

(Optional)

Sets the file compression level of XMDF map output files. If set to ON, the default, compression level 1 is applied. A maximum compression level of 9 is allowed. The greater the compression the smaller the file sizes, but the slower the access speed when writing and reading the file. Testing has indicated that a compression level of 1 reduces file sizes by 70 to 80% with little change in access speeds.

Zero Negative Depths == [{ON} | OFF]

(Optional)

This is a legacy command provided for backward compatibility. Setting to ON zeroes depths at cell corners for map output if negative. The negative depth could arise in old builds of TUFLOW when interpolating the water level at the cell corners from the surrounding cell centres, due to the ZH Zpt being higher than the interpolated water level.

Setting to OFF disables this command and should only be effective if the original cell interpolation method (Map Output Corner Interpolation == METHOD A) applies. Using the OFF option is not recommended.

Zero Rainfall Check == [{ERROR} | WARNING]

(Optional)

Introduced for the 2016-03 release to set the level for Message 2460 that reports whether the f1 and/or f2 attribute is set to zero for a 2d_rf layer. For the 2016-03 release the message is treated as an ERROR and the simulation will terminate, unless Defaults == PRE 2016 is set or this command is set to WARNING.

Prior to the 2016-03 release this message was only issued as a WARNING. For further information see the Message 2460 Wiki Page.

ZP Hazard Cutoff Depth ==
[{0.01} | <value> | <value_ZPA> <value>_ZPC <value>_ZPI]
(Optional)

This command can be used when specifying the ZPA, ZPC, and ZPI hazard map outputs, based on the People Hazard categories within the Australian Rainfall and Runoff (ARR) Project 10 Stage One report. Refer to Table 9-10 for more information.

If one value is specified, the cutoff depth to define when the Safe category applies is the same for ZPA, ZPC and ZPI. If three values are specified, these are the cutoff depths for ZPA, ZPC and ZPI respectively. The default is 0.01m, ie. if the depth is below 0.01m (1cm), the hazard category is Safe for ZPA, ZPC and ZPI.

Zpt Range Check == [{-9998, 99998} | <zmin>,<zmax>]
(Optional)

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

Appendix B 1D (.ecf) Commands

Note: These commands can be placed in the .ecf file, in the .tcf file within a Start 1D Domain block or in the .tcf file by preceding the command with "1D", for example: 1D Read GIS Network.

BC Database
BC Event Name
BC Event Text
Bridge Flow
Bridge Zero Coefficients

Create Nodes
CSV Format
CSV Maximum Number Columns
CSV Time
Conveyance Calculation
Culvert Critical H/D
Culvert Flow
Culvert Zero Coefficients

Depth Discharge Database
Depth Limit Factor

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

M Channel Approach
M11 Network
Manhole Approach
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

Order Output
Output Data Types
Output Folder
Output Interval
Output Times Same as 2D

Pit Channel Offset
Pit Default 2D Connection
Pit Default Road Crossfall
Pit Inlet Database
Pit Search Distance

Read File
Read GIS BC
Read GIS IWL
Read GIS Manhole
Read GIS Network
Read GIS Pits
Read GIS Table Links
Read GIS WLL
Read GIS WLL Points
Read Operating Control File

S Channel Approach
Set IWL
Set Variable
Start Output
Storage Above Structure Obvert
Structure Flow Levels
Structure Losses
Structure Losses Approach
Structure Routines

Taper Closed NA Table
Timestep
Trim XZ Profiles

Vel Rate Creep Factor
Vel Rate Limit
Vel Rate Limit Minimum

Weir Approach
Weir Flow
WLL Approach
WLL Automatic
WLL No Weirs
WLL Vertical Offset
WLLp Interpolate Bed
Write Check Files

XS Database

BC Database == <.csv_file>

(Optional if in .tcf file)

Sets the active BC Database file as described in Section 7.5. 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> (seeBC Event Text) occurs in the BC Database. See Section 7.5.2 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 7.5.2 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.

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, the default 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).

Refer to Section 5.7.2 for further information.

Bridge Zero Coefficients ==
[{DEFAULT} | OFF | <Cd>, <DLC>, <ELC>, <XLC>]

(1D & 2D/1D. Optional)

Changes the default BB bridge channel coefficients applied to zero (0.0) attribute values in the 1d_nwk layer(s). This command is not used for B bridge channels.

The four values and their associated 1d_nwk attributes are:

Cd Bridge deck pressure flow surcharge coefficient specified by the HConF_or_WC attribute (Default = 0.8).

DLC Bridge deck energy loss coefficient when fully submerged and not under pressure flow. Specified by the WConF_or_WEx attribute (Default = 0.5).

ELC Unadjusted entrance loss coefficient specified by the EntryC_or_WSa attribute (Default = 0.5). The entrance loss coefficient applied is adjusted every timestep according to the approach velocity as discussed in Section 5.7.2.4.

XLC Unadjusted exit loss coefficient specified by the ExitC_or_WSb attribute (Default = 1.0). The exit loss coefficient applied is adjusted every timestep according to the departure velocity as discussed in Section 5.7.2.4.

For example, Bridge Zero Coefficients == 0.75, 0.3, 0.5, 0.8 changes the default value for Cd, DLC, ELC and XLC used to set any zero (0) attribute values in 1d_nwk layer(s).

DEFAULT (which is the default setting) will set any zero values to the default values mentioned above.

OFF will allow a zero value to be applied to the ELC and XLC loss coefficients. Cd and DLC cannot be non-zero, therefore, if OFF is specified these will always have their default value above applied if the 1d_nwk attribute is zero.

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.

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 Maximum Number Columns == <max_col>

(1D & 2D/1D. Optional)

Used to specify a maximum number of columns for 1D .csv output files. The default setting is limitless. Refer also to the .tcf command CSV Maximum Number Columns.

CSV Time == [{DAYS} | HOURS | MINUTES | SECONDS]

(1D & 2D/1D. Optional)

Specifies the time values of 1D .csv outputs. The default is DAYS but can be changed to HOURS, MINUTES or SECONDS.

The .tcf command CSV Time can be specified to apply to both 1D and 2D .csv outputs.

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 (the default), a parallel channel is created for every X (distance across section) value. This approach does not cause conveyance reducing with height warnings.

Refer to Section 5.10.3.

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.

The default is OFF (ie. infinitely large H/D). Three options are additionally available:

- 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 (introduced for Build 2002-07-AC) is an adaptation of Method A to include regimes K and L (see Figure 5-1). 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 5-1).

Method D (introduced for Build 2007-07-AA) includes further improvements on Method C and corrects a bug for Regime E.

Method E is the current default and incorporates minor improvements for transitioning between Regimes A and B, and between inlet and outlet controlled regimes, for circular culverts.

Methods A, B, C and D are provided for backward compatibility.

Culvert Zero Coefficients == [DEFAULT | {OFF} | v1, v2, v3, v4, v5]

(1D & 2D/1D. Optional. Alias: Zero Culvert Coefficient)

Sets the default culvert coefficients if they are set to zero (0) in the 1d_nwk layer(s). 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.

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, Culvert Zero 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.

Depth Discharge Database == [<depth_discharge_dbase.csv>]

(1D & 2D/1D. Mandatory if any Q channels.)

Specifies the depth discharge database file that references the depth-discharge curves for non-operated Q channels. Mandatory if the model has any Q channels. Performs the same function as Pit Inlet Database. The same database file can be used for both Q channels, Q pits and pump channels.

Depth Limit Factor == [{10} | <value>]

(1D & 2D/1D. Optional)

Sets the depth limit for detecting instabilities. The default 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 5.10.4 for a more detailed discussion.

Flow Calculation == [Method A | {Method B}]

(Optional)

Method A is the original ESTRY channel flow routine.

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 (a similar check is used for 2D domains – see Section 6.3.1).

Only set to OFF for backward compatibility.

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.

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 5.10.7 for details on the process used. Set to OFF to disable this feature and force every channel to have a cross-section.

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.

M Channel Approach == [Method A | {Method B}]

(1D Only. Optional)

Method A is the original interpolation approach which uses a 4 value interpolation routine that, especially along the 0 flow diagonal, might not interpolate as accurately as using a triangular (3 point) technique. The main issue with the 4 point routine is that a slightly positive or negative flow of the wrong sign may result when interpolating close to the zero flow diagonal.

Method B (introduced for Build 2012-05-AB) uses an improved method. It uses a triangular interpolation with the long side of the triangle parallel to the zero flow diagonal, and does not experience the problems Method A has when interpolating close to the zero flow diagonal.

For backward compatibility to 2012-05-AA and prior releases, use M Channel Approach == Method A or Defaults == Pre 2012.

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 1d_nwk attributes (see Table 5-2). 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.

Manhole Approach == [METHOD A | METHOD B | {METHOD C}]

(Optional)

Method A is the original approach which has been found to be too conservative (ie. higher losses and therefore higher flood levels) as indicated by users. See Section 5.12.5.1 for further details.

Method B (introduced for Build 2012-05-AA), incorporates few bug fixes found in Method A.

Method C (the default) incorporates further bug fixes affecting manholes using the Engelund approach for calculating losses.

If using METHOD B, sensitivity testing the effects of METHOD C versus METHOD B should be carried out to check for any unacceptable differences.

Manhole Default Loss Approach == [NONE | {ENGELUND} | FIXED]

(Optional)

Sets the default approach to be used at automatically generated manholes, and manholes where no loss approach is specified. The default is ENGELUND (see Section 5.12.5 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 5.12.5 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 Engelund 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 5.12.5 for more information).

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)

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 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.

The percentage (%) 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 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.

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 .eof file.

Output Data Types == [{H Q S V} | <data_types>]

(1D & 2D. Optional)

Similar to the .tcf map output command Map Output Data Types, this command controls which 1D data types to output, The options are:

- **A** for flow area at channels
- **E** for energy
- **H** for water level
- **Q** for flow at channels
- **S** for structure and grouped structure output (see Section 9.3.2)
- **V** for velocity

Output Folder == <folder>

(1D & 2D. Optional)

Redirects all ESTRY (1D) output data to another folder. Typically used to write output to your local 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), which is 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).

The default location for 1D output is that specified using Output Folder in the .tcf file for 2D/1D models.

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.

Note: This command is rarely used in lieu of Time Series Output Interval. Recommend not using.

Output Times Same as 2D == [{ON} | OFF]

(2D/1D Only)

For 2D/1D models, 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 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 13.2.3).

This command is ignored for 1D only (ESTRY) models.

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_1D_2D>]

(1D & 2D/1D. Optional)

Sets the default value for the 1d_nwk Conn_1D_2D attribute. For example, if set to SX, then if the 1d_nwk Conn_1D_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_1D_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 Default Road Crossfall == <slope>

(1D & 2D/1D. Optional)

Increases the depth at Q pits based on the height of an imaginary triangle of the road cross-section with a crossfall slope of <slope>. <slope> is Vertical/Horizontal as a fraction (not percentage).

The imaginary triangle has the same area as the vertical flow area in the 2D cell the pit is connected to (ie. the triangle's area is the depth in the 2D cell times the width of the cell). Once the horizontal width of the triangle is greater than the width of the 2D cell, the formula changes to give an equivalent area based on a trapezoid consisting of the triangle plus a rectangle for the remaining area in excess of the triangle.

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.

Performs the same function as Depth Discharge Database. The same database file can be used for both Q channels and Q pits.

See Section 0 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.

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.

This command can be used to redirect file(s) up to a maximum of ten levels.

Read GIS BC == <gis_layer>

(Mandatory)

Reads the location and attributes of 1D model boundary conditions as described in Section 7.3.

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.

Read GIS Manhole == <gis_layer>

(1D & 2D/1D. Mandatory)

Reads manhole locations and attributes from a GIS 1d_mh layer as described in Section 5.12.5.2. 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 5-22.

Read GIS Network == <gis_layer>

(1D & 2D/1D. Mandatory)

Reads channel and node locations and attributes from a GIS 1d_nwk layer as described in Section 5.4 and 5.11. 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 Node == <gis_layer>

(1D & 2D/1D. Optional)

Reads node locations and attributes from a GIS 1d_nd layer as described in Table 5-17. Any number of 1d_na layers may be read by repeating this command. This is an alternative option to the 1d_nwk Read GIS Network command, but applies to 1D nodes only.

Read GIS Pits == <gis_layer>

(2D GPU only. Optional)

Reads virtual pipe layer for a GPU model. This functionality is described in Section 10.6.7. If using the virtual pipe functionality a Pit Inlet Database is required.

Read GIS Table Links == <gis_layer>

(1D & 2D/1D. Optional)

Reads links to tabular input of cross-section profiles (1d_xs – see Section 5.10), cross-section hydraulic parameters (see Table 5-13), nodal surface areas (1d_na – see Section 5.11.3) and bridge loss coefficients (1d_bg – see Section 5.7.2). 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.

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 9.5 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 9.5.3 for further information.

Read Operating Controls File == <.toc_filename>

(1D Only. Optional)

Directs input to the Operation Controls file (.toc) containing operating rules applied to hydraulic structures, pumps and other controllable devices. More than one .toc file can be set up and accessed should there be a need to break the control definitions into several files (for example, all pump controls could be placed in one file and sluice gate controls in another).

Refer to Section 5.8 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 (the default) was introduced 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.

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.

Set Variable <name> == <value>

(Optional)

In any TUFLOW control file use this command to define a variable's name and value. Wherever you want to refer to the variable, the variable's name must be bounded by "<<" and ">>" characters.

Refer to Set Variable description within tcf commands.

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.

Note: This command is effectively redundant in lieu of using Start Time Series Output.

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. The default is to apply 5% of the maximum surface area.

The CHANNEL WIDTH option uses the top width of B and R channels and the diameter for C channels (see Section 5.11.2). 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), 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 m^2 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 (16.4ft) above the highest obvert. Should water levels exceed this height, use Depth Limit Factor to extend the table further.

Structure Flow Levels == [{ENERGY} | WATER]

(1D Only. Optional)

For calculating structure flows sets whether to use energy or water surface levels as the global default in the flow calculations. The default is to use ENERGY. The global default can be overridden channel by channel using the "E" or "H" optional flag for the 1d_nwk Type attribute (see Table 5-1).

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 5.7.6 for details.

This setting can be overridden using an A (adjust) or F (fix) flag for B, C and R channels.

If set to ADJUST EXCEPT BG TABLE (the default) only adjusts 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 A is the original ESTRY routine.

METHOD B (the default) is an enhancement over METHOD A. It 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.

Structure Routines == [ORIGINAL | {2013}]

(1D & 2D/1D Only. Optional)

ORIGINAL only allows structures available prior to Build 2013-12-AA to be available to force users to limit their use of structures in a legacy model.

2013 allows the use of the original structures, plus the many new structures introduced for Build 2013-12-AA. This includes access to operational structures and any other new structures added since Build 2013-12-AA.

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.

Timestep == <timestep_in_seconds>

(1D & 2D/1D Only. Mandatory for 1D only.)

Specifies the computational timestep of the simulation in seconds. Value must be greater than zero. Timesteps that divide equally into one minute or one hour are recommended. For example, 0.5, 1, 2, 3, 5, 6, 7.5, 10, 12, 15, 20, 30, 45, 60, etc. seconds.

A 1D timestep different to the timesteps of the 2D domains may be specified for 1D/2D models, however the 1D timestep 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.

Trim XZ Profiles == [ON | {OFF}]

(1D & 2D/1D. Optional)

Trims the XZ profile extracted from Flood Modeller .dat files so that the treatment at the ends of the cross-section profile is similar to that used by Flood Modeller. 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 Flood Modeller, this command should be set to ON.

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.

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 Vel Rate Creep Factor value. An "L" is shown in the second space after velocity and flow time output in the .eof file, and also in the _TSF.mif output, indicating if the velocity rate limit algorithm was applied. 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.

Weir Approach == [{Method A} | Method B]

(1D & 2D/1D Only. Optional)

This command specifies the approach taken to calculate submerged flow for 'W' type weirs (refer to Table 5-7). For all other weir types, the approaches used are discussed in Section 5.7.3.

Method A (the default) utilises the Bradley submergence approach which after further analysis is the preferred approach.

Method B was initially introduced to provide better stability for weirs embedded within 2D domains. In some situations, this approach was found to not converge very well and caused large head drops.

Weir Flow == [Method A | Method B | {Method C}]

(1D & 2D/1D. Optional)

Method A is the original ESTRY routine and only applies to W weirs.

Method B is not recommended – only applies to W weirs.

Method C (the default and introduced for the 2016-03 release), applies Method A for the original W weirs, provides some minor improvement to WW weirs, and otherwise has no influence for all other weir types introduced for the 2013-12 release.

Note that for the 2016-03 release there is a general improvement to all new weirs introduced for the 2013-12 release when the weirs became drowned out. Previously it was possible for an instability (usually associated with a large head drop) and/or presence of NaNs in the results. The new approach does not experience this issue and is significantly more stable. There is no backward compatibility switch for this change.

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 (the default) allows the use of elevation points and material values to more accurately map and animate 1D results. See Section 9.5 for details on Method B and the TUFLOW 2010 Manual on Method A (noting that Method A is no longer supported).

WLL Automatic == [CULVERTS | {OFF}]

(2D/1D. Optional)

If set to CULVERTS, automatically generates 1D WLLs along culverts (C, R and I 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 vertices 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 affects 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 the TUFLOW_to_GIS utility (see Section 15.2.1) to be based on that modelled, rather than that a DTM using WLLp values (which may sometimes occur above the water surface!). Also helps to 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.

Write Check Files [{ALL} | NONE | INCLUDE | EXCLUDE] == <prefix_list_or_file_prefix>

(1D & 2D/1D. Optional)

Creates GIS check files in .mid/.mif or .shp format and text .csv files for quality control checking of model input data. A list of check files written are described in Table 12-2.

Refer to the Write Check Files command in the .tcf file for an explanation of the various options and for examples.

Specifying the Write Check Files command in the .tcf file will now automatically also write the 1D check files for 1D/2D linked models. There is no need to specify Write Check Files in the .ecf file unless a different folder path for the files is desired.

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 a Flood Modeller data file containing XZ cross-section profiles – also see Trim XZ Profiles.
- .pro indicates a Flood Modeller 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 1d_nwk attributes as discussed in Table 5-2Table 5-11.

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.

Appendix C .tgc File Commands

Allow Dangling Z Lines

Cell Size
Create TIN Zpts

Default Land Z

Grid Approach
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 Objects
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 GIS Zpts Gauge Output
Read Grid
Read Grid Location
Read Grid Zpts
Read RowCol
Read RowCol Zpts
Read TIN Zpts

Set
Set Code and Clean Zpt
Set Route Cut Off Type
Set Route Cut Off Values
Set Variable
Set Zpt
Stop

Thin Line as Thick
TIN Angles
TIN Coincident Point Distance

Write GIS Domain
Write GIS Grid
Write GIS Zpts

ZC == MIN(ZU,ZV)
Zero Z Point

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).

Cell Size == <value>

(Mandatory if not specifying in .tcf file)

Sets the grid's cell size in metres. This overrides any value specified in the .tcf file. The cell size must be specified either using this command or in the .tcf file. See Section 3.3.1.

Create TIN Zpts [{} | WRITE TIN] [XF ON | XF OFF] == <gis_layer>

(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 6.8.4.

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>

(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.

Grid Approach == [Method A | {Method B}]

(Optional)

Method A was the default approach prior to Build 2013-12-AB. For iSP (single precision) builds, this method was found to occasionally generate a NaN (not a number) at a Zpt due to precision issues. This has been fixed for Method B.

Method B introduced for Build 2012-05-AE assigns values to Zpts that fall within the outer half of the DEM cells around the DEM perimeter. Previously, if processing a group of DEM tiles (ie. using multiple Read Grid Zpt commands, one for each tile), any Zpts lying within the outer half of the DEM perimeter cells were not assigned a value. The new approach assigns values to Zpts around the DEM perimeter based on an interpolation of the two DEM cells the Zpt lies nearest to. Provided tiled DEMs have a common edge (or overlapping edge), all Zpts near the DEM edges should now be assigned a value.

The Grid Approach command may be used any number of times within the .tgc file with the latest setting prevailing when a Read Grid command is processed. If associated .xf files exist from previous simulations, and Grid Approach set to Method B is applied, TUFLOW will automatically resample and replace the existing .xf files. The Zpt values will essentially be identical or very similar except for around the perimeter of the DEM as discussed above. This also applies for all Read Grid options (eg. also applies to "Read Grid Mat").

Method B is the default setting as of Build 2013-12-AB.

Grid Size (N,M) == <rows>, <columns>

(One GRID SIZE command or Read GRID Location is mandatory)

Sets the dimensions of the grid based on the number of rows and columns. The values entered must be integer values.

See also the command Read GRID Location which defines the size and location of a 2D domain based on a DEM.

Grid Size (X,Y) == <X_length>, <Y_length>

(One GRID SIZE command or Read GRID Location 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.

See also the command Read GRID Location which defines the size and location of a 2D domain based on a DEM.

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 flow paths 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.

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.

Also see Interpolate ZC, Interpolate ZHC, Interpolate ZUVC, Interpolate ZUVH

Interpolate ZUVC [{} | ALL]

(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).

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.

Also see Interpolate ZC, Interpolate ZHC, Interpolate ZUV, Interpolate ZUVC

Orientation == <XX_in_metres>, <YX_in_metres>

(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>, <OY>

(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 metres and <OY> the Y-coordinate.

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.

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: This command can now be used in redirected file(s) up to a maximum of ten levels.

Read GIS <option> == <gis_layer>
Read GIS Code [{} | BC] [{} | INVERT] == <gis_layer>

(Optional)

Reads the <option> from a GIS layer in .mif or .shp format. Except for the "Read GIS Code BC" combination, the first attribute (column) in the file must be the value of the <option>.

The following options are available in the following table.

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.

Any cell falling within/on an object is assigned the object's value. The object may be a region (polygon), line or point. For WrF and FLC the mid-sides of the cell are used rather than the cell centre. For all other options, the cell centre must fall within the region, or if the object is a point, the point must fall within the cell.

For .tgc Read GIS commands that require only one attribute (eg. Code, Mat, Soil, GWL, GWD, IWL, CnM, WrF, FLC, CWF, SRF, Zpt), this attribute no longer needs to be the first attribute. If the attribute is not in the first field position, the position can be specified by including it as an argument before the GIS filename.

For example, a buildings' outline GIS layer called "2d_buildings.shp" has four attributes:

"Prop_ID", "Material", "SRF", "Pad_Depth"

Rather than copy the layer three times and set up the appropriate attribute as the first column, the original layer can be used without modification using the following three .tgc commands.

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

If no number exists then the first attribute is assumed.

<option>	Description
CFW	Cell flow width (see Section 6.11.2). The value entered for CFW is a factor to adjust the 2D cell flow widths (in the same manner as 2D flow constrictions (FC)), noting that the changed flow width applies to all depths. For example 0.1 will limit the flow width to 10%.
CnM	Bed resistance value. CnM is a Chezy C, Manning's n or Manning's M value as set by Bed Resistance Values.
Code	Cell code (see Section 6.7). When using "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 7-4 in Section 7.4.1. 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.
FLC	Applies the form loss attribute values to all cells within each region. Note that FLC values will need to be changed if the 2D cell size changes.
FLC/L	Applies a form loss per unit length to all cells within each region. The effect of applying the FLC in this manner is 2D cell size independent.
FRIC	Ripple height.
GWD	Groundwater depth (see Section 6.10.5). The Read GIS GWD command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
GWL	Ground water level (see Section 6.10.5). The Read GIS GWL command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
IWL	Initial water level (see Section 7.6.1.2). 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.
MAT	Material ID (see Section 6.9). The Material ID value must correspond to a value within the Materials Definition File (refer to Read Materials File). If Bed Resistance Cell Sides == INTERROGATE (the default), the material values are 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 6.9.1.
Soil	Soil ID (see Section 6.10). The Soil ID value must correspond to a value within the .tsoilf file. (refer to Read Soils File).
SRF	Storage reduction factor (see Section 6.11.1).
WrF	Weir factor. The 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). See Section 6.3.1. The WrF values are multiplied by the Global Weir Factor specified in the .tcf file.

Read GIS FC Shape [{} | Write TIN] == <gis_layer>

(Optional)

Command to read in a flow constriction shape (2d_fcsh) GIS layer which constricts the flow across a 2D cell side in a number of ways. See Section 6.12.2.

Read GIS Layered FC Shape [{} | Write TIN] == <gis_layer>

(Optional)

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 6.12.2.2 for details.

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.

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

(Optional. Alias: Read GIS Receptors)

(In Builds prior to 2016-03-AA command was named Read GIS Gauge Output.)

References a 2d_obj (GIS objects) or 2d_rec (receptors) GIS layer containing points or polygons representing receptors such as properties or buildings. At present the only option is to use the RECORD GAUGE DATA feature, with more options to record or value add information to receptors planned for future releases.

RECORD GAUGE DATA records the flood level and simulation time at one or more gauge(s) when the receptor is first inundated above a trigger inundation level (eg. floor level). Gauges are defined as a point within a 2d_po GIS layer with type "G_" (see Section 9.3.3 and Table 9-3). The levels from all gauges are recorded at each receptor once inundated.

At present up to 20 different layers may be read in by repeating this command (let us know if you want to input more than 20 layers!). As of Build 2016-03-AA both .mif and .shp formats are supported (previously only .mif format was supported).

The first attribute in the GIS layer is used to set the trigger inundation elevation (eg. floor level) to record the gauge level(s) unless the USE ZC option is specified.

The USE ZC option, sets the trigger inundation level to either the ZC elevation of the cell (for digitised point objects) or the lowest ZC elevation within the polygon (for digitised polygon objects). In this case the first attribute is not used as the trigger inundation level.

The ZPTS option allows TUFLOW to adjust the Zpts within the polygon or whole cell at a point object, essentially merging the functionality of the Read GIS Zpts command with Read GIS Objects. In this case, the first attribute is used to set the value of the Zpts within each polygon. If there is a second attribute, this is used to define the trigger inundation level, otherwise the first attribute is used for this level.

Refer to Section 9.5.1 for further information.

Read GIS Variable Z Shape [XF ON | XF OFF | Write TIN] == <gis_layer>

(Optional)

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 6.8.6.

The initial water level is also adjusted with the adjustment of the cell elevations whilst the cell is dry (otherwise water is generated if any dry cells start being lowered).

The XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <vzsh_file>. The global default for using XF files is ON and can be changed using XF Files.

Read GIS Z HX Line [{} | RIDGE or MAX or RAISE] == <gis_layer>

(Optional)

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 6.8.7.1). 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 7-5.

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 7-5. An "A" flag adjusts all elevations irrespective of whether RIDGE, MAX or RAISE are used.

```
Read GIS Z Line
[ {} | RIDGE or MAX | GULLY or MIN ]
[ {} | THICK ]
[ {} | ADD ]
== <gis_layer>
(Optional)
See also Allow Dangling Z Lines and Pause When Polyline Does Not Find Zpt commands.)
```

Reads .mif/.mid or .shp 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 layer (see Table 12-2) 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.**

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.

If neither the RIDGE nor 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.

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. 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 Section 6.8.3 for further information.

Read GIS Z Shape
[{} | RIDGE or MAX or RAISE | GULLY or MIN or LOWER]
[XF ON | XF OFF | Write TIN]
== <gis_layer>

(Optional)

Command 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 6.8.5. 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_Options attribute for a line overrides this setting. Similarly if GULLY or MIN or LOWER is specified.

The 2d_zsh_obj_check layer may be used to view the buffer polygons of wide Z Lines. Refer to Table 12-2.

XF ON and XF OFF switch the automatic generation of XF files ON or OFF specifically for <zsh_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 | Write TIN]
== <gis_layer>

(Optional)

Command 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 9.5.1 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] == <gis_layer>

(Optional)

Reads the Zpt values from a GIS layer exported in .mif/.mid or .shp file format. 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 GRID <option> == <grid_file>

(Optional)

Reads the <option> values from an ESRI ASCII (.asc) or a binary (.flt) grid. The format is controlled by the file extension (use .flt for binary grids, otherwise an ASCII grid is assumed regardless of the file's extension).

For real (float) inputs the value is interpolated from the grid. For integer inputs (such as Code), the value applied is the integer value of the ASCII grid cell that the TUFLOW point falls within.

The following options are available:

<option>	Description
CnM	Bed resistance value. CnM is a Chezy C, Manning's n or Manning's M value as set by Bed Resistance Values.
CFW	Cell flow width (see Section 6.11.2). The value entered for CFW is a factor to adjust the 2D cell flow widths (in the same manner as 2D flow constrictions (FC)), noting that the changed flow width applies to all depths. For example 0.1 will limit the flow width to 10%.
Code	Cell code (see Section 6.7).
FLC	Applies form losses. Note that FLC values will need to be changed if the 2D cell size changes.
FLC/L	Applies form losses per unit length to all cells. The effect of applying the FLC in this manner is 2D cell size independent.

<option>	Description
GWD	Groundwater depth (see Section 6.10.5). The Read Grid GWD command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
GWL	Ground water level (see Section 6.10.5). The Read Grid GWL command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
IWL	Initial water level (see Section 7.6.1.2). 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.
MAT	Material ID (see Section 6.9). The Material ID value must correspond to a value within the Materials Definition File (refer to Read Materials File). If Bed Resistance Cell Sides == INTERROGATE (the default), the material values are 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 6.9.1.
Soil	Soil ID (see Section 6.10). The Soil ID value must correspond to a value within the .tsoilf file. (refer to Read Soils File).
SRF	Storage reduction factor (see Section 6.11.1).
WrF	Weir factor. The 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.

Examples of use are:

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

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

Refer also to the commands Read Grid Zpts and Read Grid Location.

Read GRID Location == <grid_file>

(Mandatory if Read GIS Location not used)

Sets the size and location of a 2D domain based on a DEM in ESRI ASCII (.asc) or binary (.flt) grid formats. The dimensions of the grid is used to set the 2D domain's origin and X,Y dimensions (ie. replaces Origin and Grid Size (X,Y) or Read GIS Location. The orientation angle is set to zero (ie. the 2D domain will be orientated north-south). Useful where the model extent is the same as the DEM. Cell Size still needs to be specified and can be different to the DEM's cell size.

Read Grid Zpts [{} | ADD | MAX | MIN] [{XF ON} | XF OFF] == <grid_file> | <gis_layer>

(Optional)

Directly interrogates an ESRI ASCII (.asc) or binary (.flt) grid to set the Zpt elevations.

The use of this command has significant advantages over the previous method of manually carrying out a point inspection on an empty 2d_zpt layer. It will allow for the model to (very likely) become cell size independent. Changing a 2D domain's orientation and dimensions is also much simpler without the need to regenerate and point inspect a 2d_zpt layer. See Section 6.8.1 for more details.

The ADD option adds the value interrogated from the grid 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.

The XF ON and XF OFF options can be used to switch the writing of XF files (see Section 4.10) on or off for individual inputs.

This command permits a second argument, specifying a GIS layer containing one or more polygons to clip the area of Zpts to be inspected. Elevations will only be assigned to Zpts lying inside polygons within the GIS layer. See Section 6.8.1 for more details.

Like other .tgc commands, Read Grid Zpts may be specified more than once. You can also specify ADD, MIN or MAX in the same way as for other similar commands.

See also the .tgc command Grid Approach.

Read RowCol <option> == <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 <option> as listed in the table below. Any columns after the third are ignored.

Also refer to the commands Read RowCol Zpts and Read RowCol RF.

<option>	Description
CnM	Bed resistance value. CnM is a Chezy C, Manning's n or Manning's M value as set by Bed Resistance Values.
Code	Cell code (see Section 6.7).
FLC	Form loss coefficient.

<option>	Description
Fric	Ripple height.
IWL	Initial water level (see Section 7.6.1.2). 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.
MAT	Material ID (see Section 6.9). The Material ID value must correspond to a value within the Materials Definition File (refer to Read Materials File). Note the Read RowCol Mat command is incompatible with the Bed Resistance Cell Sides == INTERROGATE option. If using Read RowCol Mat, use AVERAGE M or AVERAGE n.
WrF	Weir factor. The 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.

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 GIS 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] [{XF ON} | XF OFF] == <tin_file> | <gis_layer>

(Optional)

Use to read a triangulation file (TIN) to assign elevation values to Zpts. The following is a list of accepted TINs. The type of TIN is determined by the file extension:

- SMS .tin files
- SMS .2dm files. This may be useful for quickly setting up a TUFLOW model based on a flexible mesh model that uses a .2dm file.

- 12D TINs as a .12da file.
- .xml TINs that are readily exported from 3D surface TIN software such as Autodesk's Civil3D.

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.

This command permits a second argument, specifying a GIS layer containing one or more polygons to clip the area of Zpts to be inspected. Elevations will only be assigned to Zpts lying inside polygons within the GIS layer. See Section 6.8.1 for more details.

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

The XF file can be switched on or off from the global default settings using XF Files or specifically for this layer using "Read TIN Zpts XF OFF == …".

Set <option> == <value>
Set Code [{} | ZERO ABOVE ZC] == <ZC>

(Optional)

Sets the value of <option> over the entire grid. Used for initialising grid values. The options available are listed in the table following.

Refer also to the command Set Zpt.

<option>	Description
CnM	Bed resistance value. CnM is a Chezy C, Manning's n or Manning's M value as set by Bed Resistance Values.
CFW	Cell flow width (see Section 6.11.2). The value entered for CFW is a factor to adjust the 2D cell flow widths (in the same manner as 2D flow constrictions (FC)), noting that the changed flow width applies to all depths. For example 0.1 will limit the flow width to 10%.
Code	Cell code (see Section 6.7). 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,

<option>	Description
	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.
FLC	Applies a form loss to all cells. Note that FLC values will need to be changed if the 2D cell size changes.
FLC/L	Applies a form loss per unit length to all cells. The effect of applying the FLC in this manner is 2D cell size independent.
FRIC	Ripple height.
GWD	Groundwater depth (see Section 6.10.5). The default is for the GWD/GWL to be infinitely deep. If a cell has both a GWD and GWL specified, the higher of the two (elevation wise) prevails. This can be checked by viewing the Map Output Data Type dGW, which shows the depth to groundwater (from the ground surface) in metres or feet. The Set GWD command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
GWL	Ground water level (see Section 6.10.5). The default is for the GWD/GWL to be infinitely deep. If a cell has both a GWD and GWL specified, the higher of the two (elevation wise) prevails. This can be checked by viewing the Map Output Data Type dGW, which shows the depth to groundwater (from the ground surface) in metres or feet. The Set GWL command may only be specified following Set Soil, Read GIS Soil and/or Read GRID Soil commands.
IWL	Initial water level (see Section 7.6.1.2). Note: IWLs can also be set in the .tcf file (see Set 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.
MAT	Material ID (see Section 6.9). The Material ID value must correspond to a value within the Materials Definition File (refer to Read Materials File).
Soil	Soil ID (see Section 6.10). The Soil ID value must correspond to a value within the .tsoilf file. (refer to Read Soils File).
SRF	Storage reduction factor (see Section 6.11.1).
WrF	Weir factor. The 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.

Set Code and Clean Zpt == <Z_inactive>

(Optional)

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

1. The DEM used to assign elevations has been trimmed to the catchment boundary or model extent (ie. all null areas of the DEM will be assigned an inactive code as these Zpts have not been assigned an elevation); or
2. Reading a .2dm file from another model to assign elevations (refer to the command Read TIN Zpt).

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

Set Route Cut Off Type == [{Depth} | Velocity or V | Hazard or VxD or Z0]

(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. Depth, velocity and hazard options are available. 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 9.5.1 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.

Set Route Cut Off Values == <y1, y2, ...>

(Optional)

Sets the cutoff values for the evacuation route feature (see Section 9.5.1) 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.

Set Variable <name> == <value>

(Optional)

In any TUFLOW control file use this command to define a variable's name and value. Wherever you want to refer to the variable, the variable's name must be bounded by "<<" and ">>" characters.

Refer to the Set Variable description within the tcf commands for an example.

Set Zpt == <elevation_in_metres>

(Optional)

Sets all ZC, ZU, ZV and ZH Zpts to the value specified.

Stop

(Optional)

Stops TUFLOW (useful while just developing the model grid and Zpts).

Thin Line as Thick == [ON | OFF]

(Optional)

If set to ON, treats all thin Z lines as thick. The default is OFF, unless GPU Solver == ON, in which case the default is ON.

TIN Angles == <point_angle>, <edge_angle> | {55, 30}

(Optional)

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.

<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.

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)

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.

Write GIS Domain == <gis_layer>

(Optional)

Creates 2d_dom .mif/.mid or .shp file 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 GIS Grid.

A 2d_dom layer is also created using Write Check Files, however, it will contain a rectangular region for all 2D domains.

Write GIS Grid == <gis_layer>

(Optional)

Creates a .mif/.mid or .shp file 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 or .shp 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 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.

Write GIS Zpts == <gis_layer>

(Optional)

Writes a .mif/.mid or .shp file containing the points where Zpts (model elevation) values are defined.

Tip: Use this command to check that the model's elevation data is correct. After building the topography use this command to write a temporary .mif/.mid or .shp file. Import into the GIS and check the elevations are as expected.

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)

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.

Appendix D .tbc File Commands

BC Database
BC Event Name
BC Event Text
Blank BC Type
Blank HQ Slope

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 GIS Streams
Read RowCol RF

Set Variable

Unused HX and SX Connections

BC Database == <.csv_file>

(Mandatory)

Sets the active BC Database file as described in Section 7.5. 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 7.5.2 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 7.5.2 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.

Blank BC Type == [<bc_type> | {NONE}]

(Optional)

If a blank BC type occurs the value entered is used. If NONE (the default) is specified, a BC type must be assigned to every object in 2d_bc layers.

This command can be repeated within the .tbc file as per the example lines below.

```
BLANK BC TYPE == SX

Read GIS BC == mi\2d_bc_M02_culverts_TD15006.MIF

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

Blank HQ Slope == <slope>

(Optional)

A default HQ slope can be specified directly in the .tbc, and can be repeated prior to reading different HQ boundaries (in separate layers). Note, <slope> will only be used if no boundary name is specified for the HQ boundary (as a specified for the HQ boundary slope is given preference over the name).

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^3/s and applied as a source versus time (ST).

This command applies rainfall to all active cells. Therefore if the rainfall being applied is the same for all cells, this command negates the need to use a 2d_rf layer.

If used in conjunction with the commands Global Rainfall Initial Loss and Global Rainfall Continuing Loss to apply initial and continuing losses, The Global Rainfall BC command must occur after the two preceding commands as shown in the example below else no losses will be applied:

```
Global Rainfall Initial Loss == 10
Global Rainfall Continuing Loss == 2
Global Rainfall BC == rainfall
```

See also the command Global Rainfall Area Factor.

Global Rainfall Continuing Loss == [{0} | <CL_in_mm/h>]

(Optional)

Sets the continuing loss rate in mm/h for any global rainfall. This command must occur prior to the command Global Rainfall BC else no losses will be applied.

Global Rainfall Initial Loss == [{0} | <IL_in_mm>]

(Optional)

Sets the initial loss in mm for any global rainfall. This command must occur prior to the command Global Rainfall BC else no losses will be applied.

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 7.4.1.

Read GIS RF == <gis_layer>

(Optional)

Reads the polygons for applying rainfall directly to 2D cells as described in Table 7-4 and uses 2d_rf layers as described in Table 7-7.

Also refer to the .tbc command Read RowCol RF and the .tcf command Read Grid RF.

Read GIS SA [{} | ALL | PITS | STREAM ONLY | STREAM IGNORE] [{} | RF | PO] == <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 7-4. Usually used for specifying rainfall runoff directly onto the 2D domain(s).

The RF (rainfall) option is available to specify rainfall hyetographs (mm versus hours) instead of flow hydrographs. See Table 7-4 for more information.

Negative rainfall values when using the Read GIS SA RF or Read GIS RF are treated as a loss (eg. evaporation). No IL/CL values that apply to positive rainfall are applied to negative values. Previously negative values were treated as zero.

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_1D_2D (previously Topo_ID) 1d_nwk attribute. The inflow is spread equally over the applicable 2D cells. An ERROR occurs if no 2D cells are found within the region.

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 0 for more details.

The PO option models seepage or infiltration based on a varying water level or flow rate elsewhere in the model. This feature was recently used to model the seepage of groundwater into a coastal lagoon that was dependent on the water level in the lagoon as observed from long-term historical measurements. See Section 7.4.2.5 for more details.

Two options are available for controlling streamlines (refer to Read GIS Streams):

- The STREAM ONLY option will only apply the SA inflows to the streamline cells, ie. no non-streamline wet cells in the SA region will receive an inflow. Note this is the default approach adopted by the GPU solver.
- The STREAM IGNORE option will ignore all streamline cells within the SA region(s) and distribute the inflows using the standard approach for SA inflows (ie. lowest cell if all wet, otherwise distributed over the wet cells).

Read GIS Streams == <gis_layer>

(Optional)

Streamlines allow the user to apply SA inflows along the waterways rather than to the lowest cell (when all cells are dry within the SA region).

The Read GIS Streams command can be used one or more times in the .tbc file to define streamline cells. Streamlines are typically polyline or line objects, usually representing the path of the waterways. One attribute is required being the Stream Order as an integer. GIS and other software have the ability to generate streamlines from DEMs, and usually assign a stream order to each stream polyline. If needed, rearrange (or copy) the attributes so that the first attribute is the stream order one.

Point and region objects are also recognised, for a point object the cell in which the point falls is designated as a stream cell. For a region objects all cells with the cell centre (ZC) within the polygon are assigned as stream cells. The stream cells within SA regions are output in the SAC Check file.

Note only streams with a stream order greater than zero (0) are used by TUFLOW. Therefore, streams that are not to be used for applying SA inflows can be assigned a stream order of 0 or deleted from the layer.

If SA Proportion to Depth == ON (default setting), the distribution of SA inflows according to depth only applies to wet non-streamline cells. The approach adopted is as follows:

- The total inflow assigned to streamline cells within a SA region is proportioned according to the number of streamline cells versus wet non-streamline cells.
- The distribution of the inflow allocated to streamline cells is weighted equally between cells.

- The distribution of the inflow allocated to wet non-streamline cells is weighted according to the depth of water in the cells.

By default, any wet cells that are not streamline cells are also included in the distribution of the SA inflow. See Read GIS SA STREAM ONLY and Read GIS SA STREAM IGNORE options for controlling streamline inflows.

Read RowCol RF == <mid_file>

(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 7-7. To create this layer, select all ZC points from a 2d_zpt layer, save the selection as 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 7-7. Update the Name attribute to one or more rainfall boundaries. Different proportions of different rainfall hyetographs can be applied by duplicating the layer and having one layer for each rainfall boundary.

Also refer to the .tbc command Read GIS RF and the .tcf command Read Grid RF.

Set Variable <name> == <value>

(Optional)

In any TUFLOW control file using this command defines a variable's name and value. Wherever you want to refer to the variable, the variable's name must be bounded by "<<" and ">>" characters.

Refer to Set Variable description within tcf commands.

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 .toc File Commands

Cd
Cd Gate
Cd Gate Submerged
Cd Spillway

Define Control

End Define

Gate Height Fully Open
Gate Opening
Gate Seat Vertical Offset
Gate Speed
Gate Type
Gate Width Fully Open

Method

Operation

Period Opening/Closing
Period Startup/Shutdown
Pump Capacity
Pump Number
Pump Operation

Weir Height
Weir Height Speed
Weir Width
Weir Width Speed

Cd == [{0.75} | <Cd>]

(Applies to: RO Channels)

For RO culverts, sets the discharge coefficient used in the Nair 2003 equation.

Cd Gate == [{0.6 or 0.75} | <Cd>]

(Applies to: SGO, SPO Channels)

Sets the discharge coefficient of the gate. For sluice gates the default value is 0.6, while for gated spillways the default is set to the default for Cd Spillway which is 0.75.

Cd Gate Submerged == [{0.8} | <Cd>]

(Applies to: SGO Channels)

Sets the discharge coefficient of the gate when fully submerged. The default value is 0.8.

Cd Spillway == [{0.75} | <Cd>]

(Applies to: SPO Channels)

Sets the discharge coefficient of the spillway. Prior to the 2016-03 release the default was 0.5 due to using a different version of the weir equation (see Section 5.9.2.5); but the same results would apply.

Define [Culvert | Pump | Q Channel | Sluice | Spillway | Weir] Control == <control_id>

(Mandatory)

Each Define Control block consists of three sections:

- The default settings for the control's commands, which are usually placed at the top of the definition and prior to the logical rules. The default settings are the values used for a command in the event the command is not used within the logic rules.
- User defined variables – see Section 5.9.1.2
- One or more logical rules – see Section 5.9.1.3.

Within the control definition, commands specific to the type of structure/device can be used to adjust the structure/device's operation. Further information can be found in Section 5.9.1.1.

<control_id> is a unique control definition name. For a 1d_nwk channel to use the control, <control_id> must be entered into the 1d_nwk Inlet_Type attribute. As mentioned above, more than one channel can reference the same control. For example, several pumps may utilise the same operational control logic.

Use End Define to terminate the block. An ERROR occurs if End Define is not specified.

End Define

(Mandatory)

Ends a Define Control block of .toc commands for the operating rules applied to hydraulic structures, pumps and other controllable devices.

Gate Height Fully Open == <height_in_metres>

(Applies to: RO, SGO, SPO Channels)

For vertically moving gates the height (not elevation) of the gate when fully open above the gate's seat. If not set, the 1d_nwk "Height" attribute is used.

Gate Opening [{} | (%)] == [[++ | -- | {}] <opening> | {CLOSE} | OPEN | NO CHANGE]

(Applies to: Q, RO, SGO, SPO Channels)

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

Gate Seat Vertical Offset == <offset>

(Applies to: SPO Channels)

The difference in height in metres (or feet if Units == US Customary) between the spillway crest and the seat of the gate.

Gate Speed [{} | (min)] == <speed>

(Applies to: QO, RO, SGO, SPO Channels)

The speed at which the gate moves. Units are m/s or ft/s or if "(min)" is specified in m/min or ft/min.

Gate Type == [VERTICAL UNDERFLOW | VERTICAL OVERFLOW | HORIZONTAL SINGLE | HORIZONTAL DOUBLE]

(Applies to: RO Channels)

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

Gate Width Fully Open == <width>

(Applies to: RO Channels)

For horizontally moving gates the width of the gate(s) when fully open. Units are m or ft. If not set, the 1d_nwk "Width_or_Dia" attribute is used.

Method == <method>

(Reserved for future use)

Sets which method to use for the hydraulic calculations. This command as of the 2013-12 release does not need to be applied, as it is intended for use should alternative equations become available for a structure in future releases.

Period Opening/Closing [{} | (min) | (s)] == [{0.167} | <period>]
Period Startup/Shutdown [{} | (min) | (s)] == [{0.167} | <period>]

(Applies to: All Operational Structures)

These two commands are identical. The time taken to fully open a closed gate or to fully close an open gate or the time taken to start the pump up or shut it down. Units are in hours by default, but minutes or seconds can be used if "(min)" or "(s)" is specified. For example, specifying either of the commands below will set the time taken to open/close the gate to half an hour:

```
Period Opening/Closing == 0.5
Period Opening/Closing (min) == 30
Period Opening/Closing (s) == 1800
```

If this command is omitted the default opening/closing period is 0.167hrs (or 60s).

This command is identical to Period Startup/Shutdown and the two can be used interchangeably.

Operation == [NO CHANGE]

(Applies to: All Operational Structures)

Keep the operation of the structure unchanged. UNCHANGED can also be used instead of NO CHANGE.

Pump Capacity == [<flow> | <discharge_curve>]

(Applies to: Pumps)

The flow capacity of the pump either as a constant flow or reference to a dep. If a constant flow rate, specify <flow>. If a head-discharge curve specify the name of the curve in the Depth Discharge Database.

Pump Number == <no_pumps>

(Applies to: PO Channels)

Number of pumps in parallel.

Pump Operation == [ON | OFF | NO CHANGE]

(Applies to: PO Channels)

Turns the operation of the pump on or off, or keeps the current operation unchanged.

Weir Height [{} | %] == <weir_height>

(Applies to: Operational Weirs)

The height (not elevation) of the weir above its fully down (open) state to operate towards. The % option allows the specification of the percentage of the weir height that is up (0% would indicate completely lowered and 100% completely raised).

Note that the height of the weir above the crest when fully up is set by the 1d_nwk Height_or_WF attribute.

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

(Applies to: Operational Weirs)

The speed at which the weir moves in the vertical. Units are m/s or ft/s or if "(min)" is specified in m/min or ft/min.

Weir Width [{} | %] == <weir_width>

(Applies to: Operational Weirs)

The width (not elevation) of the weir to operate towards. The % option allows the specification of the percentage of the weir width that is open (0% would indicate completely closed and 100% completely open).

Note that the full width of the weir is set by the 1d_nwk Width_or_Dia attribute.

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

(Applies to: Operational Weirs)

The speed at which the weir moves in the horizontal. Units are m/s or ft/s or if "(min)" is specified in m/min or ft/min.

Appendix F .trfc File Commands

IDW Exponent
IDW Maximum Distance
IDW Maximum Points

Maximum Hyetograph Points
Maximum RF Locations

Read GIS RF Points
Read GIS RF Polygons
Read GIS RF Triangles
RF Grid Cell Size
RF Grid Format
RF Grid Origin
RF Grid Size
RF Interpolation Method

IDW Exponent == [<p> | {2}]

(Optional)

If using the RF Interpolation Method == IDW the exponent in the IDW equation can be changed from the default value of 2.

IDW Maximum Distance == <max_dist>

(Optional)

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

IDW Maximum Points == <max_points>

(Optional)

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

Maximum Hyetograph Points == [<max_pts> | {1,000}]

(Optional)

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

Maximum RF Locations == [<max_rf_gauges> | {1,000}]

(Optional)

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

Read GIS RF Points == <gis_layer>

(Mandatory)

Read the point rainfall locations in the 2d_rf file format. For each point the attributes are Name, f1 and f2 factors. If the rainfall factors f1 and/or f2 are zero (or less than zero), these are changed to 1 and WARNING 2618 is issued.

Read GIS RF Polygons == <gis_layer>

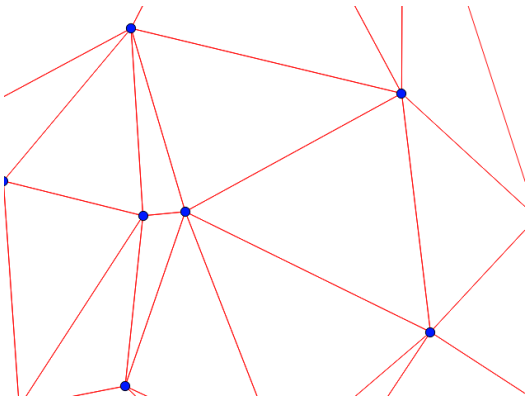
(Mandatory if using RF Interpolation Method == POLYGON)

The GIS layer contains a series of polygons or regions in the 2d_rf format. These polygons can either have the rainfall boundary Name (and f1 and f2 factors) specified on the polygon objects, or if these attributes are blank TUFLOW looks for rainfall points (specified with the Read GIS RF Points) that fall within the polygons. If the Name attribute in the polygon layer is blank and no points fall within the polygon an ERROR 2619 occurs.

Read GIS RF Triangles == <gis_layer>

(Mandatory if using RF Interpolation Method == TIN)

Reads in a GIS layer containing the triangulation of the rainfall points. The GIS objects should be polygons or regions with three vertices, with each vertex snapped to a rainfall point location as specified by Read GIS RF Points. The layer is typically produced by other software specialising in the interpolation of rainfall, but can be manually created when only a small number of rainfall locations exist. The attributes of the GIS layer are not used. For each grid cell in the rainfall output grids, the rainfall depth is based on the planar (linear) interpolation of the three rainfall depths at the vertices of the bounding triangle. An example of a triangulation polygon layer (red) connecting the rainfall point locations (blue) is shown below.



RF Grid Cell Size == <value>

(Optional)

Sets the cell size for the generated rainfall grids. If omitted, a value of 10 times the 2D domain cell size is used. Typically the rainfall can be satisfactorily represented on a much coarser resolution than that required for the hydraulic calculations, therefore, using high resolution rainfall grids is not required and unnecessarily consumes memory and disk space, and may slow down the simulation.

RF Grid Format == [ASC | FLT | NC]

(Mandatory)

This mandatory command sets the output grid format. Options are ASC (ESRI asc grid – extension .asc), FLT (ESRI binary grid – extension .flt) or NC (NetCDF extension .nc).

The rainfall grids are output to a separate folder \RFG\ <rainfall_grid> in the location of the .trfc file. If the .trfc file is in the \bc_dbase\ folder, a new folder \bc_dbase\RFG\ is created containing the output grids.

The output formats from the rainfall interpolation are compatible with the formats used by the .tcf Read Grid RF, therefore, once the rainfall grids have been generated this command can be used to apply the rainfall, rather than regenerate the rainfall grids using the .trfc file (should the .trfc input files remain unchanged).

NC: If NetCDF output is specified a single output file (.nc) containing all timesteps in a single file is created. This is given the simulation name: TUFLOW\bc_dbase\RFG\<simulation_name>.nc ensuring that the dataset is not accidentally overwritten as the simulation is running. There is no limit to the number of rainfall timesteps that are included in the NetCDF format.

A total rainfall depth is also output, however, this is not used by TUFLOW during the simulation, and it can be used for checking purposes. For more information see TUFLOW NetCDF Rainfall Format Wiki Page.

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

An index file containing a list of the times and rainfall grid filenames is written in .csv file format in the same folder, for example:
\bc_dbase\RFG\<simulation_name>\<simulation_name>_rf_index.csv

A limit of 1,000 grids exists if using the ASC or FLT format rainfall grids, if more than 1,000 grids are required the NetCDF format should be used.

RF Grid Origin == <OX>, <OY>

(Optional)

Sets the origin for the output rainfall grid. If this command is omitted the rainfall grid origin is based on the origin of the TUFLOW 2D domain(s).

RF Grid Size (N,M) == <rows>, <columns>
RF Grid Size (X,Y) == <X_length>, <Y_length>

(Optional)

Sets the size of the output rainfall grids. Similar to the .tgc Grid Size (N,M) and Grid Size (X,Y) commands. If omitted the rainfall grid size is based on the dimensions in the TUFLOW 2D domain(s).

RF Interpolation Method == [IDW | POLYGON | TIN]

(Mandatory)

Sets the interpolation approach between rainfall locations. This command must be specified with one of the options as described below.

IDW: An inverse distance weighting approach is used to calculate the rainfall depth based on the distance to the surrounding rainfall points (specified by Read GIS RF Points).

$$\hat{v}_1 = \frac{\sum_{i=1}^{n} \frac{1}{d^p{}_i} v_i}{\sum_{i=1}^{n} \frac{1}{d^p{}_i}}$$

The exponent (p) can be changed from its default value of 2 using IDW Exponent. Other commands that affect the IDW interpolation are IDW Maximum Distance and IDW Maximum Points.

POLYGON: A series of GIS polygons are specified and the rainfall for each polygon comes from the point rainfall. This can be used to apply distributions such as Thiessen polygons generated from other software. The regions or polygons are read in the 2d_rf format, and can either have the rainfall boundary Name (and F1 and F2 factors) specified via the polygon attributes, or if the attributes are blank TUFLOW will look for rainfall points (specified by Read GIS RF Points) that fall within the polygons. If the Name attribute in the polygon layer is blank and no points fall within the polygon an ERROR 2619 occurs.

This method is similar to using a series of rainfall polygons read in via the .tbc Read GIS RF command. By pre-processing using a .trfc file, however significant more memory efficiency occurs, particularly if a large number of rainfall boundaries is used.

TIN: A TIN (Triangulated Irregular Network) is specified via Read GIS RF Triangles which connects the rainfall point locations.

Appendix G Hyperlinks to All Commands

.tcf Commands

Adjust Head at ESTRY Interface
Apply Wave Radiation Stresses
Apply Wind Stresses

BC Control File
BC Database
BC Event Name
BC Event Source
BC Event Text
BC Wet/Dry Method
BC Zero Flow
Bed Resistance Cell Sides
Bed Resistance Depth Interpolation
Bed Resistance Values
Boundary Viscosity Factor
BSS Cutoff Depth

Calibration Points MI File
Cell Wet/Dry Depth
Cell Size
Change Zero Material Values to One
Check Inside Grid
Check MI Save Date
Check MI Save Ext
Command Line Processing
CSV Header Line
CSV Maximum Number Columns
CSV Time

Defaults
Define Event
Define Output Zone
Demo Model
Density of Air
Density of Water

Depth/Ripple Height Factor Limit
Display Water Level
Distribute HX Flows

End 1D Domain
End 2D Domain
End After Maximum
End Define
End Map Output
End Time
ESTRY Control File
Event File
Excel Start Date
Extrapolate Heads at Flow Boundaries

First Sweep Direction
Force File IO Display
Free Overfall
Free Overfall Factor
Froude Check
Froude Depth Adjustment

GA Convergence Value
GA Maximum Iterations
Geometry Control File
GIS Format
GIS Grid Vector Type
GIS Grid Vector Direction
GIS Grid Vector SF
GIS Grid Vector TTF
GIS Projection Check
GIS Supported Object Ignored
GIS Unsupported Object
Global FC Ch Factor
Global Weir Factor
GPU Device IDs
GPU DP Check
GPU RAM Optimisation

GPU Solver
GPU Temporal Scheme
Grid Format
Grid Output Cell Size
Grid Output Origin

HQ Boundary Approach
HX Force Weir Equation
HX ZC Check

If Event
If Scenario
Input Drive
Inside Region
Instability Water Level

Latitude
Layered FLC Default Approach
Line Cell Selection
Link 2D2D Adjust Velocity Head Factor
Link 2D2D Approach
Link 2D2D Distribute Flow
Link 2D2D Global Stability Factor
Log Folder

Map Cutoff Depth
Map Output Corner Interpolation
Map Output Data Types
Map Output Entire Model
Map Output Format
Map Output Interval
Mass Balance Output
Mass Balance Output Interval
Maximum 1D Channels
Maximum Courant Number
Maximum Points

Maximum Velocity Cutoff Depth
Maximum Vertices
Maximums and Minimums
Maximums and Minimums Only for Grids
Maximums and Minimums Time Series
Maximums Approach
Meshparts
MI Projection
Model Events
Model Output Zones
Model Platform
Model Precision
Model Scenarios
Model TUFLOW Build
Model TUFLOW Release

Negative Depth Approach
NetCDF Output Compression
NetCDF Output Start Date
NetCDF Output Time Unit
NetCDF Output Direction
NetCDF Output Format
Null Cell Checks
Number Iterations
Number 2D2D Link Iterations

Output Approach
Output Drive
Output Files
Output Folder

PO Approach
Pause

Rainfall Boundaries
Rainfall Gauges
Recalculate Chezy Interval
Read File
Read GIS Cyclone/Hurricane
Read GIS FC
Read GIS GLO
Read GIS IWL
Read GIS LP
Read GIS Output Zone
Read GIS PO
Read GIS Reporting Location
Read GIS X1D Network
Read GIS X1D Nodes
Read GIS X1D WLL
Read GIS X1D WLL Points
Read Grid IWL
Read Grid RF
Read Materials File
Read Restart File
Read RowCol IWL
Read Soils File
Reveal 1D Nodes

SA Minimum Depth
SA Proportion to Depth
Screen/Log Display Interval
Set IWL
Set Route Cut Off Type
Set Route Cut Off Values
Set Variable
Snap Tolerance
SHP Projection
Simulations Log Folder
Start 1D Domain
Start 2D Domain
Start Map Output
Start Time
Start Time Series Output
Supercritical
SX Flow Distribution Cutoff Depth
SX Head Adjustment
SX ZC Check

Time Series Output Format
Time Series Output Interval
Time Output Cutoff
Timestep
Timestep Maximum Increase
TSF Update Interval
Tutorial Model

UK Hazard Debris Factor
UK Hazard Formula
UK Hazard Land Use
Units
Unused HX and SX Connections
Use Forward Slash

Verbose
Viscosity Coefficient
Viscosity Formulation
VG Z Adjustment

Water Level Checks
Wetting and Drying
Wind/Wave Shallow Depths
Write Check Files
Write Empty GIS Files
Write PO Online
Write Restart File at Time
Write Restart File Interval
Write Restart File Version
Write Restart Filename

Write X1D Check Files

XF Files
XF Files Include in Filename
XMDF Output Compression

Zero Negative Depths
Zero Rainfall Check
ZP Hazard Cutoff Depth
Zpt Range Check

.tgc Commands

Allow Dangling Z Lines

Cell Size
Create TIN Zpts

Default Land Z

Grid Approach
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 Objects
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 GIS Zpts Gauge Output
Read Grid
Read Grid Location
Read Grid Zpts
Read RowCol
Read RowCol Zpts
Read TIN Zpts

Set
Set Code and Clean Zpt
Set Route Cut Off Type
Set Route Cut Off Values
Set Variable
Set Zpt
Stop

Thin Line as Thick
TIN Angles
TIN Coincident Point Distance

Write GIS Domain
Write GIS Grid
Write GIS Zpts

ZC == MIN(ZU,ZV)
Zero Z Point

.tbc Commands

BC Database
BC Event Name
BC Event Text
Blank BC Type
Blank HQ Slope

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 GIS Streams
Read RowCol RF

Set Variable

Unused HX and SX Connections

1D (.ecf) Commands

Note: These commands can be placed in the .ecf file, in the .tcf file within a Start 1D Domain block or in the .tcf file by preceding the command with "1D", for example: 1D Read GIS Network.

BC Database
BC Event Name
BC Event Text
Bridge Flow
Bridge Zero Coefficients

Create Nodes
CSV Format
CSV Maximum Number Columns
CSV Time
Conveyance Calculation
Culvert Critical H/D
Culvert Flow
Culvert Zero Coefficients

Depth Discharge Database
Depth Limit Factor

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

M Channel Approach
M11 Network
Manhole Approach
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

Order Output
Output Data Types
Output Folder
Output Interval
Output Times Same as 2D

Pit Channel Offset
Pit Default 2D Connection
Pit Default Road Crossfall
Pit Inlet Database
Pit Search Distance

Read File
Read GIS BC
Read GIS IWL
Read GIS Manhole
Read GIS Network
Read GIS Pits
Read GIS Table Links
Read GIS WLL
Read GIS WLL Points
Read Operating Control File

S Channel Approach
Set IWL
Set Variable
Start Output
Storage Above Structure Obvert
Structure Flow Levels
Structure Losses
Structure Losses Approach
Structure Routines

Taper Closed NA Table
Timestep
Trim XZ Profiles

Vel Rate Creep Factor
Vel Rate Limit
Vel Rate Limit Minimum

Weir Approach
Weir Flow
WLL Approach
WLL Automatic
WLL No Weirs
WLL Vertical Offset
WLLp Interpolate Bed
Write Check Files

XS Database

.toc Commands

Cd
Cd Gate
Cd Gate Submerged
Cd Spillway

Define Control

End Define

Gate Height Fully Open
Gate Opening
Gate Seat Vertical Offset
Gate Speed
Gate Type
Gate Width Fully Open

Method

Operation

Period Opening/Closing
Period Startup/Shutdown
Pump Capacity
Pump Number
Pump Operation

Weir Height
Weir Height Speed
Weir Width
Weir Width Speed

.trfc Commands

IDW Exponent
IDW Maximum Distance
IDW Maximum Points

Maximum Hyetograph Points
Maximum RF Locations

Read GIS RF Points
Read GIS RF Polygons
Read GIS RF Triangles
RF Grid Cell Size
RF Grid Format
RF Grid Origin
RF Grid Size
RF Interpolation Method