RiverFlow2D
Two-Dimensional Flood and River Dynamics Model

Reference Manual

March – 2018

Hydronia LLC

RiverFlow2D© model and documentation produced by Hydronia, LLC, Pembroke Pines, FL. USA.
Information in this document is subject to change without notice and does not represent a commitment on part of Hydronia, LLC. The software described in this document is furnished under a license agreement.
RiverFlow2D, OilFlow2D, RiverFlow2D, and RiverFlow2D GPU are copyrighted by Hydronia, LLC. 2011-2017.
SMS is trademark of Aquaveo, LLC.
All other products or service names mentioned herein are trademarks of their respective owners.
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of Hydronia, LLC.
Last document modification date: August, 2017.
Technical Support: `support@hydronia.com`

Web site: `www.hydronia.com`

Contents

List of Figures . . . vii
List of Tables . . . x
1 Introduction . . . 1
1.1 Summary of RiverFlow2D Features and Capabilities . . . 2
1.1.1 Mesh Generator . . . 2
1.1.2 Numerical Engine . . . 2
1.1.3 Hydraulic Components . . . 3
1.1.4 Input Data Formats . . . 3
1.1.5 Initial Conditions . . . 3
1.1.6 Boundary Conditions . . . 3
1.1.7 Output Options . . . 4
1.1.8 Sediment Transport Module (Optional) . . . 4
1.1.9 Pollutant Transport Module (Optional) . . . 5
1.1.10 Mud and Debris Flow Module (Optional) . . . 5
1.1.11 OilFlow2D: Overland Oil Spills . . . 5
2 Installing RiverFlow2D SMS Edition . . . 7
2.1 Hardware Requirements . . . 7
2.2 Installing RiverFlow2D and SMS . . . 7
2.3 Activating SMS . . . 8
2.4 Linking RiverFlow2D to SMS . . . 8
2.5 RiverFlow2D Documentation . . . 9
2.6 RiverFlow2D Technical Support . . . 9
3 Overview of SMS Capabilities . . . 11
3.1 Modules . . . 12
3.1.1 Mesh Module . . . 13
3.1.2 Scatter Module . . . 13
3.1.3 GIS Module . . . 14
3.1.4 Map Module/Conceptual Model . . . 15
3.1.5 Visualization Tools . . . 15
3.1.6 Vectors . . . 15
3.1.7 Plots . . . 16
3.1.8 Animations . . . 16
3.1.9 Export Options . . . 16
3.1.10 CAD Support . . . 17
3.1.11 Import Wizard . . . 17
3.1.12 Image Support . . . 17
4 Overview of RiverFlow2D SMS edition . . . 19
5 RiverFlow2D Model Equations . . . 23
5.1 Hydrodynamic Model . . . 23
5.2 Assumptions of the Hydrodynamic Model . . . 23
5.3 Unsteady Flow Model . . . 24

5.4 Finite-Volume Numerical Solution . . . 24
5.4.1 Numerical Optimizations . . . 27
5.4.2 Stability Region . . . 27
5.5 Open Boundary Conditions . . . 27
5.5.1 Single Variable Boundary Condition Types (BCTYPE 1 and 6) . . . 30
5.5.2 Discharge Rating Table (BCTYPE 9) . . . 31
5.5.3 "Free" Open Boundaries (BCTYPE 10, 11) . . . 31
5.5.4 Uniform Flow Boundary Condition (BCTYPE 12) . . . 31
5.5.5 Numerical Implementation of Open Boundaries . . . 31
5.5.6 Closed Boundaries . . . 36
5.6 Dry/Wet Cell Modeling . . . 37
5.6.1 Cell definitions Based on Dry and Wet Conditions . . . 37
5.7 Volume Conservation . . . 38
5.8 Manning's n roughness Coefficients . . . 39
5.9 Sediment Transport Model: RiverFlow2D ST Module . . . 39
5.9.1 Model Equations . . . 39
5.9.2 Sediment Transport Laws . . . 40
5.9.3 Boundary Conditions for the Bed Load Sediment Model . . . 42
5.9.4 Suspended Load . . . 42
5.9.5 Boundary Conditions for the Suspended Sediment Model . . . 45
5.9.6 Geomorphological Collapse . . . 45
5.9.7 Sediment Transport Finite-Volume Numerical Solution . . . 45
5.9.8 Entering Data for the Sediment Transport Model . . . 46
5.9.9 Assumptions of the Sediment Transport Model . . . 46
5.10 Mud and Debris Flow Model: RiverFlow2D MD module . . . 47
5.10.1 Assumptions of the Mud and Debris Flow Model . . . 47
5.10.2 Rheological Formulation . . . 47
5.10.3 Entering Data for the Mud and Debris Flow Model . . . 50
5.11 Pollutant Transport Model: RiverFlow2D PL Module . . . 50
5.11.1 Model Equations . . . 50
5.11.2 Pollutant Transport Finite-Volume Numerical Solution . . . 51
5.11.3 Entering Data for the Pollutant Transport Model . . . 53
5.11.4 Assumptions of the Pollutant Transport Model . . . 54
6 Code Parallelization . . . 55
6.1 RiverFlow2D CPU . . . 55
6.2 RiverFlow2D GPU . . . 55
7 Hydraulic and Hydrologic Components . . . 57
7.1 Bridges Component . . . 58
7.1.1 Bridge Geometry Data File . . . 60
7.1.2 Bridge Calculations . . . 62
7.2 Bridge Piers . . . 66
7.2.1 Bridge Pier Calculation . . . 67
7.3 Culverts Component . . . 68
7.3.1 Culvert Calculation using a Rating Table (CulvertType = 0) . . . 70
7.3.2 Culvert Calculation using a Culvert Characteristics (CulvertType = 1,2) . . . 70
7.3.3 Assumptions of Culvert Calculations . . . 75
7.4 Gates Component . . . 75
7.4.1 Gate Calculations . . . 78
7.5 Rainfall, Evaporation and Infiltration . . . 78
7.5.1 Rainfall . . . 79
7.5.2 Infiltration . . . 79

7.5.3 Entering Rainfall, Evaporation and Infiltration Data . . . 83
7.6 Wind Component . . . 84
7.7 Internal Rating Tables . . . 84
7.7.1 Internal Rating Table Calculations . . . 87
7.7.2 Assumptions of Internal Rating Table Calculations . . . 87
7.8 Sources and Sinks . . . 87
7.9 Weirs . . . 89
7.9.1 Weir Calculations . . . 91
7.9.2 Assumptions of Weir Calculations . . . 92
7.10 Dam Breach Modeling . . . 92
8 RiverFlow2D Data Input Program (DIP) . . . 95
8.1 Control Data Panel (.DAT files) . . . 96
8.2 Sediment Transport Data Panel for the RiverFlow2D Plus model (.SEDS and .SEDB Files) . . . 100
8.3 Mud and Debris Flow Data Panel for the RiverFlow2D Plus model (.MUD File) . . . 103
8.4 Oil Spill on Land for the OilFlow2D model (.OILP File) . . . 104
8.5 Graphic Output Options Tab (.PLT File) . . . 104
8.6 Profile Output Panel (.PROFILES File) . . . 106
8.7 Cross Section Output Panel (.XSECS File) . . . 107
8.8 Inflow Boundary Data Panel . . . 107
8.9 Outflow Boundary Data Panel . . . 109
8.10 Culverts Panel (.CULVERTS File) . . . 111
8.11 Internal Rating Tables Panel (.IRT File) . . . 113
8.12 Weirs Panel (.WEIRS File) . . . 114
8.13 Sources/Sinks Panel (.SOURCES File) . . . 115
8.14 Open Boundary Conditions Panel (.IFL File) . . . 116
8.15 Bridge Piers Panel (.PIERS File) . . . 117
8.16 Observation Points Panel (.OBS File) . . . 117
8.17 Tools Panel . . . 119
8.17.1 Two-Dimensional Model Result Interpreter: 2dMRI . . . 119
8.17.2 HEC-RAS Data Extraction Tool . . . 119
9 Input Data File Reference . . . 121
9.1 Run Control Data . . . 123
9.1.1 Run Control Data File: .DAT . . . 123
9.2 Mesh Data . . . 130
9.2.1 Mesh Data File: .FED . . . 130
9.2.2 Open Boundary Conditions Data Files: .IFL and .OBCP . . . 132
9.2.3 Mesh Boundary Data File: .TBA . . . 137
9.3 Component data . . . 138
9.3.1 Bridges . . . 138
9.3.2 Bridges Data File: .BRIDGES . . . 139
9.3.3 Bridge Cross Section Geometry Data File . . . 140
9.3.4 Bridge Piers Drag Forces File: .PIERS . . . 143
9.3.5 Culverts Data File: .CULVERTS . . . 145
9.3.6 GATES Data Files: .GATES . . . 152
9.3.7 Gate Aperture Time Series File . . . 153
9.3.8 Internal Rating Table Data File: .IRT . . . 154
9.3.9 Mud and Debris Flow Data File: .MUD . . . 155
9.3.10 Oil Spill on Land Data File: .OILP . . . 157
9.3.11 Pollutant Transport Module Data File: .SOLUTES . . . 158
9.3.12 Pollutant Transport Module Initial Concentration Data File: .CINITIAL . . . 160

9.3.13 Manning's n Variable with Depth Data File: .MANNN . . . 160
9.3.14 Rainfall And Evaporation Data File: .LRAIN . . . 162
9.3.15 Infiltration Data File: .LINF . . . 164
9.3.16 RiverFlow2D Plus Sediment Transport Data Files: .SEDS and .SEDB . . . 166
9.3.17 Sources and Sinks Data File: .SOURCES . . . 171
9.3.18 Weirs Data File: .WEIRS . . . 172
9.3.19 Wind Data File: .WIND . . . 173
9.4 Output control data . . . 175
9.4.1 Observation Points Data File: .OBS . . . 175
9.4.2 Graphical Output Control Data File: .PLT . . . 176
9.4.3 Data for Profile Result Output: .PROFILES . . . 181
9.4.4 Cross Section Data for Result Output File: .XSECS . . . 181
9.5 Elevation data . . . 183
9.5.1 X Y Z data with header . . . 183
9.5.2 General X Y Z data . . . 184
9.6 Boundary conditions data . . . 184
9.6.1 One Variable Boundary Condition Files . . . 184
9.6.2 Two Variables Boundary Condition Files . . . 185
9.6.3 Multiple-Variable Boundary Condition Files . . . 186
9.6.4 Stage-Discharge Data Files . . . 187
9.6.5 Culvert Depth-Discharge Data Files . . . 188
10 Output File Reference . . . 189
10.1 Output File Overview . . . 189
10.2 General Output Files . . . 192
10.2.1 Run Options Summary . . . 192
10.2.2 Mesh Data and Mesh Metrics . . . 193
10.2.3 Run Summary . . . 197
10.2.4 Tabular Output Files . . . 198
10.2.5 Area of Inundation Files . . . 199
10.2.6 Maximum Value Tabular Files . . . 200
10.2.7 Observation Point Output Files . . . 201
10.2.8 Hot Start File . . . 202
10.3 Component Output Files . . . 203
10.3.1 Culvert Output Files . . . 203
10.3.2 Internal Rating Table Output Files . . . 205
10.3.3 Weir Output Files . . . 206
10.4 Profile and Cross Section Output Files . . . 207
10.4.1 General Cross Section Output Files . . . 207
10.4.2 Cross Section Hydrograph Output Files . . . 208
10.4.3 Profile Output Files . . . 209
10.5 Output Files for GIS Post-processing . . . 211
10.5.1 General Spatial Distribution Files for SMS (.HDF5) . . . 211
10.5.2 General Spatial Distribution Files (.EXP) . . . 212
10.5.3 Pollutant Concentration Files . . . 214
10.5.4 Suspended Sediment Concentration Files . . . 215
10.5.5 Maximum Value GIS Files . . . 216
10.5.6 Time-to-Depth GIS File . . . 216
10.5.7 Support Files to Generate ESRI Shapefiles . . . 217
10.5.8 Support Files to Generate USBR Hazard Maps in ESRI Shapefiles format (only for Argus ONE Post Processing) . . . 217

10.6 Output Files for QGIS Post-processing 218
10.6.1 General Results at Cells for QGIS 218
10.6.2 Maximum Values at Cells QGIS Files 219
10.6.3 Time-to-Depth at Cells QGIS File 219
10.6.4 Hazard Intensity Values at Cells QGIS Files 220
10.7 Graphic Output Files 220
10.7.1 DXF Files 220
10.7.2 GIF Output Files 220
10.7.3 VTK Output Files 221
11 Data Input Program Tools 223
11.1 HEC-RAS Data Extraction Tool 223
12 References 225

List of Figures

1.1 RiverFlow2D triangular-cell mesh. 2
1.2 SMS main window showing a velocity field computed with RiverFlow2D. 5

2.1 SMS executable Properties dialog with the *Run as administrator* option set. 8
2.2 Location of the RiverFlow2D Documentation folder. 9

3.1 SMS main window. 12
3.2 Mesh module. 13
3.3 Scatter module. 14
3.4 GIS module. 14
3.5 Map module. 15
3.6 Three-dimensional view of mesh terrain. 16

5.1 Piecewise uniform representation of the flow variables. 25
5.2 Cell parameters. 25
5.3 Cell parameters. 25
5.4 Open and closed boundary conditions. 28
5.5 Required gap between adjacent open boundary conditions. 28
5.6 Inflow water discharge imposed as velocities (BCTYPE 6). 30
5.7 Rectangular inlet cross section. 33
5.8 Irregular inlet cross section. 34
5.9 Evaluation of d_{min}. 34
5.10 New water level for the inlet section. 36
5.11 Solid wall condition. 37
5.12 Cell parameters. 46
5.13 RiverFlow2D Sediment Transport panel. 46
5.14 Rheological diagram of some common non-Newtonian formulations. 48
5.15 RiverFlow2D Mud/Debris Flow panel. 50
5.16 Physical representation of solute mass exchange between cells with $q^{\downarrow}_{i-1/2}, q^{\downarrow}_{i+1/2} > 0$. 52
5.17 Extraction of mass solute in an outlet boundary cell. 53
5.18 RiverFlow2D *Pollutant Transport* tab. 53

6.1 Speed up using RiverFlow2D parallelized code as a function of number of processor cores. 56
6.2 Speed up using RiverFlow2D parallelized code as a function of number of processor cores. 56

7.1 RiverFlow2D *Control Data* panel with Bridges component selected. 58
7.2 Bridge polyline. 59
7.3 Feature arc attributes. 59
7.4 Bridge dialog. 59

7.5 Bridge polyline on mesh. 60
7.6 Front view of a bridge cross section. 61
7.7 Top view of a bridge showing the cross sections of interest. Only two piers are depicted for simplicity. 63
7.8 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in free surface bridges. 63
7.9 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in a partially submerged bridges. 63
7.10 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in fully submerged bridges. 64
7.11 Application of the scheme in triangular structured meshes. Normal bridge (left) and oblique bridge (right). 64
7.12 Bridge pier proportions used to asses the influence of the structure width. 65
7.13 Influence of the structure width on the total head change (ΔH) across the bridge as a function of the Froud number downstream. 65
7.14 RiverFlow2D *Control Panel* dialog with Bridge Piers component selected. 66
7.15 Pier Data dialog. 67
7.16 Piers inside cells. 67
7.17 Schematic view of a rectangular pier. 68
7.18 RiverFlow2D *Global Parameters* dialog with Culverts Component selected. 68
7.19 Schematic cut view perpendicular to a gate structure. 75
7.20 Flow modes across gates. 76
7.21 RiverFlow2D *Control Data* panel with Gates Component selected. 76
7.22 Gate polyline. 77
7.23 Feature arc attributes. 77
7.24 Gates dialog. 77
7.25 Water levels for discharge under a gate in submerged conditions formulated as in (G1). 78
7.26 Water levels for discharge under a gate in submerged conditions formulated as in (G2). 78
7.27 RiverFlow2D Rainfall/Infiltration panel. 84
7.28 RiverFlow2D *Control Data* panel with IRT component selected. 85
7.29 Internal Rating Table polyline. 85
7.30 Feature arc attributes. 86
7.31 *Internal Rating Table* dialog. 86
7.32 Window to enter data pairs (discharge-WSE) for an Internal Rating Table. 86
7.33 *Internal Rating Table* dialog. 87
7.34 RiverFlow2D *Control Data* panel with the Sources and Sinks component selected. . 88
7.35 Source/Sink dialog. 89
7.36 Time series editor to enter discharge for a Source/Sink. 89
7.37 RiverFlow2D Control Data panel with Weirs component selected. 90
7.38 Weir polyline. 90
7.39 Feature arc attributes. 91
7.40 Weirs data dialog. 91
7.41 Coordinate system for the dam breach representation. 93

8.1 RiverFlow2D DIP Open Project Dialog. 95
8.2 Main RiverFlow2D Data Input Program window. 96
8.3 RiverFlow2D Model Selection Dialog. 96
8.4 Control Data panel. 97
8.5 Sediment Transport Panel (RiverFlow2D Plus model). 101
8.6 Mud/Debris Flow Panel (RiverFlow2D Plus model). 103

8.7 Oil Spill on Land Panel (OilFlow2D model). 104
8.8 Graphic Output Panel. 105
8.9 Color Palettes. 105
8.10 Profile Output File. 107
8.11 Cross Section Output Panel. 108
8.12 Inflow Boundary Data Panel. 108
8.13 Inflow hydrograph plot. 109
8.14 Outflow Boundary Data Panel. 110
8.15 Culverts Panel showing data in rating curve. 111
8.16 Internal Rating Tables Panel. 113
8.17 Weirs Panel. 114
8.18 Sources/Sinks Panel. 115
8.19 Open Boundary Conditions Panel. 116
8.20 Bridge Piers Panel. 117
8.21 Observation Points Panel. 118
8.22 Two-dimensional Model Result Interpreter: 2dMRI. 119
8.23 Two-dimensional Model Result Interpreter: 2dMRI. 120

9.1 Example of a RiverFlow2D Mesh. 121
9.2 Front view of a bridge cross section. 141
9.3 Color palettes. 179

10.1 Output color coded by depth and with velocity vectors 221

11.1 HEC-RAS command on tools panel. 224

List of Tables

4.1 SMS Menu for RiverFlow2D. 19
4.2 SMS Modules on RiverFlow2D SMS edition. 21

5.1 Required boundary conditions. 29
5.2 Open Boundary condition types for hydrodynamics. 30
5.3 Summary of bed load transport formulas. 41
5.4 Formulas to calculate sediment settling velocity ω_s. 43
5.5 Rheological flow resistance formulations used in RiverFlow2D MD. 49

7.1 Variable Descriptions for the bridge cross section geometry file. 62
7.2 Manning's n roughness coefficients for various culvert materials. Adapted from Froehlich (2003). 71
7.3 Entrance loss coefficients K_e. Adapted from Froehlich (2003). 71
7.4 Culvert inlet control formula coefficients. Adapted from Froehlich (2003). 72
7.5 Culvert inlet configurations. Adapted from *www.xmswiki.com/xms/*. 73
7.6 Horton initial infiltration for different soils. Source: Akan(1993). 79
7.7 Horton final infiltration for different soils. Source: Akan(1993). 80
7.8 Mean values and standard deviation for Green-Ampt model parameters. Source: Rawls et al. (1983). 81
7.9 Antecedent Moisture Content groups (adaptation from Mishra et al. (2003) . . . 83

8.1 Buttons in the Control Data Panel. 97
8.2 Time Control Data Frame on the Control Data Panel. 98
8.3 Units Frame on the Control Data Panel. 98
8.4 Computation Control Data Frame on the Control Data Panel. 98
8.5 Closed Boundary Conditions Data Frame on the Control Data Panel. 98
8.6 Output Options Data Frame on the Control Data Panel. 98
8.7 Components Data Frame on the Control Data Panel. 99
8.8 Components Data Frame on the Control Data Panel. 100
8.9 Parameters on the Sediment transport Mode frame and buttons of the Sediment Transport Panel (RiverFlow2D Plus model). 100
8.10 Parameters on the Suspended Sediment transport frame of the Sediment Transport Panel (RiverFlow2D Plus model). 101
8.11 Parameters on the Bed Load Sediment transport frame of the Sediment Transport Panel (RiverFlow2D Plus model). 102
8.12 Parameters on the Mud/Debris Flow Panel (RiverFlow2D Plus model). 103
8.13 Parameters on the Oil Spill on Land Panel (OilFlow2D Plus model). 104
8.14 Parameters on the Graphic Output Option Panel. 105
8.15 Parameters on the Profile Output Panel. 107
8.16 Parameters on the Inflow Boundary Data Panel. 107
8.17 Parameters on the Inflow Boundary Data Panel. 109

8.18 Parameters on the Outflow Boundary Data Panel. 110
8.19 Parameters on the Culverts Panel. 111
8.20 Parameters on the Internal Rating Tables Panel. 113
8.21 Parameters on the Weirs Panel. 114
8.22 Parameters on the Sources/Sinks Panel. 115
8.23 Parameters on the Open Boundary Condition Panel. 116
8.24 Parameters on the Bridge Piers Panel. 117
8.25 Parameters on the Observation Points Panel. 118
8.26 Files generated by the HEC-RAS Data Extraction Tool. 120

9.1 List of Input Data Files. 122
9.2 Variable Descriptions for the .DAT File. 125
9.3 Variable Descriptions for the .FED File. 131
9.4 Variable Descriptions for the .IFL and .OBCP Files. 133
9.5 Boundary Condition Types. 134
9.6 Supercritical Flow Regime Boundary Conditions. 135
9.7 Variable Descriptions for the .DAT File. 138
9.8 Variable Descriptions for the .BRIDGES File. 140
9.9 Variable Descriptions for the bridge cross section geometry file. 142
9.10 Variable Descriptions for the .PIERS File. 143
9.11 Drag Coefficients for Bridge Piers. Adapted from Froehlich (2003). 144
9.12 Variable Descriptions for the .CULVERTS File. 145
9.13 Variable Descriptions of Culvert Depth-Discharge Data files. 146
9.14 Variable Descriptions for the Culvert Characteristic file. 147
9.15 Manning's n roughness coefficients for various culvert materials. Adapted from Froehlich (2003). 148
9.16 Entrance loss coefficients K_e. Adapted from Froehlich (2003). 148
9.17 Culvert inlet control formula coefficients. Adapted from Froehlich (2003). 149
9.18 Culvert inlet configurations. (Adapted from `www.xmswiki.com/xms/`). 151
9.19 Variable Descriptions for the .GATES File. 153
9.20 Variable Descriptions for the .GATES File. 153
9.21 Variable Descriptions for the .IRT File. 154
9.22 Variable Descriptions for the .MUD File. 156
9.23 Flow resistance relation. 156
9.24 Variable Descriptions for the .OILP File. 157
9.25 Explanation of the example .SOLUTES file. 159
9.26 Variable Descriptions for the .SOLUTES File. 160
9.27 Variable Descriptions for the .MANNN File. 161
9.28 Variable Descriptions for the Manning's n Cariable with Depth Data File. 162
9.29 Variable Descriptions for the .LRAIN File. 163
9.30 Variable Descriptions for the Hyetograph and Evaporation Data File. 163
9.31 Variable Descriptions for the .LINF File. 165
9.32 Variable Descriptions for the Infiltration Parameter File. 166
9.33 Variable Descriptions for the .SEDS File. 168
9.34 Variable Descriptions for the .SEDS File. 170
9.35 Variable Descriptions for the .SOURCES File. 172
9.36 Variable Descriptions for the .WEIRS File. 173
9.37 Variable Descriptions for the .WIND File. 174
9.38 Variable Descriptions for the Wind Velocity File 175
9.39 Variable Descriptions for the .OBS File 176
9.40 Variable Descriptions for the .PLT File 177

9.41 Supported image formats and their corresponding world file extensions. 180
9.42 Variable Descriptions for the .PROFILES File 181
9.43 Variable Descriptions for the .XSECS File 182
9.44 Variable Descriptions for the .EXP File. 183
9.45 Variable Descriptions of Boundary Condition Files. 185
9.47 Variable Descriptions of Two-Variable Boundary Condition Files. 186
9.49 Variable Descriptions of Multiple-Variable Boundary Condition Files. 187
9.50 Variable Descriptions of Two-Variable Boundary Condition Files. 188
9.51 Variable Descriptions of Culvert Depth-Discharge Data Files. 188

10.1 List of Output Data Files. 189
10.2 Variables reported on the .MESHOUTI and .MESHOUTE Files. 193
10.3 Variables Reported on the Observation Point Files. 200
10.4 Variables Reported on the Observation Point Files. 201
10.5 Variables Reported on the Weir Point Files. 206
10.6 Variables Reported on _RESULTS.HDF5 Output Files. 211
10.7 Dynamic Variables Reported on the <ProjectName>_MAXIMUM_RESULTS.HDF5 files. 212
10.8 Dynamic Variables Reported on the <ProjectName>_MAXIMUM_RESULTS.HDF5 files. 212
10.9 Variables Reported on TIME_.EXP Output Files. 213
10.10 Variables Reported on CONC*.EXP Output Files. 214
10.11 Variables Reported on ST*.EXP Output Files. 215
10.12 Variables Reported on the <ProjectName>_..._MAX.EXP 216
10.13 Variables Reported on the <ProjectName>_time2depths.exp File. 217
10.14 USBR Hazard Classification in USBR_Hazard.EXP file. 217
10.15 Variables Reported on CELL_TIME_*.TEXTOUT Output Files. 218
10.16 Variables Reported on the <ProjectName>_..._CELLS_MAX.TEXTOUT 219
10.17 Variables Reported on the <ProjectName>_TIME2DEPTHS_CELLS.TEXTOUT File. 220
10.18 Variables Reported on the <ProjectName>_CELLS_HAZARD.TEXTOUT File. . 220

11.1 HEC-RAS files with elevation data. 223

1 Introduction

RiverFlow2D is a combined hydrologic and hydraulic, mobile bed and pollutant transport finite-volume model for rivers, estuaries and floodplains. It can route floods in rivers and simulate inundation over floodplains and complex terrain at high resolution and with remarkable speed, stability, and accuracy. The use of adaptive triangular-cell meshes enables the flow field to be resolved around key features in any riverine environments.

RiverFlow2D user interface is based upon Surface-Water Modeling Solution (SMS) developed by Aquaveo. This GIS-integrated software system provides interactive functions to generate and refine the flexible mesh used by RiverFlow2D. SMS uses familiar GIS objects such as points, arcs (polylines), and polygons to construct a high level representation of the model, facilitating assigning boundary conditions and roughness values, and allowing the user to efficiently manage the entire modeling process. SMS offers a comprehensive set of visualization tools including 3D rendering, animations, and exporting graphs to Google Earth.

RiverFlow2D computation engine uses an accurate, and stable finite-volume solution method that eliminates the boundary and hot start difficulties of some two-dimensional flexible mesh models. The model can integrate hydraulic structures such as culverts, weirs, bridges, gates and internal rating tables. The hydrologic capabilities include spatially distributed rainfall, evaporation and infiltration. This version also accounts for distributed wind stress on the water surface.

This reference manual provides instructions to install the RiverFlow2D model, and explains the fundamentals of the model and its components, as well as the numerical methods used to solve the governing equations. It also presents detailed description of the input data files, and output files. A separate document includes several tutorials that help you getting started using RiverFlow2D and learning to apply several model components such as bridges, culverts, rainfall and infiltration, weirs, etc.

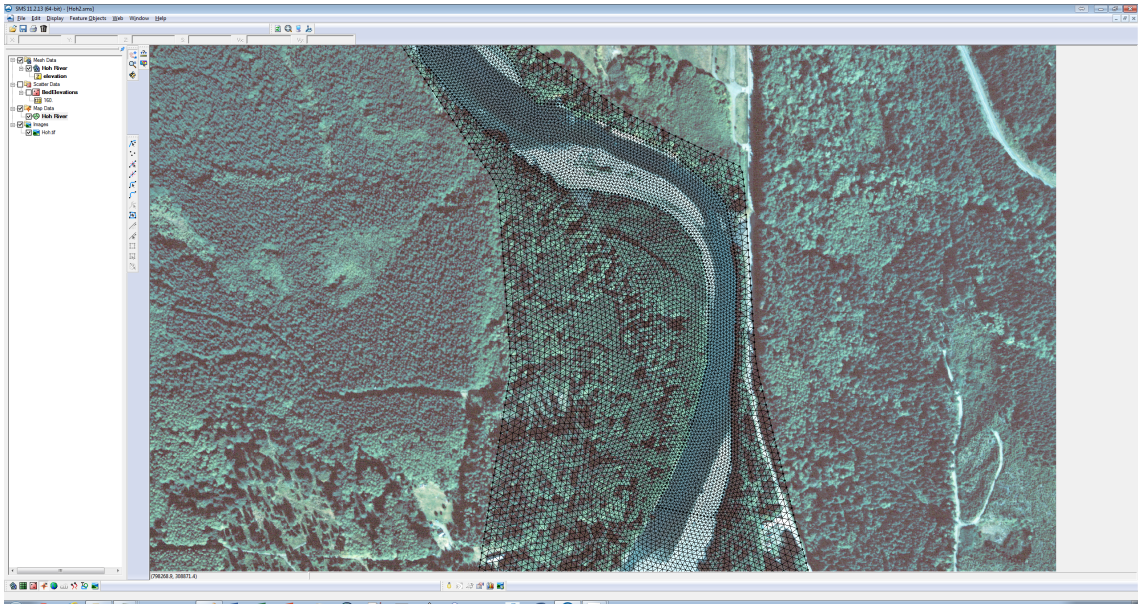


Figure 1.1 – RiverFlow2D triangular-cell mesh.

1.1 Summary of RiverFlow2D Features and Capabilities

1.1.1 Mesh Generator

- Automatic generation of flexible triangular-cell mesh
- Unlimited mesh refinement
- Mesh quality report
- Spatially varied interpolation methods including
  - Triangular (TIN)
  - Inverse Distance Weighting
  - Sampling, and many more

1.1.2 Numerical Engine

- Spatial discretization using triangular elements
- High performance Finite-Volume engine
- Double-precision computations for higher accuracy
- Fully parallelized with OpenMP for faster execution in Multiple-Core computers
- GPU version for >150X faster simulations

1.1.3 Hydraulic Components

- Culverts using the US Federal Highway Administration (FHWA) formulation
- Bridge hydraulics in 2D including pressure flow and overtopping
- Bridge pier drag forces
- Weirs with variable crest elevations
- Gates
- Internal hydraulic structures
- Sources and sinks
- Spatially distributed rainfall and evaporation
- Spatially distributed infiltration
- Spatially distributed wind stress
- Depth dependent Manning's n

1.1.4 Input Data Formats

- Metric or English units
- ASCII X, Y, Z
- ESRI ASCII grid files
- USGS DEM
- ESRI shapefiles
- Autodesk DXF
- TIFF, GIF, JPG, etc. raster's

1.1.5 Initial Conditions

- Dry-bed
- User-defined water elevations

1.1.6 Boundary Conditions

- Water discharge hydrograph
- Water discharge and water elevation vs time
- Water elevation vs time
- Uniform flow
- Rating tables

- Free outflow
- Inflow suspended sediment concentrations (ST Module)
- Inflow pollutant concentrations (ST Module)

1.1.7 Output Options

- Results at cross sections and profiles
- Observation points: time series at user selected locations
- Dynamic plots while the model runs
- Velocity field, depth and water surface elevations
- Bed shear stress
- Erosion and deposition depths
- Maximum velocity, water elevations, erosion or deposition depths, etc.
- Bed elevation changes
- Froude number
- Sediment fluxes
- Time to 0.3 m (1 ft), time to 1 m (3 ft) and time to peak depth
- Inundation time during which depth is greater than 0.1 m or 4 in
- Polygon, line and point ESRI shapefiles
- Autodesk DXF mesh and velocity fields
- GIS post processing plots including shapefiles and raster images
- Paraview VTK

1.1.8 Sediment Transport Module (Optional)

- Separate calculation of suspended and bed load sediment transport
- Multiple size fractions
- Bed-changes (erosion-deposition)
- 10 sediment-transport formulas
- Sediment transport over rigid bed
- Maximum erosion depth areas

1.1.9 Pollutant Transport Module (Optional)

- Advection-Dispersion-Reaction
- Reaction rates between pollutants/solutes
- Simultaneous computation of multiple solutes

1.1.10 Mud and Debris Flow Module (Optional)

- Non-Newtoninan fluids
- Seven rheological formulations
- Granular flow

1.1.11 OilFlow2D: Overland Oil Spills

- Model to simulate spills of crude oils and viscous fluids over complex terrain



Figure 1.2 – SMS main window showing a velocity field computed with RiverFlow2D.

2 Installing RiverFlow2D SMS Edition

RiverFlow2D requires SMS version 11.2 or later. This section will assist you to setup the RiverFlow2D model to enable using the it from SMS.

2.1 Hardware Requirements

RiverFlow2D is supported on 64-bit computers running MS-Windows 7, 8.1, 10 platforms. It is recommended to have a minimum of 4 GB of RAM and at least 10 GB of free hard disk space. RiverFlow2D is capable of running in modern Intel single processor computers. If multiple-core processors (Duo, Quad, etc.) are available, the model can execute in parallel processor mode, thereby running much faster than in single processor computers. In addition, if you have a GPU license, the model can take advantage of Graphic Processing Unit (GPU) cards to run up to 150 times faster than in single-processor computers.

It is recommended to uninstall all previous RiverFlow2D SMS setups before installing this version.

2.2 Installing RiverFlow2D and SMS

To install RiverFlow2D, please follow these steps.

1. When installing from a DVD, please insert the RiverFlow2D installation disk into your DVD drive, double click on the RiverFlow2D SMS setup file and follow the on-screen instructions. If you received a setup file link, please download the file and double-click on it.

2. If you are installing on a PC running Windows 7 or later, you must be logged on the PC as an administrator before you begin the installation.

2.3 Activating SMS

Depending on the SMS licensing options you have, you may need to use different procedures to activate the software. Please go to Aquaveo website at `http://www.aquaveo.com/license-instructions` to find specific instructions to your particular SMS license.

2.4 Linking RiverFlow2D to SMS

In order to use RiverFlow2D from SMS GUI, it is necessary to link RiverFlow2D to SMS as follows:

1. Run the SMS program
2. On the *Edit* menu, click *Preferences...*
3. Click the *File Locations* Tab
4. Scroll down in the Model Executables frame till you see the Generic model.
5. Click the *Browse* button and find the folder `C:\ProgramFiles(x86)\SMSRiverFlow2DPlusv4` and select the model executable file `RiverFlow2Dm4.exe`.
6. Click OK to exit.

Often you may find necessary to run SMS in administration mode. To set that mode of operation, assuming you have SMS v12.1, use the Windows File Explorer and go to the `C:\ProgramFiles\SMS12.164-bit` folder, right-click on `SMS121.exe`, and then select *Run as administrator* as indicated in Figure 2.1

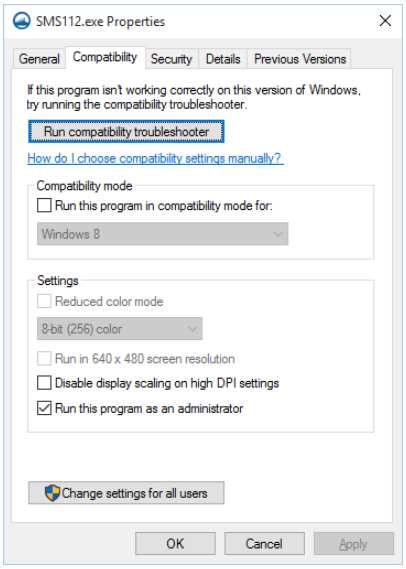


Figure 2.1 – SMS executable Properties dialog with the *Run as administrator* option set.

2.5 RiverFlow2D Documentation

Find the RiverFlow2D documentation including this manual in the following folder:

`\RiverFlow2D_SMS\Documentation` usually installed under `...\Documents` or `\MyDocuments` (see Figure 2.2).

Also under `\RiverFlow2D_SMS`, you will find example projects, videos, presentations, and other useful resources.

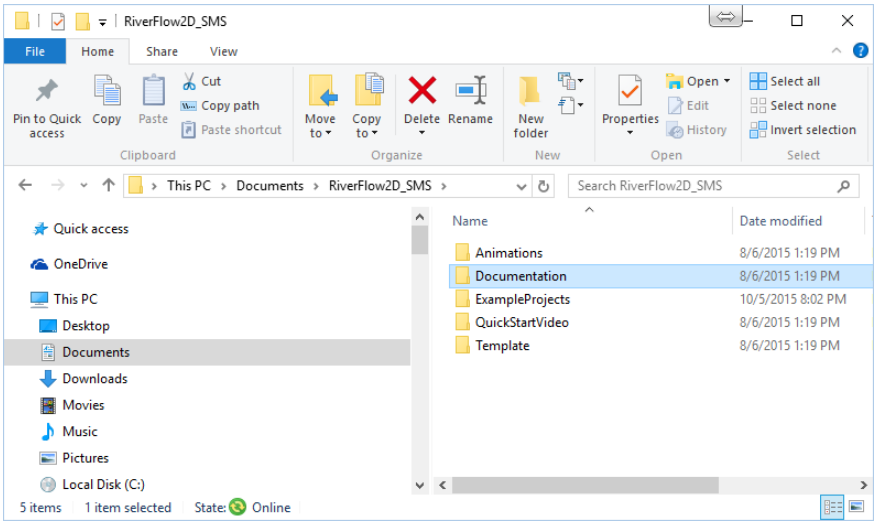


Figure 2.2 – Location of the RiverFlow2D Documentation folder.

2.6 RiverFlow2D Technical Support

If you have any questions or need assistance, please email our support team at: `support@hydronia.com`. Please make sure you visit our web site `www.hydronia.com` regularly to find out about new products and updates.

3 Overview of SMS Capabilities

SMS is a general purpose Graphical User Interface (GUI) for free-surface flow models. SMS represents the hydraulic or hydrologic problem based on a *Conceptual Model* that consists of a simplified representation of the situation being simulated. The Conceptual Model includes the geometric attributes of the area such as the extents, the boundary conditions acting on the domain (such as inflow discharges or water levels) and the physical characteristics (such as bed roughness). The Conceptual Model does not include numerical details such as mesh elements, etc. It is helpful to construct the model using feature objects in the Map module and having an aerial image in the background as a reference. After building the Conceptual Model, a mesh can be generated and the data will be automatically transferred to the mesh cells and nodes. Figure 3.1 shows the main parts of the SMS window.

A variety of tools in SMS allow the user to manage the data, set up the project and post-process output results. The primary tools and windows can be described as follows:

- Main Graphic Window: the data management is done in this window.
- Toolbox:
    1. Modules: these icons control what menu commands and tools are available at any given time while operating in SMS. Each module corresponds to a specific type of data: triangular-cell mesh, scattered data, etc. The commands that operate each module are available once the module is active.
    2. Static Tools: set of tools used to manipulate the display. They do not change for different modules.
    3. Dynamic Tools: tools used to create and edit entities specific to the module. They change according to the selected module and the active model.
    4. Macros: shortcuts for menu commands. They can be set in the *Preferences* dialog accessible on the *Edit* menu and clicking *Preferences...*
- Project Explorer or Tree Window: to view all data that makes up a part of a project. It is used to switch modules, select coverage to work with, select a base set to be active, and set display settings of the various entities in the active coverage. By right clicking on various entities in the project explorer, the user may also transform, copy, or manipulate the entity.

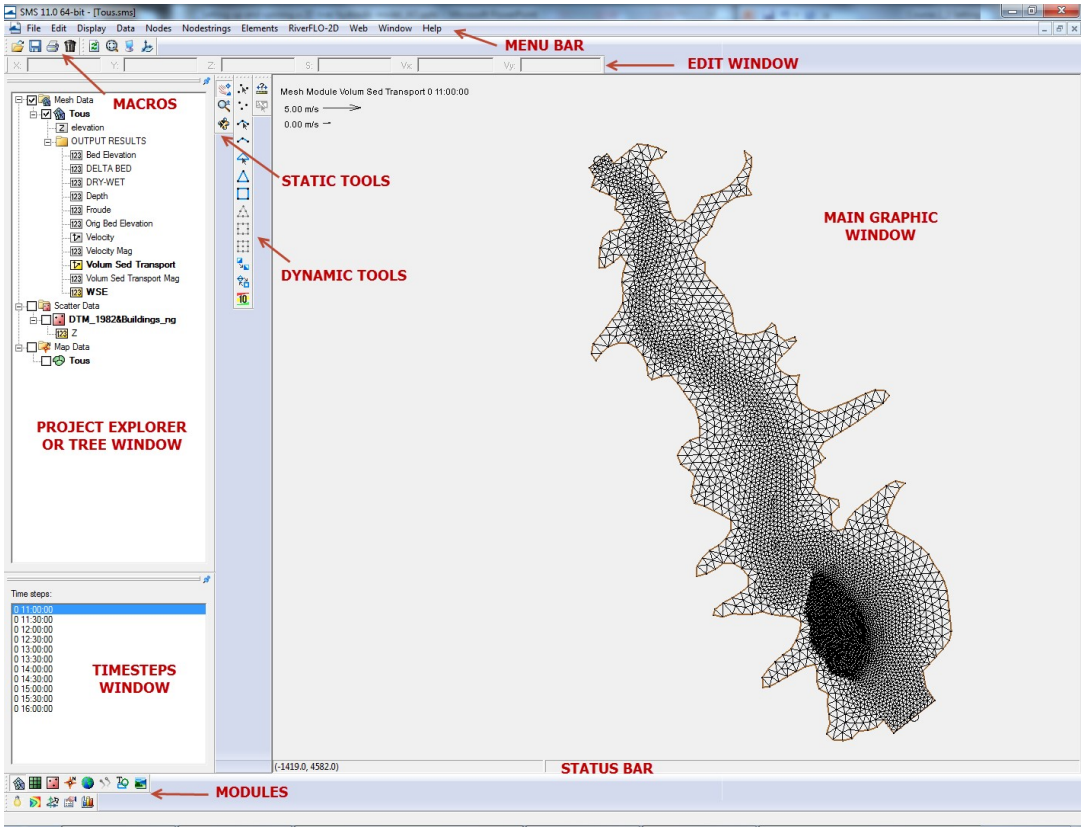


Figure 3.1 – SMS main window.

- Time Steps Window: it is used to select results for specific time steps when a transient dataset has been loaded into the project.
- Edit Window: It is used to show and/or change the coordinates of selected entities.
- Menu Bar: Allow managing data depending on the active module.
- Status Bar: it shows the mouse coordinates when the model is in plan view and the information for selected entities.

3.1 Modules

Data is divided into modules based upon the data type they can accommodate.



- There is always one active module.
- The menus and toolbars change depending on the active module.
- The current module may be selected in the module bar or by selecting an object in the project explorer.

3.1.1 Mesh Module

The mesh module is used to create, edit, and visualize mesh data. RiverFlow2D meshes are also referred to as unstructured grids and are defined by nodes and elements also called cells.

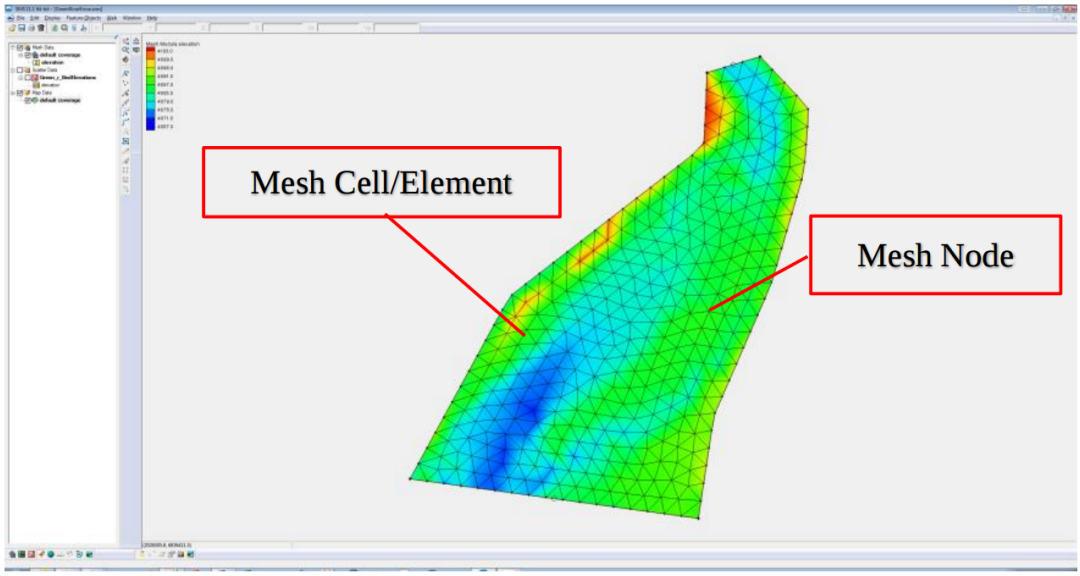


Figure 3.2 – Mesh module.

3.1.2 Scatter Module

The Scatter Module is used to create, edit, and visualize Triangulated Irregular Networks (TINs).

- DEM's can be read in and converted to TINs.
- Filter scatter sets to eliminate redundant data.
- Datasets can be interpolated to other modules, e.g. a mesh.

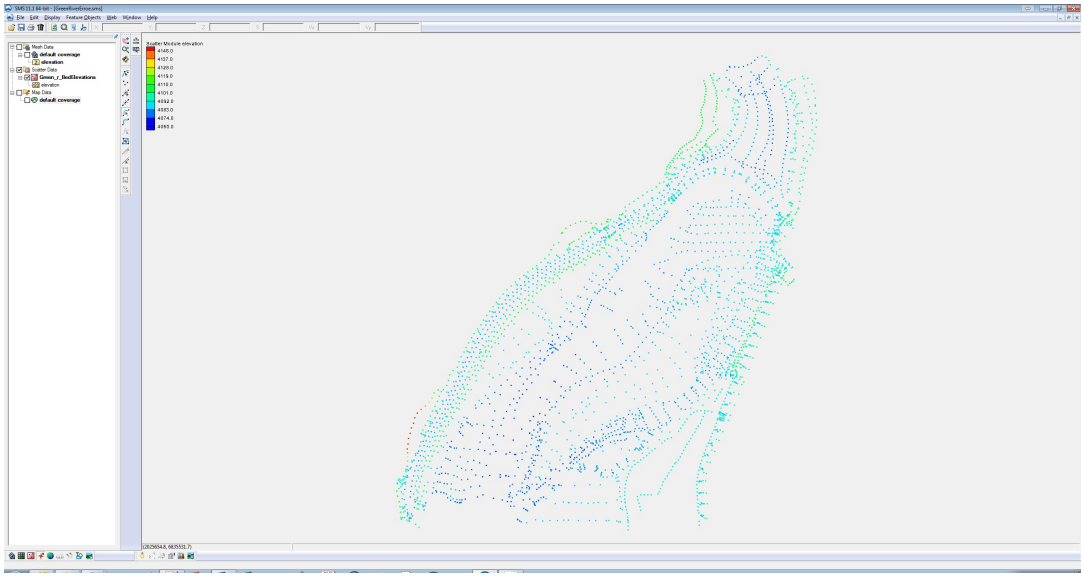


Figure 3.3 – Scatter module.

3.1.3 GIS Module

The GIS Module is used to import and visualize GIS data such as shapefiles.

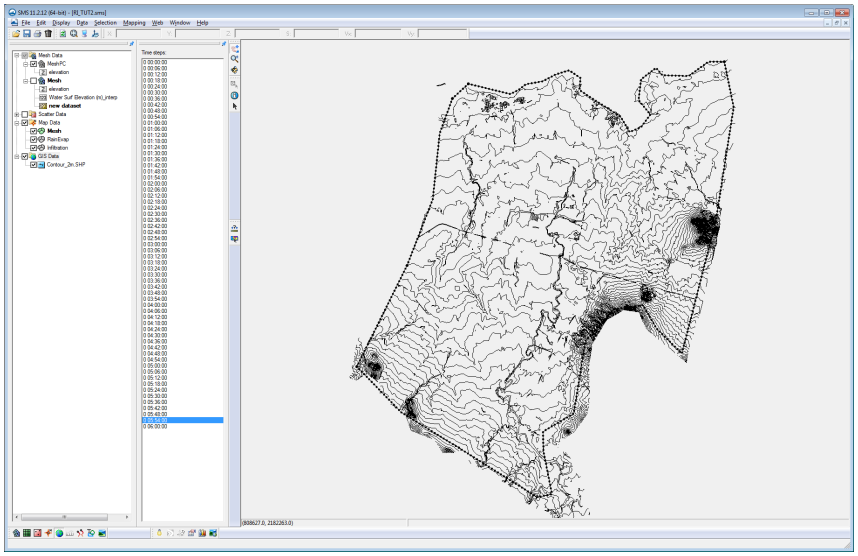


Figure 3.4 – GIS module.

- Supports ESRI and MapInfo formats.
- Uses ESRI files to use ArcGIS visualization options.
- GIS data can be converted to feature data to the Map module.

3.1.4 Map Module/Conceptual Model

The Map Module provides the GIS containers that allow creating the conceptual model

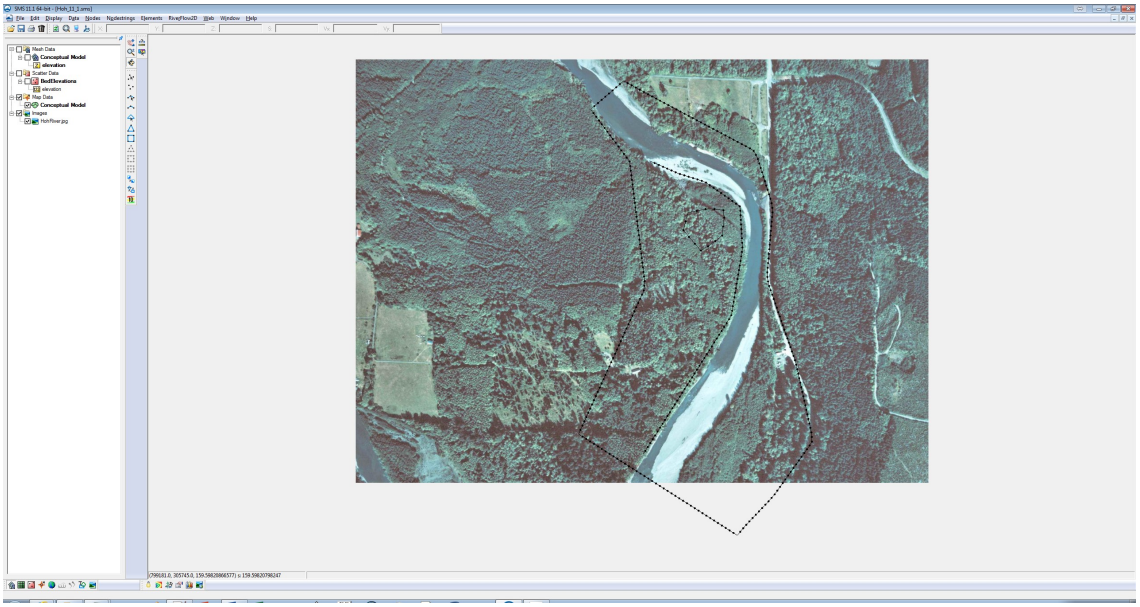


Figure 3.5 – Map module.

- Create and edit GIS data.
- Used to create conceptual models as well as data for other purposes.
- Conceptual model is a geometry, mesh independent representation of the numeric model domain and its boundary conditions.
- Features can be converted to model geometry and boundary conditions.
- Conceptual model makes it easier to create, and edit model data.

3.1.5 Visualization Tools

- Visualize scalar and vector datasets.
- Linear, color filled or both.
- Transparency.
- Full control of ranges and colors.
- Precision control for labels and legends.

3.1.6 Vectors

- Visualize vector datasets as arrows.
- Constant size or vary by magnitude.

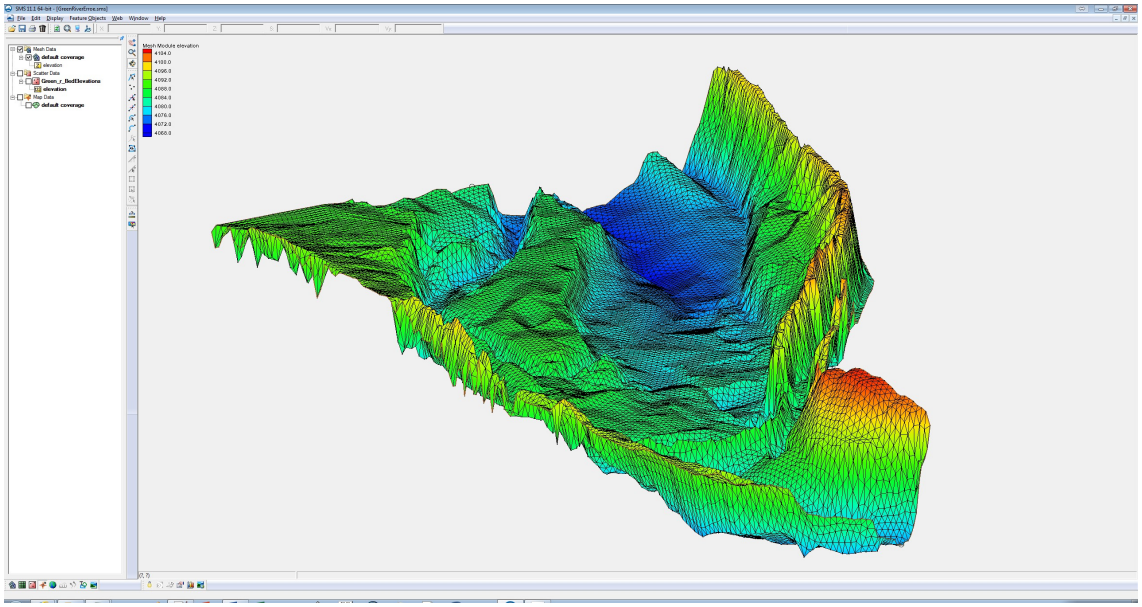


Figure 3.6 – Three-dimensional view of mesh terrain.

- Show just a range of magnitudes.
- Color by magnitude.

3.1.7 Plots

- 2D plots to visualize results and compare to measured values.
- Profile plots view scalar data along polylines.
- Time-series plots view scalar, vector, or flux (flowrate) data at a point or across a polyline.
- Several kinds of plots can be used to compare model results with measured data.

3.1.8 Animations

- Several types of AVI animations (film loops) can be generated.
- Transient data animation shows model changes through time (contours, vectors, etc.).
- Flow trace uses vector data to generate flow paths through the geometry.
- Drogue plots use user specified starting locations and show how the particles would flow through a vector field.
- Multiple view animations show the data while transitioning between different views.

3.1.9 Export Options

- Graphics window can be copied to the clipboard.
- Current view can be exported in KML format for visualization in Google Earth.

3.1.10 CAD Support

- AutoCAD DXF and DWG files can be read into SMS.
- CAD data is displayed in 3D.
- CAD data can be converted to map or scatter data.

3.1.11 Import Wizard

- Read columnar ASCII data into SMS.
- Columns can be fixed width or delimited by specific characters.
- Data can be read as mesh, scatter, or map data.

3.1.12 Image Support

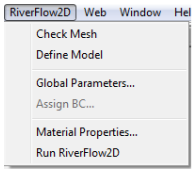
- Multiple images can be read/viewed at the same time.
- Many image formats are supported including JPG, TIFF, PNG, MrSID, ECW, and more.
- Images can be geo-referenced to view images along with other data.
- Images can be draped over mesh or scatter data.
- Image pyramids can be created which improves viewing at various zoom levels.

4

Overview of RiverFlow2D SMS edition

RiverFlow2D SMS Generic Model includes a set of standard options organized in groups that facilitate the user interaction with the model. The standard window groups include: Check Mesh, Define Model, Global Parameters, Assign BC, Material Properties and Run RiverFlow2D shown in Table 4.1.

Table 4.1 – SMS Menu for RiverFlow2D.

RiverFlow2D Menu	Tab Name	Type	Content
	Check Mesh	Information	Identifies potential issues in the construction of the mesh or data.
	Define Model	Template	Group locked. It contains the structure of a RiverFlow2D Template.
	GLOBAL PARAMETERS	TAB	WINDOW TO ENTER DATA
	Control Data	Data	Model run control parameters such as simulation time, etc.
	Graphic Output Options	Data	Parameters that can be set for the graphical output plots that are generated during runtime.
	Sediment Transport	Data	Parameters for the Sediment Transport Model.
	Mud and Debris Flow	Data	Parameters for the Mud and Debris Flow Model.
	Pollutant Transport	Data	Parameters for the Pollutant Transport Model.
	Rainfall / Evaporation	Data	Rainfall and Evaporation Data.

Continued on next page

Table 4.1 – continued from previous page

RiverFlow2D Menu	Tab Name	Type	Content
	Component Controls	Data	Turn on and off the different components as piers, weirs, culverts, sources and sinks, bridges, gates and internal rating tables.
	ASSIGN BOUNDARY CONDITIONS	TAB	ASSIGN BC's
	Exterior	Data	Specify boundary conditions on exterior boundaries.
	Interior	Data	Specify internal conditions on a polyline.
	Bridges	Data	Specify data for the Bridges component.
	Gates	Data	Specify data for the Gates component.
	Weir	Data	Specify data for the Weirs component.
	MATERIAL PROPERTIES	TAB	ASSIGN ROUGHNESS PARAMETERS
	Materials	Data	Enter Manning's n that will be assigned to polygons in the conceptual model.
	Run	RiverFlow2D	To execute RiverFlow2D
	MESH	MODULE	CREATION OF NODES/ELEMENTS
	Mesh	2D unstructured grids	Define the computational domain of the numerical model that consists of nodes group together to form elements.
	Elevation	Terrain Data	Bed elevations data file.
	Output Results	Nodal Values	Model output (.hdf5) will be read as mesh datasets. Scalar datasets are identified by the 123 icon, and the vector dataset have the vector icon.
	SCATTER	MODULE	INTERPOLATE SPATIAL DATA VALUES
	Scatter	Nodal values	Interpolate values using one of the available interpolation schemes including linear, natural neighbor and inverse distance weighted. The module can be also used to view and edit survey data.
	MAP	MODULE	USE OF GIS OBJECTS TO BUILD CONCEPTUAL MODEL
	Conceptual Model	GIS Objects	GIS objects such as points, nodes, arcs and polygons are grouped into layers or coverages and use to create the hydraulic model. The coverage type determines the attributes available for the objects in the coverage.

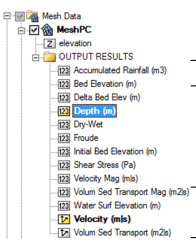
Continued on next page

Table 4.1 – continued from previous page

RiverFlow2D Menu	Tab Name	Type	Content
	GIS	MODULE	TO LINK WITH GIS DATA
	GIS	Data	Allow users to handle large files more efficiently importing large files and then select and convert to feature objects only the portions that are needed.

The commands in SMS are divided based on the types of data they operate on. The dynamic toolbar and available menu commands instantaneously change when the user switches from one module to another. Only one module is active at any given time. The data associated with a module (e.g. a 2D mesh) is preserved when the user switches to a different module. The different SMS Modules used to set up a project in SMS are described below:

Table 4.2 – SMS Modules on RiverFlow2D SMS edition.

SMS Module	Name	Type	Content
	Check Mesh	Information	Identifies potential issues in the construction of the mesh or data.
	MESH	MODULE	CREATION OF NODES/ELEMENTS
	Mesh	2D unstructured grids	Define the computational domain of the numerical model that consists of nodes group together to form elements.
	Elevation	Terrain Data	Bed elevations data file.
	Output Results	Nodal Values	Model output (.hdf5) will be read as mesh datasets. Scalar datasets are identified by the 123 icon, and the vector dataset have the vector icon.
	SCATTER	MODULE	INTERPOLATE SPATIAL DATA VALUES
	Scatter	Nodal values	Interpolate values using one of the available interpolation schemes including linear, natural neighbor and inverse distance weighted. The module can be also used to view and edit survey data.
	MAP	MODULE	USE OF GIS OBJECTS TO BUILD CONCEPTUAL MODEL
	Conceptual Model	GIS Objects	GIS objects such as points, nodes, arcs and polygons are grouped into layers or coverages and use to create the hydraulic model. The coverage type determines the attributes available for the objects in the coverage.
	GIS	MODULE	TO LINK WITH GIS DATA
	GIS	Data	Allow users to handle large files more efficiently importing large files and then select and convert to feature objects only the portions that are needed.

5 RiverFlow2D Model Equations

5.1 Hydrodynamic Model

One-dimensional hydraulic models are not adequate to simulate flooding when flows are unconfined or velocities change direction during the course of the hydrograph. The cost of non-simplified three-dimensional numerical models can be avoided using depth averaged two dimensional (2D) shallow water equations [Toro 2001].

When dealing with the shallow water equations, realistic applications always include source terms describing bed level variation and bed friction that, if not properly discretized, can lead to numerical instabilities. In the last decade, the main effort has been put on keeping a discrete balance between flux and source terms in cases of still water, leading to the notion of well-balanced schemes or C property [Vázquez-Cendón (1999), Toro (2001), Rogers et al. (2003), Murillo et al. (2007)]. Recently, in order to include properly the effect of source terms in the weak solution, augmented approximate Riemann solvers have been presented [Rosatti et al. (2003), Murillo and García-Navarro (2010)]. In this way, accurate solutions can be computed avoiding the need of imposing case dependent tuning parameters which are used frequently to avoid negative values of water depth and other numerical instabilities that appear when including source terms.

This section presents the system of equations, the formulation of the boundary conditions, and the finite-volume scheme used in RiverFlow2D.

5.2 Assumptions of the Hydrodynamic Model

1. RiverFlow2D uses the Shallow Water Equations resulting from the vertical integration of the Navier-Stokes equation. Therefore, the model does not calculate vertical accelerations, vertical velocities and consequently cannot resolve secondary flows.

2. The bed shear stress is assumed to follow the depth-average velocity directions.

3. The model does not include dispersion nor turbulence terms. Turbulence dissipation and energy loses are accounted for only through the Manning's n term in the momentum equations.

5.3 Unsteady Flow Model

Shallow water flows can be described mathematically by depth averaged mass and momentum conservation equations with all the associated assumptions (Vreugdenhil 1994). That system of partial differential equations will be formulated here in a conservative form as follows:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{S}(\mathbf{U}, x, y) \tag{5.1}$$

where $\mathbf{U} = (h,\ q_x,\ q_y)^T$ is the vector of conserved variables with h representing the water depth, $q_x = uh$ and $q_y = vh$ the unit discharges, with (u, v) the depth averaged components of the velocity vector $\mathbf{u}$ along the (x, y) coordinates respectively. The flux vectors are given by:

$$\mathbf{F} = \left(q_x,\ \frac{q_y^2}{h} + \frac{1}{2}gh^2,\ \frac{q_x q_y}{h}\right)^T, \qquad \mathbf{G} = \left(q_y,\ \frac{q_x q_y}{h},\ \frac{q_y^2}{h} + \frac{1}{2}gh^2\right)^T \tag{5.2}$$

where g is the acceleration of the gravity. The terms $\frac{1}{2}gh^2$ in the fluxes have been obtained after assuming a hydrostatic pressure distribution in every water column, as usually accepted in shallow water models. The source term vector incorporates the effect of pressure force over the bed and the tangential forces generated by the bed stress

$$\mathbf{S} = (0,\ gh(S_{0x} - S_{fx}),\ gh(S_{0y} - S_{fy}))^T \tag{5.3}$$

where the bed slopes of the bottom level z_b are

$$S_{0x} = -\frac{\partial z_b}{\partial x}, \qquad S_{0y} = -\frac{\partial z_b}{\partial y} \tag{5.4}$$

and the bed stress contribution is modeled using the Manning friction law so that:

$$S_{fx} = \frac{n^2 u\sqrt{u^2+v^2}}{h^{4/3}}, \qquad S_{fy} = \frac{n^2 v\sqrt{u^2+v^2}}{h^{4/3}} \tag{5.5}$$

with n the roughness coefficient.

5.4 Finite-Volume Numerical Solution

To introduce the finite-volume scheme, (5.1) is integrated in a volume or grid cell Ω using Gauss theorem:

$$\frac{\partial}{\partial t}\int_\Omega \mathbf{U} d\Omega + \oint_{\partial\Omega} \mathbf{E}\mathbf{n} dl = \int_\Omega \mathbf{S} d\Omega \tag{5.6}$$

where $\mathbf{E} = (\mathbf{F}, \mathbf{G})$ and $\mathbf{n} = (n_x, n_y)$ is the outward unit normal vector to the volume Ω. In order to obtain a numerical solution of system (5.6) the domain is divided into computational cells, Ω_i, using a fixed mesh. Assuming a piecewise representation of the conserved variables (Figure 5.1) and an upwind and unified formulation of fluxes and source terms (Murillo & García-Navarro 2010*b*)

$$\frac{\partial}{\partial t}\int_{\Omega_i} \mathbf{U} d\Omega + \sum_{k=1}^{NE} (\mathbf{E}\mathbf{n} - \bar{\mathbf{S}})_k l_k = 0 \tag{5.7}$$

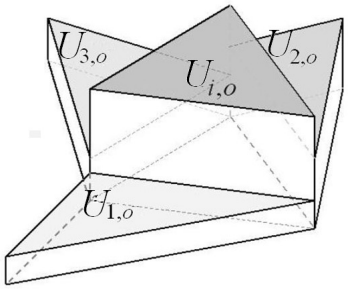


Figure 5.1 – Piecewise uniform representation of the flow variables.

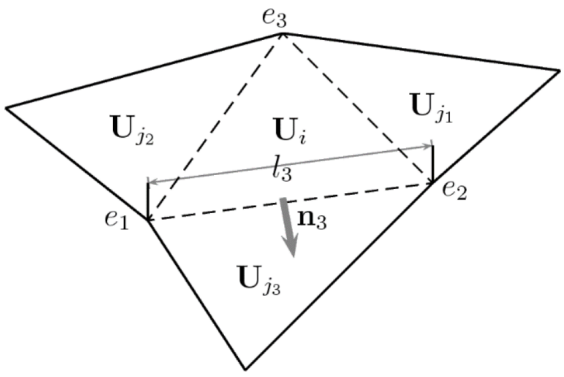


Figure 5.2 – Cell parameters.

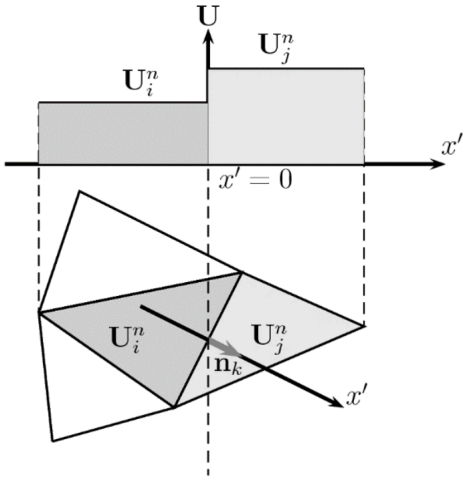


Figure 5.3 – Cell parameters.

The approximate solution can be defined using an approximate Jacobian matrix $\widetilde{\mathbf{J}}_{\mathbf{n},k}$ (Roe 1981) of the non-linear normal flux $\mathbf{E_n}$ and two approximate matrices $\widetilde{\mathbf{P}} = (\widetilde{\mathbf{e}}^1, \widetilde{\mathbf{e}}^2, \widetilde{\mathbf{e}}^3)$, and $\widetilde{\mathbf{P}}^{-1}$, built using the eigenvectors of the Jacobian, that make $\widetilde{\mathbf{J}}_{\mathbf{n},k}$ diagonal

$$\widetilde{\mathbf{P}}_k^{-1}\widetilde{\mathbf{J}}_{\mathbf{n},k}\widetilde{\mathbf{P}}_k = \widetilde{\mathbf{\Lambda}}_k \tag{5.8}$$

with $\widetilde{\mathbf{\Lambda}}_k$ is a diagonal matrix with eigenvalues $\widetilde{\lambda}_k^m$ in the main diagonal

$$\widetilde{\mathbf{\Lambda}}_k = \begin{pmatrix} \widetilde{\lambda}^1 & 0 & 0 \\ 0 & \widetilde{\lambda}^2 & 0 \\ 0 & 0 & \widetilde{\lambda}^3 \end{pmatrix}_k \tag{5.9}$$

Both the difference in vector $\mathbf{U}$ across the grid edge and the the source term are projected onto the matrix eigenvectors basis

$$\delta\mathbf{U}_k = \widetilde{\mathbf{P}}_k\mathbf{A}_k \quad (\bar{\mathbf{S}})_k = \widetilde{\mathbf{P}}_k\mathbf{B} \tag{5.10}$$

where $\mathbf{A}_k = (\alpha^1, \alpha^2, \alpha^3)_k^T$ contains the set of wave strengths and $\mathbf{B} = (\beta^1, \beta^2, \beta^3)_k^T$ contains the source strengths. Details are given in (Murillo & García-Navarro 2010*b*). The complete linearization of all terms in combination with the upwind technique allows to define the numerical flux function $(\mathbf{En} - \bar{\mathbf{S}})_k$ as

$$(\mathbf{En} - \bar{\mathbf{S}})_k = \mathbf{E}_i\mathbf{n}_k + \sum_{m=1}^{3}\left(\widetilde{\lambda}^- \theta\alpha\widetilde{\mathbf{e}}\right)_k^m \tag{5.11}$$

with $\widetilde{\lambda}^- = \frac{1}{2}(\widetilde{\lambda} - |\widetilde{\lambda}|)$ and $\theta_k^m = \left(1 - \frac{\beta}{\widetilde{\lambda}\alpha}\right)_k^m$ that when inserted in (5.71) gives an explicit first order Godunov method (Godunov 1959)

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \sum_{k=1}^{NE}\left[\mathbf{E}_i\mathbf{n}_k + \sum_{m=1}^{3}\left(\widetilde{\lambda}^- \theta\alpha\widetilde{\mathbf{e}}\right)_k^m\right]\frac{l_k}{A_i}\Delta t \tag{5.12}$$

As the quantity $\mathbf{E}_i$ is uniform per cell i and the following geometrical property is given at any cell

$$\sum_{k=1}^{NE}\mathbf{n}_k l_k = 0 \tag{5.13}$$

(5.12) can be rewritten as

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \sum_{k=1}^{NE}\left[\sum_{m=1}^{3}\left(\widetilde{\lambda}^- \theta\alpha\widetilde{\mathbf{e}}\right)_k^m\right]\frac{l_k\Delta t}{A_i} \tag{5.14}$$

The finite-volume method can be written using a compact wave splitting formulation as follows:

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \sum_{k=1}^{NE}\left(\delta\mathbf{M}_{i,k}^-\right)^n\frac{l_k}{A_i}\Delta t \tag{5.15}$$

with

$$\delta\mathbf{M}_{i,k}^- = \sum_{m=1}^{3}\left(\widetilde{\lambda}^- \theta\alpha\widetilde{\mathbf{e}}\right)_k^m \tag{5.16}$$

The use of (5.15) is efficient when dealing with boundary conditions. At the same time it ensures conservation. In (Murillo & García-Navarro 2010*b*) it was demonstrated how for a numerical scheme written in splitting form, the total amount of contributions computed inside the domain at each cell edge, is equal to the balance of fluxes that cross the boundary of the domain, proving exact conservation.

5.4.1 Numerical Optimizations

Once wave propagations in $\delta \mathbf{M}_i^-$ in (5.16) are computed, the first order method can be applied averaging the contributions of the local Riemann Problems (RPs) shaping the contour cell.

The approximate solution is always constructed as a sum of jumps or shocks, even in cases involving rarefactions. One widely reported problem of linearized solvers is the entropy violation in sonic rarefactions (LeVeque 2002, Toro 2001), that produces negative values of depth in the shallow water equations, even in absence of source term. The solution is restored by means of a suitable redefinition of the approximate solution by means of entropy fixes.

The time and space linearization of the source terms in (5.16) can also have negative consequences, as numerical instabilities may arise when approximating their value. Their influence over the approximate RP solutions is the key to construct appropriate fixes that avoid unphysical results. In (Murillo & García-Navarro 2010*b*) it was shown how errors in the integral approaches done over the source terms can be avoided if imposing physically based restrictions over the approximate solution. By simply modifying the source strength coefficients β correct solutions are restored when necessary.

5.4.2 Stability Region

Once numerical fixes are applied the stability region for the homogeneous case can be used to compute the size of the time step. In the 2D framework, considering unstructured meshes, the relevant distance, that will be referred to as χ_i in each cell i must consider the volume of the cell and the length of the shared k edges.

$$\chi_i = \frac{A_i}{\max_{k=1,NE} l_k} \tag{5.17}$$

Considering that each k RP is used to deliver information to a pair of neighboring cells of different size, the distance $\min(A_i, A_j)/l_k$ is relevant. The time step is limited by

$$\Delta t \leq CFL\ \Delta t^{\tilde{\lambda}} \qquad \Delta t^{\tilde{\lambda}} = \frac{\min(\chi_i, \chi_j)}{\max|\tilde{\lambda}^m|} \tag{5.18}$$

with CFL=1/2, as the construction of finite-volume schemes from direct application of one-dimensional fluxes leads to reduced stability ranges (Toro 2001).

RiverFlow2D solution method uses variable time steps. The maximum allowed time-step is controlled by the user-set Courant-Friederich-Lewy (CFL) number that is proportional to the local element size, but also inversely proportional to velocity and depth. Smaller elements lead to smaller time-steps. The maximum theoretical CFL value is 1, but in some runs it may be necessary to reduce this number to lower values.

5.5 Open Boundary Conditions

There are two main boundary condition types that can be used in RiverFlow2D: Open boundaries where flow can enter of leave the modeling area and closed boundaries that are solid no-flow walls

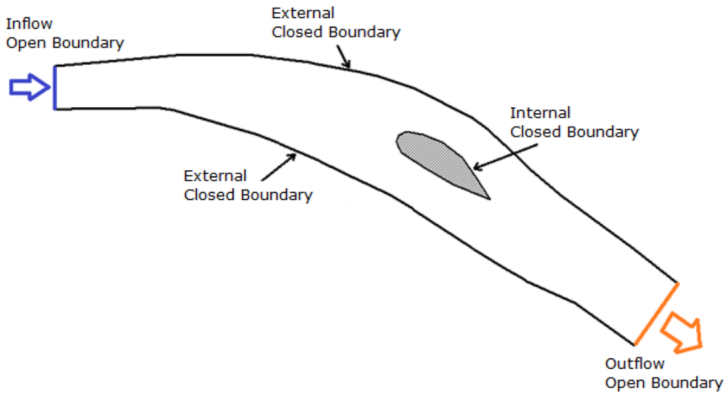


Figure 5.4 – Open and closed boundary conditions.

(see Figure 5.4). There is no restriction on the number of inlet or outlet boundaries. This section describes the open boundary conditions.

If for some reason you need to impose open conditions on boundary segments that are adjacent, do it in such a way that each segment is separated by a gap of one or more cells (see Figure 5.5). Setting two or more open conditions without this separation will lead to incorrect detection of the open boundaries.

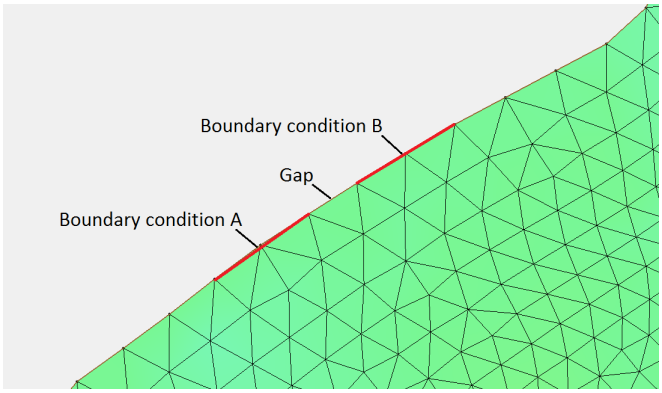


Figure 5.5 – Required gap between adjacent open boundary conditions.

RiverFlow2D allows having any number of inflow and outflow boundaries with various combinations of imposed conditions. Proper use of these conditions is a critical component of a successful RiverFlow2D simulation. Shallow water equation theory indicates that for two-dimensional subcritical flow it is required to provide at least one condition at inflow boundaries and one for outflow boundaries. For supercritical flow all conditions must be imposed on the inflow boundaries and no boundary condition should be imposed at outflow boundaries. The table below helps determining which conditions to use for most applications.

Table 5.1 – Required boundary conditions.

FLOW REGIME AT BOUNDARY	**INFLOW BOUNDARY CONDITION**	**OUTFLOW BOUNDARY CONDITION**
Subcritical	Q or Velocity	Water Surface Elevation
Supercritical	Q and Water Surface Elevation	

It is recommended to have at least one boundary where water surface or stage-discharge is prescribed. Having only discharge and no water surface may result in instabilities due to violation of the theoretical boundary condition requirements of the shallow water equations.

The open boundary condition options are described in the table below.

Table 5.2 – Open Boundary condition types for hydrodynamics.

BC TYPE	DESCRIPTION
1	Imposes Water Surface Elevation. An associated boundary condition file must be provided.
5	Imposes water discharge and water surface elevation.
6	Converts water discharge in velocities perpendicular to boundary line.
9	Imposes single-valued stage-discharge rating table.
10	"Free" inflow or outflow condition. Velocities and water surface elevations are calculated by the model.
11	"Free" outflow condition. Velocities and water surface elevations are calculated by the model, but only outward flow is allowed.
12	Uniform flow outflow condition.

5.5.1 Single Variable Boundary Condition Types (BCTYPE 1 and 6)

When imposing a single variable (water surface elevation, or Q), the user must provide a time series for the corresponding variable. To model steady state the time series should contain constant values for all times. There is no restriction on the time interval used for the time series. When imposing water surface elevation it is important to check that the imposed value is higher than the bed elevation.

5.5.1.1 Water Discharge Converted in Velocities (BCTYPE 6)

In this inflow condition the program calculates the flow area and the average water velocity corresponding to the imposed discharge that can be variable in time. Then, velocity is assigned to each cell assuming perpendicular direction to the boundary line as shown:

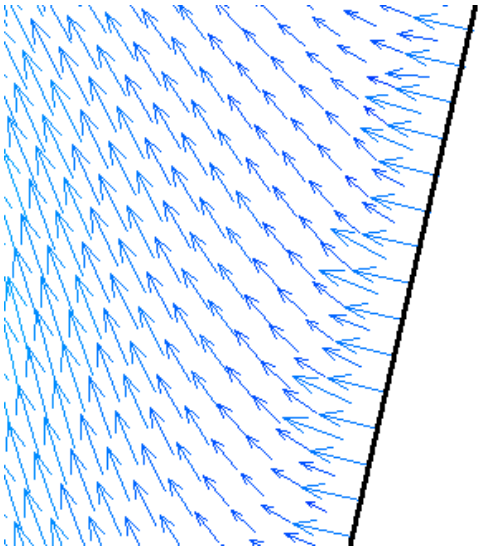


Figure 5.6 – Inflow water discharge imposed as velocities (BCTYPE 6).

5.5.2 Discharge Rating Table (BCTYPE 9)

When using a single valued stage-discharge condition the model first computes the discharge on the boundary then interpolates the corresponding water surface elevation from the rating table and imposes that value for the next time step. If the boundary is dry, it functions as a "free" condition boundary. Water surface elevations are imposed only on wet nodes. This condition requires providing an ASCII file with the table values entries. See section 9.6.1 for details on the file format.

Since these condition may generate wave reflection that can propagate upstream, it is important to locate the downstream boundary on a reach sufficiently far from the area of interest, therefore minimizing artificial backwater effects. Unfortunately, there is no general way to select such place, but numerical experimenting with the actual model will be necessary to achieve a reasonable location.

In most small slope rivers, the stage-discharge relationship is affected by hysteresis. In other words, the stage-discharge curve is looped with higher discharges occurring on the rising limb than on the rescission limb of the hydrograph. This is mainly caused by the depth gradient in the flow direction that changes in sign throughout the hydrograph. In practice, this implies that there can be two possible stages for the same discharge. Loop stage-discharge relationships are not considered in this RiverFlow2D version.

5.5.3 "Free" Open Boundaries (BCTYPE 10, 11)

On free condition boundaries, the model calculates velocities and water surface elevations applying the full equations from the internal elements. In practice this is be equivalent to assuming that derivatives of water surface elevations and velocities are 0. In subcritical flow situations, it is advisable to use these conditions only when there is at least another open boundary where water surface elevation or stage-discharge is imposed. BCTYPE 10 allows water outflow and inflow, while BCTYPE 11 will only allow flow out of the mesh.

5.5.4 Uniform Flow Boundary Condition (BCTYPE 12)

To apply this boundary condition the user provides only the bed slope S_0. The model will use S_0, Manning's n, and discharge to create a rating table. Then for each time-interval, the program will impose the water surface elevation corresponding to the boundary discharge interpolating on the rating table. The rating table is calculated every 0.05 m (0.16 ft.) starting from the lowest bed elevation in the outflow cross section up to 50 m (164 ft.) above the highest bed elevation in the section. If $S_0 = -999$, the model will calculate the average bed slope perpendicular to the boundary line.

5.5.5 Numerical Implementation of Open Boundaries

Many simulation models are based on reliable and conservative numerical schemes. When trying to extend their application to realistic problems involving irregular geometries at boundaries a special care has to be put in preserving the properties of the original scheme. Conservation, in particular, is damaged if boundaries are careless discretized.

In the cells forming the inlet discharge region the flow is characterized by the negative sign of the following scalar product in the k_Γ boundary edges

$$\mathbf{q}_i \cdot \mathbf{n}_{i,k_\Gamma} = (h\mathbf{u})_i \cdot \mathbf{n}_{i,k_\Gamma} < 0 \tag{5.19}$$

and by the state of the flow, defined commonly through the Froude number

$$Fr_i = \frac{\mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma}}{c_i} \tag{5.20}$$

with $c_i = \sqrt{gh_i}$. When the Froude number defined as in (5.20) is greater than one, the flow is supercritical and all the following eigenvalues are negative:

$$\lambda^1 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} + c_i < 0 \qquad \lambda^2 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} < 0 \qquad \lambda^3 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} - c_i < 0 \tag{5.21}$$

therefore the values of h, u, v, and ϕ must be imposed. The water solute concentration ϕ is independent of the eigenvalues, and therefore has to be provided at the inlet region for all flow regimes.

The cells in the outlet discharge region are defined by

$$\mathbf{q}_i \cdot \mathbf{n}_{i,k_\Gamma} = (h\mathbf{u})_i \cdot \mathbf{n}_{i,k_\Gamma} > 0 \tag{5.22}$$

for supercritical flow, all the following eigenvalues are positive:

$$\lambda^1 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} + c_i < 0 \qquad \lambda^2 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} < 0 \qquad \lambda^3 = \mathbf{u}_i \cdot \mathbf{n}_{i,k_\Gamma} - c_i < 0 \tag{5.23}$$

in consequence, no extra information is required.

When in both inlet and outlet discharge region, the flow state is subcritical, the updating information is not complete. The same happens at the cell edges acting like solid walls, that cannot be crossed by the flow. Commonly the extra information provided upstream and downstream are discharge functions. And, on solid boundaries, a zero normal discharge function is defined.

To decide whether we are dealing with a supercritical or a subcritical inlet or outlet is not easy in a 2D mesh. A cell based characterization of the boundary flow regime at the boundaries leads to complicated situations both from the physical and from the numerical point of view. On the other hand, physical or external boundary conditions usually refer to average quantities such as water surface level or total discharge that have to be translated into water depth or velocity at each cell, depending on the practitioner criterion. To handle these situations, a suitable connection between the two-dimensional and the one-dimensional models is required at the open boundaries. The section Froude number is defined once the boundary section has a uniform water level as:

$$Fr_s = \frac{w}{\sqrt{g(S_T/b_T)}} \tag{5.24}$$

being the cross sectional velocity $w = Q/S_T$ and defining the total wet cross section S_T and total breath as:

$$S_T = \sum_{j=1}^{NB} S_j = \sum_{j=1}^{NB} h_j l_j \qquad , b_T = \sum_{j=1}^{NB} l_j \tag{5.25}$$

where NB is the number of wet boundary cells, l_j is the length of each edge conforming the wet boundary and h_j is the water depth at each boundary cell.

5.5.5.1 Inlet discharge boundary

This is one of the boundary conditions that poses most difficulties because a correct and conservative representation of the steady or unsteady incoming flow must be defined and there is not one obvious form to implement it. The total inflow discharge hydrograph $Q = Q(t)$ is the usual function given in flooding simulation, and it is important to analyse the best way to impose it since it involves the full inlet cross section and we are dealing with a 2D discrete representation in computational cells. Different cases may be found.

5.5.5.2 Simple cases

When the inlet cross section is of rectangular shape (Figure 5.7), that is, of flat bottom and limited by vertical walls, the inlet wet cross section is just rectangular.

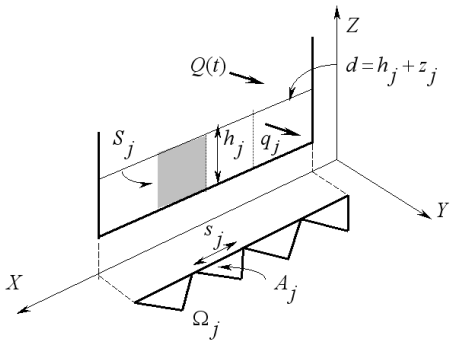


Figure 5.7 – Rectangular inlet cross section.

The total inlet discharge at time t, $Q_I(t)$, can be distributed along the inlet cross section using a constant discharge per unit width, $q_I(m^2 s^{-1})$, that can be calculated as

$$q_j = q_I = \frac{Q_I(t)}{b_T} \tag{5.26}$$

In this simple case, q_I is uniform along the inlet boundary and so is the resulting modulus of the velocity, $w = q_I/h$, with $w = (u^2+v^2)^{1/2}$. It should be noted, that the direction of the entering discharge is not necessarily the same as the direction normal to the inlet boundary. However, this direction is usually chosen as the default information.

5.5.5.3 Complex cases

In real problems of general geometry the inlet cross section may change shape as water level changes (drying/wetting boundary), and so does the number of boundary cells involved (Figure 5.8).

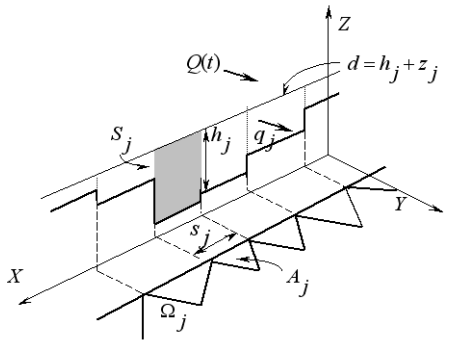


Figure 5.8 – Irregular inlet cross section.

When dealing with inlet sections like that in Figure 5.8, a uniform value of q_I as in (5.26) leads to a completely unrealistic state of faster water at the section borders and slower water at the middle of the cross section. Since the resulting velocities depend on the value of water depth h, higher values will appear in those cells where water depth is smaller.

In order to seek a more appropriate distribution, a uniform modulus of the water velocity w is enforced in the whole inlet boundary cross section. In this case, the unit discharge at each boundary cell j is variable and defined depending both on the total cross section area, S_T, and on the individual cell transverse area, S_j as follows:

$$q_j = Q_I \frac{S_j}{S_T l_j} \tag{5.27}$$

On the other hand, the updating of the water depth values at the inlet cells provided by the numerical scheme leads in the general case to a set of new water depths h_j^{n+1} (Figure 5.9) associated, in general, to different water surface levels d_j $d_j = h_j + z_j$.

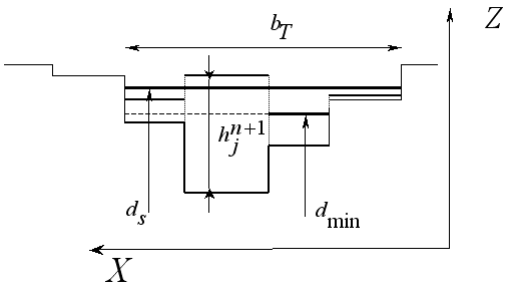


Figure 5.9 – Evaluation of d_{min}.

For our purposes a horizontal water surface level is required in that region, in order to help in the translation between the 2D and the 1D points of view at the open boundary. The value of that uniform cross sectional water level is fixed taking into account mass conservation, that is, conservative redistribution of water volume. The minimum value of the water levels among all the wet cells in the inlet boundary, d_{min}, is found and the water volume V_S stored in the inlet section above d_{min} is evaluated as

$$V_S = \sum_{j=1}^{NB} (d_j - d_{min}) A_j|_{d_j > d_{min}} \tag{5.28}$$

and the wet surface above that level, A_w, is defined:

$$A_w = \sum_{j=1}^{NB} A_j|_{d_j > d_{min}} \tag{5.29}$$

They are used to redistribute the volume over the inlet section, keeping constant the wet section breadth b_T. As Figure 3 shows a new uniform water level at the section, d_S, is given by:

$$d_s = d_{min} + \frac{V_S}{A_w} \tag{5.30}$$

Apart from helping to decide the flow regime at the boundary, the modifications described above make easier the treatment of supercritical inflow conditions. When modeling unsteady river flow, high peaks in the hydrograph can be encountered. If those peaks are not correctly handled from the numerical point of view, they can lead to local and unrealistic supercritical states in the inlet boundary.

In that case of supercritical inlet flow, the specification of all the variables at the inlet boundary cells is required. However, in many practical problems only the discharge hydrograph is available as a function of time, with no data, in general, on the water level distribution or discharge direction at the inlet boundary.

The alternative proposed is, when the inlet Froude number is bigger than 1

$$Fr_s = \frac{w}{\sqrt{g(S_T/b_T)}} > 1 \tag{5.31}$$

to enforce a maximum Froude number, $Fr_{s,max}$, to the inlet flow. For that purpose, keeping the section breadth b_T, a new inlet wet cross section area, S_T^*, is computed from the $Fr_{s,max}$ imposed:

$$S_T^* = (\frac{Q_I^2}{gFr_{s,max}^2/b_T)})^{1/3} \tag{5.32}$$

If S_T^* is greater than S_T, it provides a new water surface level for the inlet section, d^*, also greater than d_s (Figure 5.10). The associated increment in water volume is balanced by means of a reduction in the imposed discharge $Q_I(t)$ in that time step.

Occasionally, both conditions, $Q_I(t)$ and $d(t)$ are known at supercritical inlets. For those cases, imposing both data at the inlet boundary is enough. However, due to the discrete time integration method used, this procedure does not follow the mass conservation criterion. To guarantee that the mass balance is preserved, one of the conditions is imposed, the other must be modified, so that the fluxes calculated in the following step lead to mass conservation. The best solution is to impose directly the global surface water level at the inlet boundary section,$d(t)$, and to adapt the discrete inlet discharge to ensure that the final volume is conserved. The imposed value of d sets an input volume that can be transformed into discharge by means of dividing it by the time step. This value is added to the discharge leading to a correct mass balance.

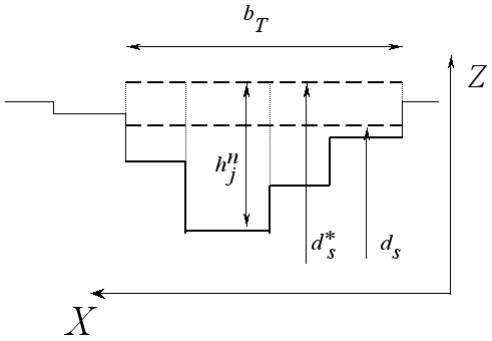


Figure 5.10 – New water level for the inlet section.

When the boundary cell belongs to an open boundary where the inlet flow discharge is the condition imposed and the flow is subcritical, the discharge is computed using (5.27) and imposed in the boundary cell. Moreover, the water level is computed as a results of the contributions from that other cell edges in (5.15) when updating the conserved values in the boundary cell at time level $n+1$ and is carefully redistributed as explained before.

5.5.5.4 Outlet boundaries

The analysis of the flow at the outlet boundary is simpler. For supercritical outflow no external conditions have to be imposed. In RiverFlow2D, a preliminary sweep is performed over the wet outlet boundary cells in order to evaluate the cell Froude number. If a supercritical cell is found, the whole flow at the outflow boundary section is considered supercritical and no external condition has to be enforced. Otherwise, all the cells are in a subcritical state, and receive an analogous treatment to that of the inlet boundary described above. As before, a uniform cross sectional water level is generated and a velocity distribution is set in cases in which a discharge rating curve is the boundary condition to impose.

5.5.6 Closed Boundaries

Closed boundaries are rigid or solid walls that completely block the flow such as river banks or islands. They constitute vertical walls that the flow can never overtop. A very thin viscous sublayer occurs near these boundaries that would require extremely small elements to be appropriately resolved. RiverFlow2D uses slip condition on closed boundaries and the model will set zero normal flow across the boundary, but tangential velocities are allowed. The RiverFlow2D model detects closed boundaries automatically.

This kind of boundary condition does not require any special treatment. As no flow must cross the boundary, the physical condition $\mathbf{u}\cdot\mathbf{n}=0$ is imposed on the cell velocity $\mathbf{u}$ after adding all the wave contributions from the rest of the cell edges, where $\mathbf{n}$ is the solid wall normal (Figure 5.11). In other words, if the boundary is closed, the associated boundary edge k_Γ is a solid wall, with a zero normal velocity component. As there are no contributions from that edge, $\delta\mathbf{M}^-_{i,k_\Gamma}=0$ is set in (5.15) when updating the conserved values in the boundary cell at time level $n+1$.

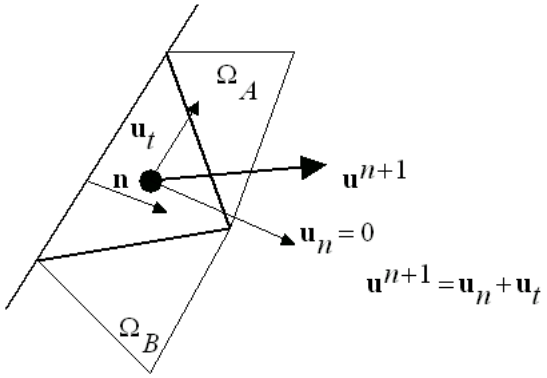


Figure 5.11 – Solid wall condition.

5.6 Dry/Wet Cell Modeling

RiverFlow2D is able to simulate the drying and wetting of the bed. This model capability is important when simulating flood wave progression down an initially dry channel. In this case both the channel bed and floodplain will get inundated. The channel bed can also dry again as the flood wave recedes.

In RiverFlow2D the triangular-cell mesh can cover both dry and wet areas and the model will handle these conditions using two distinct algorithms and depending on the following element classification.

5.6.1 Cell definitions Based on Dry and Wet Conditions

A cell is considered dry if its water depth is less than a fraction of a millimeter. There is not a partially dry cell situation. A cell edge is considered inactive if it separates two dry cells and is excluded from the computation. Otherwise, the cell edge always contributes to the updating of the variables on both sides. The so called wet/dry situation takes place at a cell edge when all the following conditions hold:

- One of the neighbor cells is wet and the other is dry.
- The water level a the wet cell is below the bed level at the dry cell.
- Flow is subcritical.

In that case, the procedure to follow is well described in (Murillo & García-Navarro 2010*b*).

RiverFlow2D drying and wetting algorithm is an adaptation of the the one originally proposed by (Brufau, García-Navarro & Vázquez-Cendón 2004) and later improved by (Murillo, Burguete, Brufau & García-Navarro 2007) and (Murillo & García-Navarro 2010*b*) in the finite-volume context and works as follows:

1. At the beginning of each time-step all cells are classified as wet or dry according to the definition.
2. If a cell is dry and completely surrounded by dry cells, it is removed from the computations and velocity components are set to zero for the ongoing time step.

3. All the internal cell edges are classified as active or inactive according to the definition.
4. Wet/dry cell edge contributions are computed assuming the edge is a solid boundary and the velocities on both sides are set to zero.
5. The rest of the cell edge contributions are computed according to the numerical scheme as described above.
6. Wet cells and dry cells surrounded by at least one wet cell are retained in the computation and solved with the updating scheme using the contributions from the cell edges.

This method generates stable numerical solutions without spurious velocities over dry areas and offers machine accuracy mass conservation errors allowing the use of the classical CFL condition.

5.7 Volume Conservation

The volume conservation or volume balance in the simulation domain can be defined through a discharge contour integral:

$$\Delta M(\Delta t) = \int_t^{t+\delta t} (\mathbf{Q}_I \cdot \mathbf{n}_I - \mathbf{Q}_O \cdot \mathbf{n}_O) dt \tag{5.33}$$

where $\mathbf{Q}_I$ and $\mathbf{Q}_O$ are the total discharge functions at the inlet and at the outlet boundaries respectively, and $\mathbf{n}_I$ and $\mathbf{n}_O$ are the normal vectors to the boundaries. The normal discharge at solid walls is zero. This balance is actually evaluated integrating at the contour cell by cell as follows

$$\Delta M(\Delta t) = \sum_{j=1}^{NB_I} q_{I,j} l_j (\mathbf{n}_I \cdot \mathbf{n}_j) \Delta t - \sum_{m=1}^{NB_O} q_{O,m} l_m (\mathbf{n}_O \cdot \mathbf{n}_m) \Delta t \tag{5.34}$$

where $\mathbf{n}_j$ and $\mathbf{n}_m$ are the directions of the flow in the inlet and in the outlet cells respectively.

The volume variation in the domain of calculation can be only due to

$$\Delta M(\Delta t) \neq 0 \tag{5.35}$$

Therefore, the mass error of the numerical solution is measured by comparing the total amount of water calculated at time $t + \Delta t$

$$Vol(t + \Delta t) = \sum_{i=1}^{NCELLS} h_i^{n+1} S_i \tag{5.36}$$

with the total amount of water existing at time t

$$Vol(t) = \sum_{i=1}^{NCELLS} h_i^{n} S_i \tag{5.37}$$

as follows

$$Error = [Vol(t + \Delta t) - Vol(t)] - \Delta M(\Delta t) \tag{5.38}$$

This is usually expressed in relative terms as follows:

$$Relerror = \frac{[Vol(t + \Delta t) - Vol(t)] - \Delta M(\Delta t)}{Vol(t) + \Delta M(\Delta t)} \tag{5.39}$$

5.8 Manning's *n* roughness Coefficients

The Manning's n usually estimated to determine head losses in channel and river flow is a global measure that accounts not only for the effects of bed roughness, but also for internal friction and variations in shape and size of the channel cross section, obstructions, river meandering (Ven Te Chow, 1959). Therefore, estimations of Manning's n applicable for 1D models should be adjusted, because 2D model equations consider two-dimensional momentum exchange within the cross section that is only lumped in the 1D simplification. Several researchers have found in practical applications of 2D models that the n values required can be 30% lower than those normally used for 1D models on the same river reach (Belleudy, 2000). However, 2D models do not account for lateral friction, therefore the final selection of Manning's n coefficients should be the outcome of a calibration process where the model results are adjusted to measured data.

5.9 Sediment Transport Model: RiverFlow2D ST Module

The science of sediment transport deals with the interrelationship between flowing water and sediment particles. Despite of having been studied since the 1950s and being widely employed in engineering practice, (Nielsen 1992), (Julien 1998), the sediment transport modeling remains at present one of the most active topic in the field of hydraulic research. Although numerical modeling of free surface flows with bed load transport over erodible bed in realistic situations involves transient flow and movable flow boundaries, the conventional and pioneer methods for performing morphodynamic simulations in coastal areas and rivers decouple the hydrodynamic and the erosion and deposition components (Cunge, Holly & Vervey 1980, De Vriend, Zyserman, Nicholson, Roelvink, Péchon & Southgate 1993, Abderrezzak & Paquier 2011). Ignoring unsteady hydrodynamic effects means that the time scales of the morphodynamics changes are smaller in comparison with the hydrodynamic ones. Assuming this hypothesis, only a quasi-steady process of slowly varying bed-load can be reasonably modeled, so that rapidly varying flows containing shocks or discontinuities remain excluded. For this reason RiverFlow2D allows accurate simulation ranging from slow-evolving events to abrupt river bed changes.

5.9.1 Model Equations

The relevant formulation of the model derives from the depth averaged equations expressing water volume conservation, solute volume conservation and water momentum conservation. That system of partial differential equations is formulated here in coupled form as follows

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{S}(\mathbf{U}) + \mathbf{R}(\mathbf{U}) \tag{5.40}$$

where

$$\mathbf{U} = \left(\; h, \quad q_x, \quad q_y, \quad h\phi_1, \quad \ldots, \quad h\phi_{N_p}, \quad \sum_{p=1}^{N_p} \left[z_p(1-p_p) \right] \; \right)^T \tag{5.41}$$

are the conserved variables with h representing the water depth, $q_x = hu$ and $q_y = hv$ the unit discharges, with (u, v) the depth averaged components of the velocity vector $\mathbf{u}$ along the x and y coordinates respectively and ϕ_p, with $p = 1, ..., N_p$ representing the scalar depth-averaged concentration of the N_p different sediments transported in suspension. The term z is the bed level

and p_p is the bed porosity of each p bed sediment. The sum of $\sum_{p=1}^{N_p}[z_p(1-p_p)]$ is motivated for the possibility of a heterogeneous soil, where different fractions of material may coexist.

On the other hand the fluxes are given by

$$\begin{aligned}\mathbf{F} &= \left(\; q_x, \quad \tfrac{q_x^2}{h} + \tfrac{1}{2}gh^2 \quad q_x q_y/h, \quad q_x\phi_1, \quad \ldots \quad q_x\phi_{N_p}, \quad q_{sx} \; \right)^T \\ \mathbf{G} &= \left(\; q_y, \quad q_x q_y/h, \quad \tfrac{q_y^2}{h} + \tfrac{1}{2}gh^2 \quad q_y\phi_1, \quad \ldots, \quad q_y\phi_{N_p}, \quad q_{sy} \; \right)^T\end{aligned} \tag{5.42}$$

where g is the acceleration of the gravity and q_{sx} and q_{sy} are the bed load transport terms computed by means of an empirical law. The source terms of the system are split in three kind of terms. The term $\mathbf{S}$ is defined as

$$\mathbf{S} = \left(0, \; \frac{p_{b,x}}{\rho_w} - \frac{\tau_{b,x}}{\rho_w}, \; \frac{p_{b,y}}{\rho_w} - \frac{\tau_{b,y}}{\rho_w}, 0, \ldots, 0, 0\right)^T \tag{5.43}$$

with $p_{b,x}, p_{b,y}$ and $\tau_{b,x}, \tau_{b,y}$ are the pressure force along the bottom and the bed shear stress in the x and y direction respectively, with ρ_w the density of water. The former can be formulated in terms of the bed slopes of the bottom level z

$$\frac{p_{b,x}}{\rho_w} = -gh\frac{\partial z}{\partial x} = ghS_{0x}, \qquad \frac{p_{b,y}}{\rho_w} = -gh\frac{\partial z}{\partial y} = ghS_{0y} \tag{5.44}$$

and the friction losses are written in terms of the Manning's roughness coefficient n

$$\frac{\tau_{b,x}}{\rho_w} = ghS_{fx} \quad S_{fx} = \frac{n^2 u\sqrt{u^2+v^2}}{h^{4/3}}, \qquad \frac{\tau_{b,y}}{\rho_w} = ghS_{fy} \quad S_{fy} = \frac{n^2 v\sqrt{u^2+v^2}}{h^{4/3}} \tag{5.45}$$

The reaction source terms $\mathbf{R}$, having a volumetric character, is defined as

$$\mathbf{R} = \left(0, 0, 0, R_1, \ldots, R_{N_p}, -\sum_{p=1}^{N_p} R_p\right)^T \tag{5.46}$$

and is evaluated as

$$R_p = \omega_{s_p}(\phi_{*_p} - \phi_p) \tag{5.47}$$

where the term ϕ_{*_p} is employed for defining the equilibrium concentration, which is obtained through a solid transport discharge law. The term ϕ_p contains the information about the suspension sediment quantity which is transported. Both concentrations take in consideration the settling velocity, ω_{s_p}.

5.9.2 Sediment Transport Laws

Two different ways of sediment transport govern the dynamics of the mobile bed considered in RiverFlow2D: the suspended load and the bed load. Both of them may coexist or one may be dominant.

As each sediment transport law is derived from different laboratory and field data sets, a calibration parameter in the form of a correction factor is considered in order to adjust the numerical results.

5.9.2.1 Bed Load

When bed load is the dominant sediment transport mechanism and the influence of the suspended load is negligible, system (5.40) reduces to

Mass conservation

$$\frac{\partial(h)}{\partial t} + \frac{\partial(hu)}{\partial x} + \frac{\partial(hv)}{\partial y} = 0 \tag{5.48}$$

Momentum conservation in x direction

$$\frac{\partial(hu)}{\partial t} + \frac{\partial[hu^2 + (1/2)gh^2]}{\partial x} + \frac{\partial(huv)}{\partial y} = \frac{p_{bx}}{\rho_w} - \frac{\tau_{bx}}{\rho_w} \tag{5.49}$$

Momentum conservation in y direction

$$\frac{\partial(hu)}{\partial t} + \frac{\partial(huv)}{\partial x} + \frac{\partial[hv^2 + (1/2)gh^2]}{\partial y} = \frac{p_{by}}{\rho_w} - \frac{\tau_{by}}{\rho_w} \tag{5.50}$$

Bed elevation changes

$$\frac{\partial \sum_{p=1}^{N_p} [z_p(1-p_p)]}{\partial t} + \frac{\partial q_{sx}}{\partial x} + \frac{\partial q_{sy}}{\partial y} = 0 \tag{5.51}$$

Where N_pm and the terms of q_{sx} and q_{sy} are the volumetric sediment fluxes per unit width can be determined through several deterministic laws or sediment transport formulas and include the total transport from all sediment fractions.

The modulus of the sediment transport rate, q_s, is defined as:

$$|q_s| = \sqrt{q_{sx}^2 + q_{sy}^2} \tag{5.52}$$

The bed load transport is often represented by the following dimensionless parameter,

$$\Phi = \frac{|q_s|}{\sqrt{g(s-1)d_{50}^3}} \tag{5.53}$$

where s is the solid material vs water density ratio and d_{50} is the median diameter.

Table 5.3 collects the formulas that are implemented in RiverFlow2D, where d_{90}, d_{50} and d_{30} are the grain diameter for which 90%, 50% and 30% of the weight of a non-uniform sample is finer respectively, ρ_s is the sediment density, θ_c is the critical Shields parameter and θ_c^s is the critical Shield parameter as expressed by Smart (1984).

Table 5.3 – Summary of bed load transport formulas.

FORMULA	Φ	INPUT PARAMETERS
Meyer-Peter & Mueller (1948)	$8\,(\theta-\theta_c)^{3/2}$	d_{50},θ_c,ρ_s
Ashida Michiue (1972)	$17\,(\theta-\theta_c)\left(\sqrt{\theta}-\sqrt{\theta_c}\right)$	d_{50}, θ_c, ρ_s
Engelund and Fredse (1976)	$18.74\,(\theta-\theta_c)\left(\sqrt{\theta}-0.7\sqrt{\theta_c}\right)$	d_{50}, θ_c,ρ_s
Fernandez-Luque and van Beek (1976)	$5.7\,(\theta-\theta_c)^{3/2}$	d_{50}, θ_c, ρ_s
Parker fit to Einstein (1979)	$11.2\,(1-\theta/\theta_c)^{9/2}$	d_{50},θ_c,ρ_s
Smart (1984)	$4\,(d_{90}/d_{30})^{0.2}\,S_0^{0.6}C\theta^{1/2}\,(\theta-\theta_c^s)$	d_{30},d_{50}, d_{90}, θ_c, θ_c^S, ρ_s
Nielsen (1992)	$12\theta^{1/2}\,(\theta-\theta_c)$	d_{50}, θ_c,ρ_s
Wong	$4.93\,(\theta-\theta_c)^{1.6}$	d_{50}, θ_c, ρ_s
Wong	$3.97\,(\theta-\theta_c)^{3/2}$	d_{50}, θ_c, ρ_s
Camenen-Larson	$12\theta^{3/2}exp\,(-4.5\theta_c/\theta)$	d_{50}, θ_c, ρ_s

The sum of $\sum_{p=1}^{N_p}[z_p(1-p_p)]$ in (5.51) can consider heterogeneous soil, where different fractions of material may coexist. In order to take into account this heterogeneity, every sediment transport discharge (q_{sx}, q_{sy}) associated to each sediment size is multiplied by its corresponding soil fraction.

For every sediment particle size it is necessary to include the fraction of the material.

5.9.3 Boundary Conditions for the Bed Load Sediment Model

When using the bed load sediment model, the user may impose inflow boundary condition BCType = 6 and provide only the water discharge. In that case the model will determine the inflow sediment discharge based on the selected sediment transport formula assuming equilibrium conditions.

Alternatively you may provide the sediment discharge for each sediment fraction at all inflow boundaries using BCType = 26. This data should be included in the same file that sets the discharge time series at each inlet. For instance, if you select two sediment fractions BCType = 26 where the inflow is set as discharge a typical file would be like this one:

3			
0.0	1000.	0.16	0.23
2.0	7000.	1.16	2.24
100.0	7000.	1.16	2.24

In file there are 3 times: 0, 2 and 100 hours. The first column represents the time in hours. The second column indicates the water discharge in m^3/s or ft^3/s. The last two columns have the sediment discharge for each given fraction in in m^3/s or ft^3/s.

5.9.4 Suspended Load

When the suspension load plays the key role in the mobilization of the bed, the term of bed load can be omitted and system (5.40) is written as

Mass conservation

$$\frac{\partial(h)}{\partial t}+\frac{\partial(hu)}{\partial x}+\frac{\partial(hv)}{\partial y}=0 \tag{5.54}$$

Momentum conservation in x direction

$$\frac{\partial(hu)}{\partial t}+\frac{\partial[hu^2+(1/2)gh^2]}{\partial x}+\frac{\partial(huv)}{\partial y}=\frac{p_{bx}}{\rho_w}-\frac{\tau_{bx}}{\rho_w} \tag{5.55}$$

Momentum conservation in y direction

$$\frac{\partial(hu)}{\partial t}+\frac{\partial(huv)}{\partial x}+\frac{\partial[hv^2+(1/2)gh^2]}{\partial y}=\frac{p_{by}}{\rho_w}-\frac{\tau_{by}}{\rho_w} \tag{5.56}$$

Sediment mass of the fluid layer for the suspended sediment p

$$\frac{\partial(h\phi_p)}{\partial t}+\frac{\partial(hu\phi_p)}{\partial x}+\frac{\partial(hv\phi_p)}{\partial x}=\omega_{s_p}(\phi_{*_p}-\phi_p) \tag{5.57}$$

The term ϕ_{*_p} defines the equilibrium volume concentration, which is obtained through a solid transport discharge law. The term ϕ_p contains the information about the suspension sediment quantity which is transported. Both concentrations take in consideration the presence of the

settling velocity, ω_{s_p}. The settling velocity can be computed through several laws as a function of the sediment density and the grain diameter.

Table 5.4 collects the formulas considered in RiverFlow2D. It is worth noticing that these laws have been derived for one single particle and under steady situations. Hence, in case of considering a transient situation with a sediment concentration in the water column, the influence of the nearby particles must be taken into consideration, which can strongly reduce the fall velocity in comparison with clear water. This effect, known as hindered settling (Wu 2008, Baldock, Tomkins, Nielsen & Hughes 2004), can be determined by (Sha 1965), computing a new settling velocity ω_{sm}

$$\omega_{sm} = \left(1 - \frac{\phi}{2\sqrt{d_{50}}}\right)^n \omega_s \tag{5.58}$$

where ω_s is the settling velocity of a single particle in clear water, ϕ is the volumetric concentration, d_{50} is the mean diameter and n is an empirical exponent with a usual value equal to 3.

Table 5.4 – Formulas to calculate sediment settling velocity ω_s.

FORMULA	ω_s	OBSERVATIONS
Rubey (1933)	$\omega_s = F\sqrt{\left(\frac{\rho_s}{\rho_w} - 1\right) g d_s}$	If $d_s > 1$ mm, $F = 0.79$, otherwise $F = \left(\frac{2}{3} + \frac{36\nu^2}{g d_s^3 (\rho_s/\rho_w - 1)}\right)^{0.5} - \left(\frac{36\nu^2}{g d_s^3 (\rho_s/\rho_w - 1)}\right)^{0.5}$
Zhang (1961)	$\omega_s = \sqrt{\left(13.95\frac{\nu}{d_s}\right)^2 + 1.09\left(\frac{\rho_s}{\rho_w} - 1\right) g d_s} - 13.95\frac{\nu}{d_s}$	Valid for a wide range of d_s
Zanke (1977)	$\omega_s = 10\frac{\nu}{d_s}\left[\left(1 + 0.01\left(\frac{\rho_s}{\rho_w} - 1\right)\frac{g d_s^3}{\nu^2}\right)^{0.5} - 1\right]$	
Van Rijn (1984)	$\omega_s = \left(\frac{1}{18}\frac{\rho_s - \rho_w}{\rho_w} g \frac{d_s^2}{\nu}\right)$	$d_s < 0.1$ mm
Raudkivi (1990)	$\omega_s = \left(\frac{(2/3)(\rho_s/\rho_w - 1) g d_s^2}{\nu} - 32\right)\frac{\nu}{1.2 d_s}$	$d_s > 1$ mm
Julien (1995)	$\omega_s = \left(\frac{(2/3)(\rho_s/\rho_w - 1) g d_s^2}{\nu} - 24\right)\frac{\nu}{1.5 d_s}$	
Cheng(1997)	$\omega_s = \frac{\nu}{d_s}\left(\sqrt{25 + 1.2 D_*^2} - 5\right)^{1.5}$	$D_* = d_s\left(\left(\rho_s/\rho_w - 1\right) g/\nu^2\right)^{1/3}$
Jiménez-Madsen (2003)	$\omega_s = (C_1 + C_2 S_*)\sqrt{(s-1) g d_s}$	C_1, C_2 and S_* as in Jiménez-Madsen
Wu-Wang(2006)	$\omega_s = \frac{M\nu}{N d_s}\left(\sqrt{\frac{1}{4}\left(\frac{4N}{3M^2} D_*^3\right)^{1/n}} - 0.5\right)^n$	M, N and n as in Wu and $D_* = d_s\left(\left(\rho_s/\rho_w - 1\right) g/\nu^2\right)^{1/3}$

5.9.4.1 Suspended Load Formulas

As it has been mentioned before, the term ϕ_{*_p} which appears in (5.57) is employed for defining the equilibrium concentration, which is obtained through a solid transport discharge law as follows,

$$\phi_{*p} = \begin{cases} 0 & \theta < \theta_c \\ \dfrac{q_{susp_p}}{hu} & \theta \geq \theta_c \end{cases} \tag{5.59}$$

where the subscript p makes reference to the sediment class p. The formulas employed for computing the solid transport discharge are depicted below.

- Bagnold (Bagnold 1973) considered that the shear stress is proportional to the flow velocity and it was established the following formula,

$$q_{susp_p} = 0.01 \, \frac{\rho_s - \rho_w}{\rho_s} \, \frac{\tau_b |u|^2}{g \, \omega_s} \tag{5.60}$$

being τ_b the shear stress generated at the bottom by the bed roughness which is taken into account through the Manning's coefficient, n

$$\tau_b = \frac{g n^2 |u|^2}{h^{1/3}} \tag{5.61}$$

- Van Rijn (Van Rijn 1984*c*) proposed calculating the suspended load integrating the sediment flux within the layer where the suspension plays a key role, i.e. between the term h_s and h,

$$q_{susp_p} = \int_{h_s}^{h} \phi_m u dh \tag{5.62}$$

where, q_{susp} is expressed by volume per unit time, and the terms of ϕ_m and h_s were defined as follows

$$\phi_m = \phi_s \left(\frac{\frac{h}{h'} - 1}{\frac{h}{h_s} - 1} \right)^{\omega_s / (kU_*)} \tag{5.63}$$

with

$$\phi_s = \frac{0.117}{d_s} \left[\frac{\nu^2}{(s-1)g} \right]^{1/3} \left(\frac{\theta}{\theta_c} - 1 \right) \tag{5.64}$$

$$h_s = 0.3 d_s \left[d_s \left(\frac{(s-1)g}{\nu^2} \right)^{1/3} \right]^{0.7} \sqrt{\frac{\theta}{\theta_c} - 1} \tag{5.65}$$

where θ is the non-dimensional shear stress, calculated using (5.66) and (5.67) and related to n, the Manning roughness coefficient, θ_c is the Shields's parameter, d_s is the sediment diameter, s is the rate between densities, $s = \frac{\rho_s}{\rho_w}$ and ν is the kinematic viscosity.

$$\theta = \frac{\tau_b}{\rho_w (s-1) g d_s} \tag{5.66}$$

with

$$\tau_b = \frac{g n^2 |u|^2}{h^{1/3}} \tag{5.67}$$

In equation 5.63, U_* is the friction velocity defined as $U_* = \sqrt{\tau_b}$ and κ is the Von-Karman constant $\kappa \approx 0.41$.

```
3
0          1000.      0.001      0.002
2          67000.     0.001      0.002
100        67000.     0.001      0.002
```

5.9.5 Boundary Conditions for the Suspended Sediment Model

When using the suspended sediment model, the user must impose volume concentration for each sediment fraction at all inflow boundaries. This data should be included in the same file that sets the discharge time series at each inlet. For instance, if you select two sediment fractions and BCType = 6 where the inflow is set as discharge, a typical file should be like this:

Where the first line indicates that there are 3 times. The first column corresponds to the time in hours: 0, 2 and 100 hours in this example. The second column is the water discharge in m^3/s or ft^3/s. The third and fourth columns indicate the sediment volume concentrations for each of the two given fractions respectively.

5.9.6 Geomorphological Collapse

When managing transient geomorphological flows in realistic cases, the geotechnical equilibrium bank characteristics can be ruined, leading to dramatic channel metamorphosis. This effect needs to be modeled to reproduce correctly bed geometry evolution in combination with flow action. In RiverFlow2D, the effect of the geomorphological collapse is introduced in the simulation by a simple mass conservative mechanism of slope sliding failure, assuming that the angle of repose of submerged material of the bed can be approximated by the friction angle. The failure mechanism is applied by comparison between the bed slope in each cell edge and the angle of repose of saturated bed material, (Murillo & García-Navarro 2010*a*).

5.9.7 Sediment Transport Finite-Volume Numerical Solution

The system of equations (5.40) is integrated in a volume or grid cell Ω using Gauss theorem:

$$\frac{\partial}{\partial t}\int_{\Omega}\mathbf{U}d\Omega+\oint_{\partial\Omega}\mathbf{En}dl=\int_{\Omega}\mathbf{S}d\Omega+\int_{\Omega}\mathbf{R}d\Omega \tag{5.68}$$

where $\mathbf{n}=(n_x,n_y)$ is the outward unit normal vector to the volume Ω.

In order to obtain a numerical solution of system (5.40) we divide the domain in computational cells, Ω_i, using a mesh fixed in time, and (5.68) is applied to each cell

$$\frac{\partial}{\partial t}\int_{\Omega_i}\mathbf{U}d\Omega+\sum_{k=1}^{NE}\int_{e_k}^{e_{k+1}}(\mathbf{En})_k^{\downarrow}dl_k=\int_{\Omega_i}\mathbf{S}d\Omega+\int_{\Omega_i}\mathbf{R}d\Omega \tag{5.69}$$

with $(\mathbf{En})_k^{\downarrow}$ the value of the interface flux function through the edge k to be defined, $\mathbf{n}_k=(n_x,n_y)$ is the outward unit normal vector to the cell edge k, and NE is the number of edges in cell i. A sketch of the fluxes is showed in Figure 5.12.

Assuming a piecewise representation of the conserved variables

$$\mathbf{U}_i^n=\frac{1}{A_i}\int_{\Omega_i}\mathbf{U}(x,y,t^n)d\Omega \tag{5.70}$$

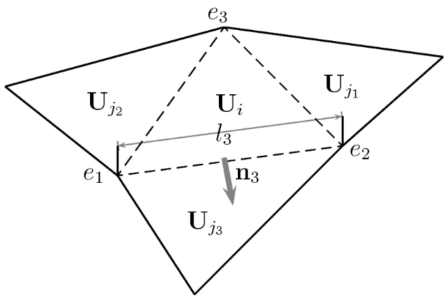


Figure 5.12 – Cell parameters.

Equation (5.69) is written as

$$\frac{\partial}{\partial t}\int_{\Omega_i}\mathbf{U}d\Omega+\sum_{k=1}^{NE}(\mathbf{En})_k^{\downarrow}l_k=\int_{\Omega_i}\mathbf{S}d\Omega+\int_{\Omega_i}\mathbf{R}d\Omega \tag{5.71}$$

where l_k is the corresponding edge length. System (5.71) is solved following the theory of Roe's Riemann solver and using the upwind discretization [(Murillo & García-Navarro 2010*a*, Juez, Murillo & P. 2014)].

5.9.8 Entering Data for the Sediment Transport Model

To enter data for a sediment transport simulation use the *Sediment Transport* pannel in the RiverFlow2D Data Input Program.

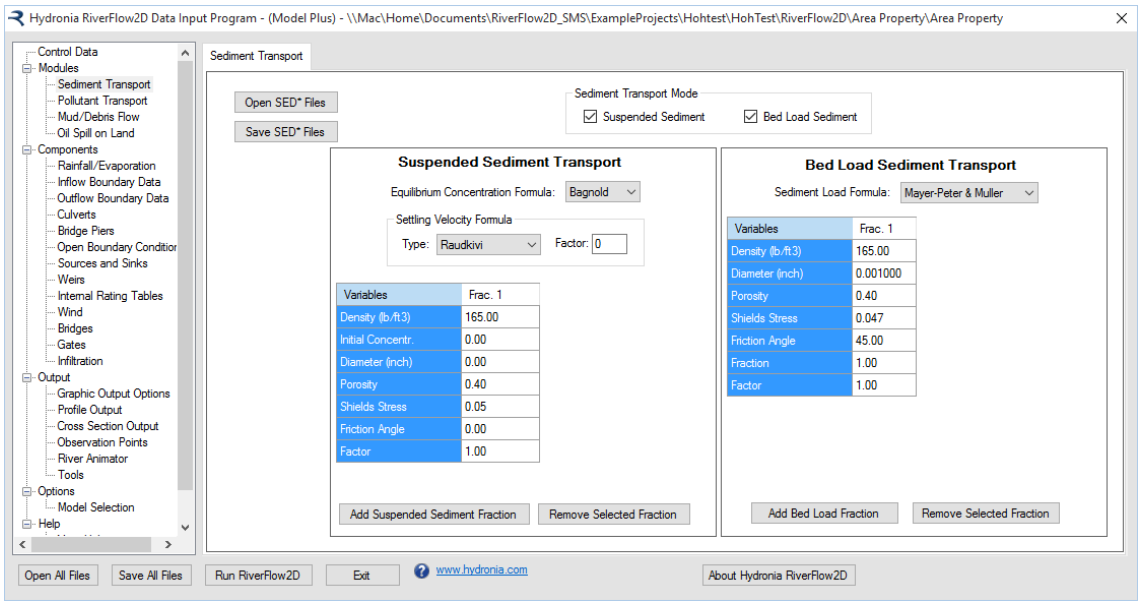


Figure 5.13 – RiverFlow2D Sediment Transport panel.

5.9.9 Assumptions of the Sediment Transport Model

The main assumptions involved in the present version of RiverFlow2D model are:

- When using the bed load option, the inflow sediment transport rate is equal to the transport capacity under equilibrium conditions.
- Each Sediment particle size is considered using a single characteristic diameter (D_{50}) for all formulas except for that of Smart (1984) that considers three sizes (D_{30}, D_{50} and D_{90}).
- The model does not account for bed armoring.
- When assigning maximum erosion bed elevation, the model blocks further erosion when the bed elevation reaches the given limit.

Since the formulations for the bed load discharge, the suspended load discharge and the settling velocity are based on deterministic laws supported by experimentation, tuning parameter factors have been considered for each one. Hence, the model makes possible the calibration of the computed sediment transport for each particular situation.

5.10 Mud and Debris Flow Model: RiverFlow2D MD module

Mud/debris floods are unsteady flow phenomena in which the flow changes rapidly, and the properties of the fluids, typically non-Newtonian, are those of a hyperconcentrated mixture water and sediment and include stop and go mechanisms. The global resistance behavior of the mud/debris flow depends on the relative importance of the shear stresses arising from different sources that, apart from turbulent shear stress at the river boundary, include viscous stress, yield stress, dispersive stress and inelastic collisions of solid particles within the fluid mixture Naef et al. 2006.

The mathematical model adopted in RiverFlow2D is based on that of Murillo & Garcia-Navarro (2012), and regards bed and internal friction for free-surface flows ranging from clear water to hyperconcentrated mixtures of sediments.

5.10.1 Assumptions of the Mud and Debris Flow Model

The model involves the following assumptions:

- The flow is confined to a layer which is thin compared to the horizontal scale of interest;
- The flow is governed by equation (5.1) with the friction terms evaluated as explained below; The mixture of water and sediments is described by using the continuum approach and assuming the same velocity for the liquid and for the solid phase;
- The river bed does not erode;
- The fluid is assumed to be an homogeneous single-phase mix of water and sediment and has constant properties: e.g. density, yield stress, etc.

5.10.2 Rheological Formulation

The single-phase rheological formulation in RiverFlow2D MD accounts for different friction terms that represent a variety of hyperconcentrated non-Newtonian fluids (see Figure 5.14). The friction term in equation (5.1) is depth-averaged. Hence, the equations that describe the tangential forces

generated by the stresses can be lumped into the same mathematical formula despite of having a different nature. In this way, several shear stresses can be considered to cover a wide range of mud and debris flow conditions, including:

- Turbulent stress τ_t
- Dispersive stress τ_d
- Coulomb-type frictional stress τ_f
- Yield stress τ_y and
- Viscous stress τ_μ

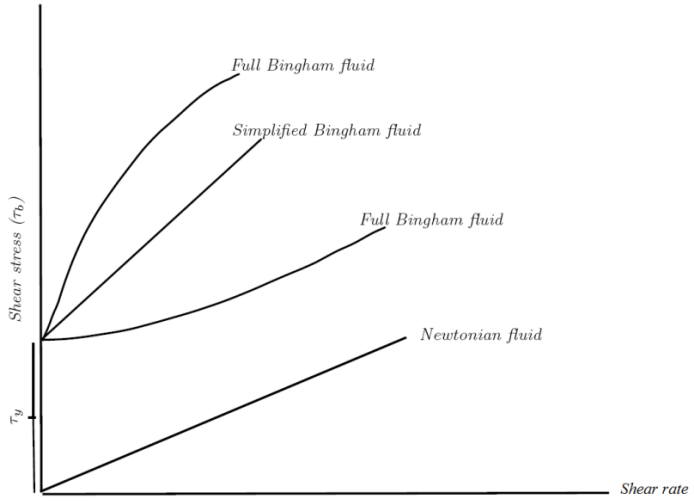


Figure 5.14 – Rheological diagram of some common non-Newtonian formulations.

The equations that describe the tangential forces generated by the stresses involve turbulent stress τ_t, dispersive stress τ_d, Coulomb-type frictional stress τ_f, yield stress τ_y, and viscous stress τ_μ. Not all stresses act simultaneously in the same location of the water column. However, since the RiverFlow2D conceptual model is depth-averaged, all terms may actually coexist and are mathematically lumped in the same formula. In the case of a viscous Newtonian fluid, the bed stress τ_b is expressed in function of the depth averaged velocity u and depth h as

$$\tau_b = 3\tau_\mu \quad \text{with} \quad \tau_\mu = u/h \tag{5.72}$$

A Bingham fluid will not flow until a certain threshold value of the stress, the yield stress τ_y, is reached. Once flowing, the movement is characterized by a plastic viscosity of the mixture. The bed stress τ_b can be obtained from the following cubic formula

$$2\tau_b^3 - 3(\tau_y + 2\tau_\mu)\tau_b^2 + \tau_y^3 = 0 \tag{5.73}$$

and in cases of stress ratio τ_y/τ_b smaller than 0.5 can be reduced to

$$\tau_b = \frac{3}{2}\tau_y + 3\tau_\mu \tag{5.74}$$

If only the Coulomb-type frictional stress is considered, the bed shear stress at the bed is given by

$$\tau_b = \tau_f = g\rho h \cos\theta \tan\theta_b \tag{5.75}$$

where θ is the bed slope angle and θ_b is the friction angle of the solid material. Also, if τ_b only includes dispersive and turbulent effects near the bed, a friction coefficient c_f can be used:

$$\tau_b = \tau_t = \rho c_f u^2 \tag{5.76}$$

In general the total shear stress τ_b can be written as a combination of stress components:

$$\tau_b = f(\tau_t, \tau_d, \tau_y, \tau_\mu, \tau_f) \tag{5.77}$$

Table 5.5 shows the rheological friction laws available in RiverFlow2D MD. The Bingham-type relations include the standard cubic formulation

$$f_1(\tau_0, \tau_1) = 2\tau_b^3 - 3(\tau_y + 2\tau_\mu)\tau_b^2 + \tau_y^3 = 0 \tag{5.78}$$

The *Quadratic* formulation in Table 5.5 includes turbulent-collisional, yield stress and laminar flow resistance terms in a quadratic rheologic law, where a standard value of $\kappa = 24$ is assumed.

Table 5.5 – Rheological flow resistance formulations used in RiverFlow2D MD.

FORMULATION	FLOW RESISTANCE TERM	INPUT PARAMETERS
Turbulent	$\tau_b = \tau_t$	Manning's n
Full Bingham	$\tau_b = \tau_0$ with $2\tau_0^3 - 3(\tau_y + 2\mu q/h^2)\tau_0^2 + \tau_y^3 = 0$	Bingham viscosity, Yield Stress, Density
Simplified Bingham	$\tau_b = 1.5\tau_y + 3\tau_\mu$, $\tau_\mu = \mu q/h^2$	Bingham viscosity, Yield Stress, Density
Turbulent and Coulomb	$\tau_b = \tau_t + \tau_f$	Manning's n, Friction angle, Density
Turbulent and Yield	$\tau_b = \tau_t + \tau_y$	Manning's n, Yield Stress
Turbulent, Coulomb and Yield	$\tau_b = \tau_t + min\left(\tau_y, \tau_f\right)$	Manning's n, Yield Stress, Friction angle, Density
Quadratic	$\tau_b = \tau_t + \tau_y + \frac{\kappa}{8}\tau_\mu$	Manning's n, Yield Stress, Friction angle, Density
Granular	$\tau_b = \tau_f = g\rho h \cos\theta \tan\theta_b$	Manning's $n = 0$, Friction angle

5.10.3 Entering Data for the Mud and Debris Flow Model

To enter data for a mud and debris flow simulation use the *Mud/Debris Flow* panel in the RiverFlow2D Data Input Program. Also make sure that the *Mud/Debris Flow* check box is active in the *Control Data* panel.

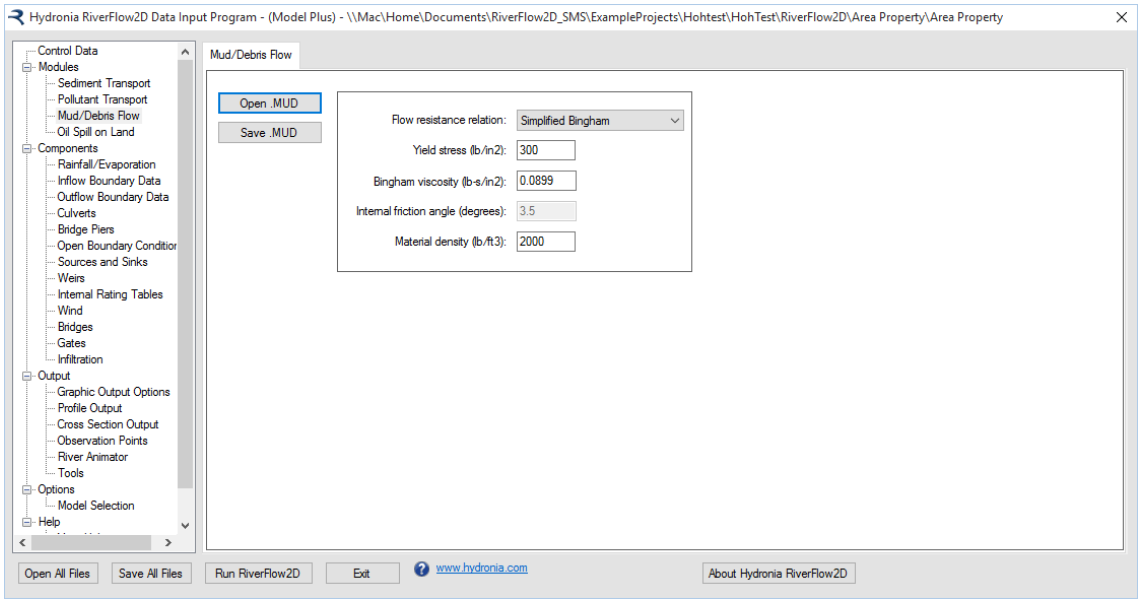


Figure 5.15 – RiverFlow2D Mud/Debris Flow panel.

Depending on the selected flow resistance relation, the user will need to enter different parameters. For example the Full Bingham formulation requires yield stress, Bingham viscosity and Material density, etc..

5.11 Pollutant Transport Model: RiverFlow2D PL Module

The study of solute transport phenomena and river mixing has become a great concern in hydraulic and environmental problems. The RiverFlow2D Pollutant Transport Model provides a tool to calculate concentrations of multiple pollutants in a variety of riverine and estuarine situations.

A solute or pollutant is defined as any substance that is advected by water and well mixed in the vertical direction. The interest is usually focused around the time evolution of a solute concentration within a complex hydrodynamic system, that is, given the solution concentration at a specific time and space, the model determines the spatial distribution of the solute concentrations at for future times. This physical process is accounted for the advection-dispersion equation and can incorporate the effect of reaction with the water and with other solutes.

5.11.1 Model Equations

Although RiverFlow2D PL can handle multiple pollutants simultaneously, for the sake of clarity in this section the transport of only one solute is presented coupled to the 2D model. The solute

transport equations will be expressed in a conservative form, assuming that the velocities and the water depth may not vary smoothly in space and time.

Correspondingly, the 2D shallow water model with solute transport can be written in unique coupled system:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{H}(\mathbf{U}) + \mathbf{R}(\mathbf{U}) + \mathbf{D}(\mathbf{U}) \tag{5.79}$$

where

$$\mathbf{U} = (h, q_x, q_y, h\phi)^T$$

$$\mathbf{F} = \left(q_x, \tfrac{q_x 2}{h} + \tfrac{1}{2} g h 2, \tfrac{q_x q_y}{h}, h\phi u\right)^T, \qquad \mathbf{G} = \left(q_y, \ \tfrac{q_x q_y}{h}, \ \tfrac{q_y 2}{h} + \tfrac{1}{2} g h 2, \ h\phi v\right)^T \tag{5.80}$$

$$\mathbf{H} = (0, \ gh(S_{0x} - S_{fx}), \ gh(S_{0y} - S_{fy}), \ 0)^T$$

and ϕ is the depth-averaged solute concentration. The sources terms associated to the solute transport equation are expressed as follows:

$$\mathbf{R} = (0, 0, 0, -Kh\phi)^T \qquad \mathbf{D} = \left(0, 0, 0, \overrightarrow{\nabla}(Dh\overrightarrow{\nabla}\phi)\right)^T \tag{5.81}$$

where K is the uptake constant and D is an empirical diffusion matrix.

5.11.2 Pollutant Transport Finite-Volume Numerical Solution

In RiverFlow2D, the solute transport has been considered letting aside the consideration concerning diffusion terms. However many strategies such as splitting and computing separately the advection and the diffusion terms or solving the diffusion implicitly (Murillo, Burguete, Brufau & García-Navarro 2005, Murillo, García-Navarro & Burguete 2008), have been developed to avoid small values in the time step size due to the combination of the CFL and Peclet number.

The numerical resolution of the solute transport equation under an explicit finite-volume method is frequently performed by solving the depth-averaged concentration apart from the shallow water equations, that is, using a simpler decoupled algorithm. Once the hydrodynamic equations have been solved, the corresponding substances or solutes are advected with these flow field previously computed.

In order to get a fully conservative method, RiverFlow2D considers the complete system including the hydrodynamic and the transport equations. Mathematically, the complete system conserves the hiperbolicity property, implying the existence of a 4×4 Jacobian matrix for the 2D model. On this basis we can apply the straightforward procedure described above, allowing a Roe's local linearization and expressing the contributions that arrive to the cell as a sum of waves. To ensure conservation and bounded values in the final solute concentration even in extreme cases, a conservative redistribution of the solute maximum fluxes as proposed in (Murillo, García-Navarro & Burguete 2012) was implemented in RiverFlow2D.

According to (Murillo et al. 2012), once the hydrodynamic part is properly formulated, a simple numerical flux $q^{\downarrow}$, directly related to the Roe's linearization, which is able to completely decouple the solute transport from the hydrodynamic system in a conservative way is used. Therefore,

$$q_k^{\downarrow} = q_i + \sum_{m=1}^{3} \left(\widetilde{\lambda}^{-} \ \widetilde{\gamma} \ \widetilde{\mathbf{e}}_1\right)_k^m \tag{5.82}$$

where $q_i = (h\mathbf{un})_i$ and the decoupled numerical scheme for the solute transport equation is written as:

$$(h\phi)_i^{n+1} = (h\phi)_i^n - \frac{\Delta t}{A_i}\sum_{k=1}^{N_E}(q\phi)_k^{\downarrow} l_k \tag{5.83}$$

where

$$\phi_k^{\downarrow} = \begin{cases} \phi_i & if \quad q_k^{\downarrow} > 0 \\ \phi_j & if \quad q_k^{\downarrow} < 0 \end{cases} \tag{5.84}$$

in cell i. A sketch of the fluxes is showed in Figure 5.12.

From a physical point of view, the new solute mass at a fixed cell can be seen as exchanging water volumes with certain concentration through the neighboring walls and mixing them (finite-volume Godunov's type method) with the former mass existing in the previous time (Figure 5.16). According to this philosophy, the outlet boundary cells will require a special treatment when applying this technique in order to extract the corresponding solute mass through the boundary walls. For this reason, it is necessary to define $q^{\downarrow} = (h\mathbf{u}\cdot\mathbf{n})_{BC}$ and $\phi^{\downarrow} = \phi_{BC}$ at the boundary wall and to include this contribution for the updating of the boundary cell BC (see Figure 5.17).

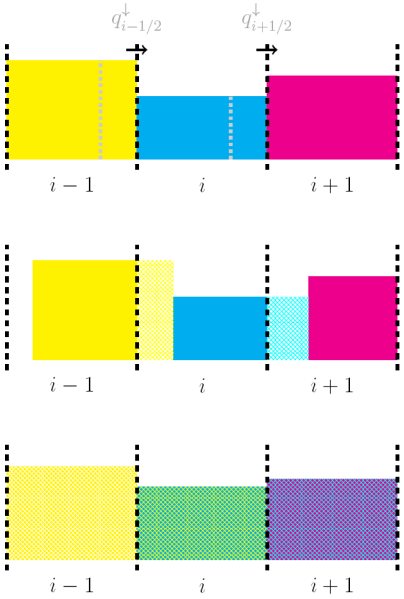


Figure 5.16 – Physical representation of solute mass exchange between cells with $q^{\downarrow}_{i-1/2}, q^{\downarrow}_{i+1/2} > 0$.

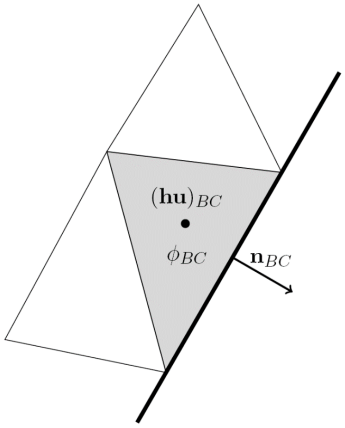


Figure 5.17 – Extraction of mass solute in an outlet boundary cell.

As shown, the formulation reduces to compute a class of numerical flux $q^{\downarrow}$ using the already computed averaged values at each edge for the hydrodynamic pa*Pollutant Transport* tabrt. Apart from ensuring a perfect conservation and bounded free-oscillatory solutions (Murillo et al, 2012), this simply discretization decreases substantially the number of computations that would be necessary for the complete coupled system.

5.11.3 Entering Data for the Pollutant Transport Model

To enter data for a pollutant transport simulation use the *Pollutant Transport* tab in the *Global Parameters* dialog accessible from the RiverFlow2D menu. Also make sure that the Pollutant Transport check box is active in thee Control Data tab.

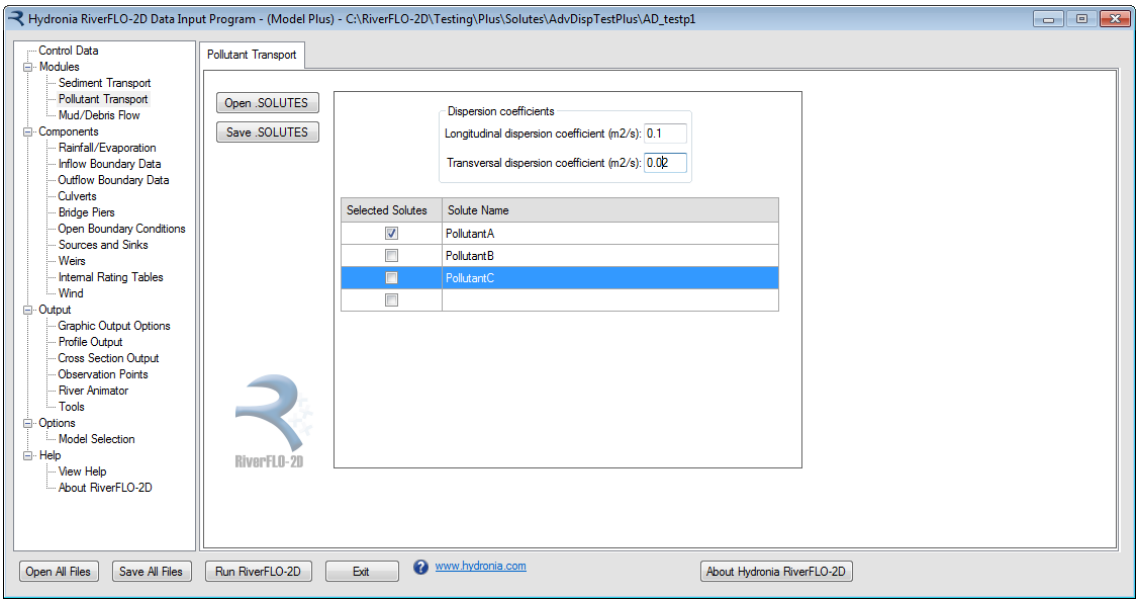


Figure 5.18 – RiverFlow2D *Pollutant Transport* tab.

5.11.4 Assumptions of the Pollutant Transport Model

The main assumptions involved in the present version of RiverFlow2D model are:

1. There is no predetermined limit to the number of pollutants.
2. Interaction between solutes and between each solute and water are assumed to be first order reactions.
3. All inflow boundaries where either discharge or water elevation is imposed must provide a concentration time series for each pollutant.

6 Code Parallelization

6.1 RiverFlow2D CPU

RiverFlow2D code has been parallelized using OpenMP directives available in the Intel C++ compiler version 13.0. OpenMP Application Program Interface (API) supports multi-platform shared-memory parallel programming in C/C++ and Fortran on architectures, including MAC OS, Unix and Windows platforms (OpenMP, 2009). OpenMP provides instructions to parallelize existing serial codes to run in shared-memory platforms ranging from affordable and widely available multiple-core computers to supercomputers. Using this parallelization approach RiverFlow2D dynamically distributes the computational workload between as many processors or cores as are available. In this way the model optimizes its computations to the particular architecture of each computer.

Figure 6.1 shows the speedup of the model with respect to the number of processors/cores on a DELL Precision 7400 computer with 2 Intel Xeon CPU X5472 @3.00GHz and 16GB of RAM. With 8 cores, the model runs more than 4 times faster than with the non-parallelized model. One hour simulation takes approximately 6 minutes using the parallelized model in this particular computer platform.

6.2 RiverFlow2D GPU

The GPU version of the RiverFlow2D model offers amazing speedups that considerably reduce run times. RiverFlow2D GPU implements two strategies: OpenMP parallelization and GPU techniques. Since dealing with transient inundation flows the number of wet elements changes during the simulation, a dynamic task assignment to the processors that ensures a balanced work load has been included in the Open MP implementation. RiverFlow2D strict method to control volume conservation (errors of Order $10^{-14}\%$) in the numerical modeling of the wetting/drying fronts involves a correction step that is not fully local which requires special handling to avoid degrading the model performance. The GPU version reduces the computational time by factors of up to 150X when compared with non-parallelized CPU (1-core) runs. Figure 6.2 shows performance tests using recent GPU hardware technology, that demonstrate that the parallelization techniques

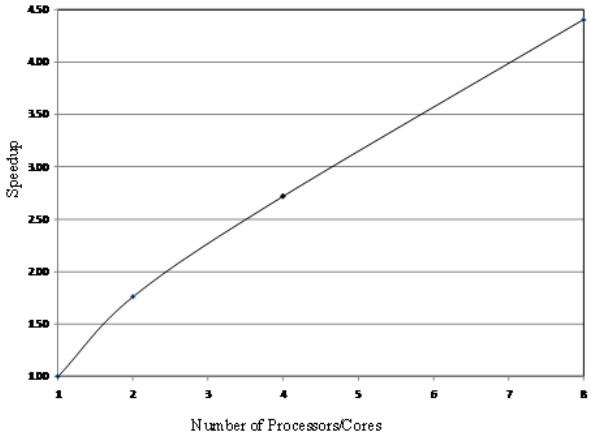


Figure 6.1 – Speed up using RiverFlow2D parallelized code as a function of number of processor cores.

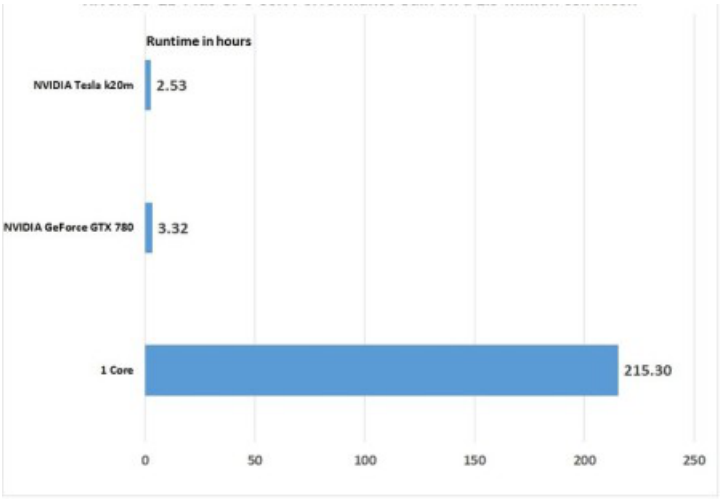


Figure 6.2 – Speed up using RiverFlow2D parallelized code as a function of number of processor cores.

implemented in RiverFlow2D GPU can significantly reduce the computational cost.

7

Hydraulic and Hydrologic Components

RiverFlow2D components are internal boundary conditions that can be used to complement calculations that may not be directly handled using the 2D flow equations. Components can be specified on polygons, polylines or points, depending on the required data.

The following hydrologic components are set over polygons:

- **Rainfall and Evaporation:** accounts for spatially distributed rainfall and evaporation.
- **Infiltration:** accounts for infiltration losses.
- **Wind:** allows incorporating the effect of spatially distributed wind stress on the water surface.

The following hydraulic components are set over polylines (feature arcs):

- **Bridges:** account for general geometry bridges including pressure flow and overtopping.
- **Internal Rating Tables:** provide an internal relationship of water elevation and discharge..
- **Gates:** used to represent sluice gate structures.
- **Weirs:** represent crested structures such as weirs, levees, sound walls, etc., where there is a unique relationship between discharge and depth.

Hydraulic components that are entered on points are:

- **Bridge Piers:** account for pier drag forces in a simplified formulation.
- **Culverts:** one dimensional conveyance conduits where discharge can be calculated using equations for circular or box structures, and rating tables.
- **Sources and Sinks:** provide a mean to enter point inflows or outflows that may vary in time.

7.1 Bridges Component

RiverFlow2D provides several options to integrate bridge hydraulics into the 2D mesh calculations. The most common option is to create the pier plan geometry generating a 2D triangular-cell mesh that represents each pier as a solid obstacle. In that case, the model will compute the flow around the pier and account for the pier drag. This would be the preferred approach when the user needs to know the detailed flow around the piers, but it does not account for pressure flow or overtopping conditions. In this option, the resulting mesh around piers has commonly very small cells which can lead to increased computer times.

The *Bridges* component is a comprehensive bridge hydraulics computation tool that does not require capturing bridge pier plan geometry in detail, therefore allowing longer time steps, while allowing calculating the bridge hydraulics accounting for arbitrary plan alignment, complex bridge geometry, free surface flow, pressure flow, overtopping, combined pressure flow and overtopping, and submergence all in 2D.

This component requires defining the bridge alignment in plan and the bridge geometry cross section. The bridge alignment is given in the `.bridges` data file which is generated by RiverFlow2D model based on the user defined data in the RiverFlow2D Data Input Program. To run a simulation with the bridges component, you need to select the option in the *Control Data* panel as shown in Figure 7.1.

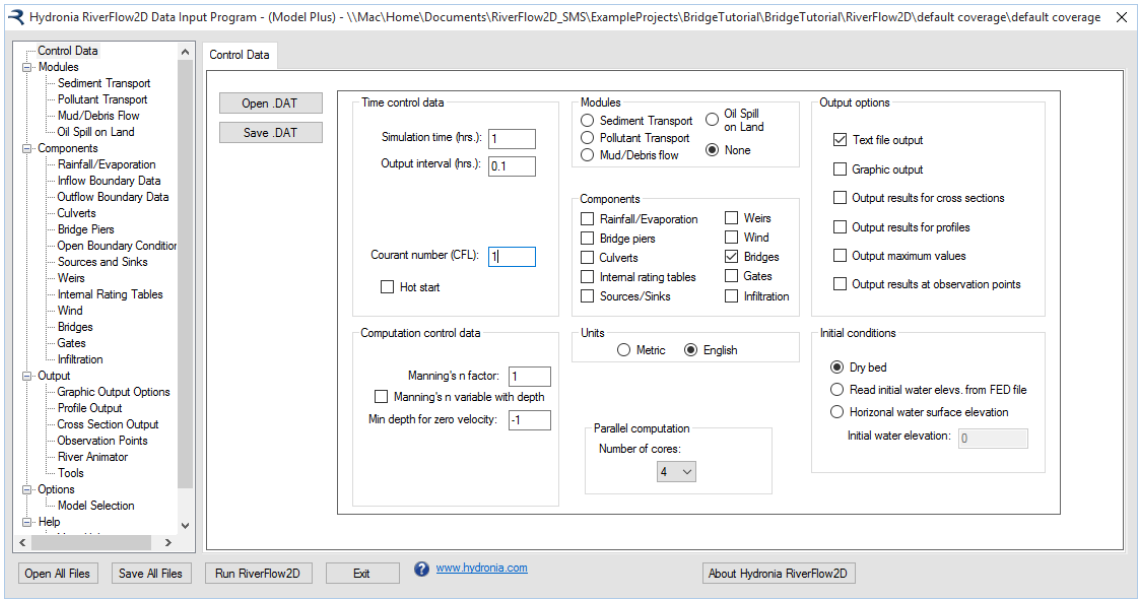


Figure 7.1 – RiverFlow2D *Control Data* panel with Bridges component selected.

The bridge plan data is entered in the SMS conceptual model coverage. To create a bridge, first make sure that the coverage where you defined the domain outline is active. Then, create an arc that will define the bridge geometry in plan.

Now, draw a polyline clicking on each vertex until you reach the desired end when you can double click to complete the polyline.

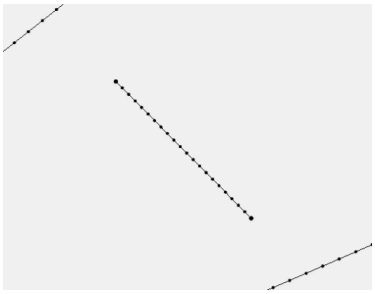


Figure 7.2 – Bridge polyline.

Then select the polyline using the Select Feature Arch icon, and right click to redistribute the vertices on the polyline. While the polyline is selected, right click on the arc and select *Attributes...* and on the *Feature arc Attributes* dialog, select *Boundary conditions* and click *Options...*

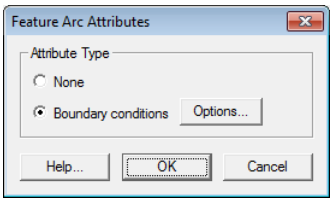


Figure 7.3 – Feature arc attributes.

On the *Boundary Conditions* Dialog, select the *Bridges* component and enter the desired BridgeID (any text), and the bridge geometry file name that contains the bridge geometry data and is explained in the next section.

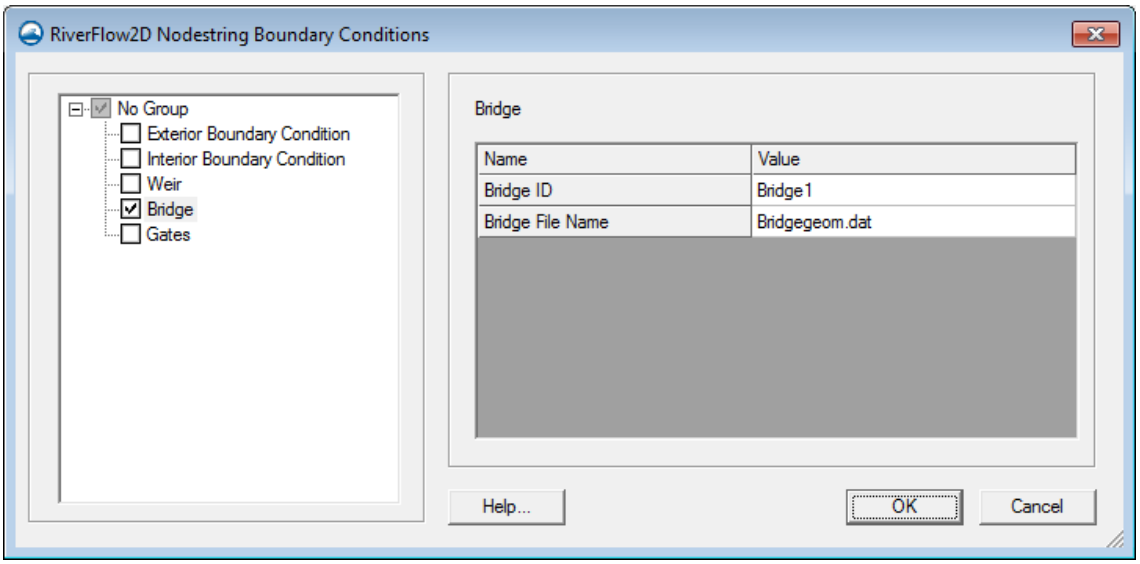


Figure 7.4 – Bridge dialog.

Since the bridge polyline must pass through nodes, it is essential that mesh generation engine creates nodes along the bridge polyline. This is easily done recreating the mesh using the *Feature Objects/Map → 2D Mesh* command.

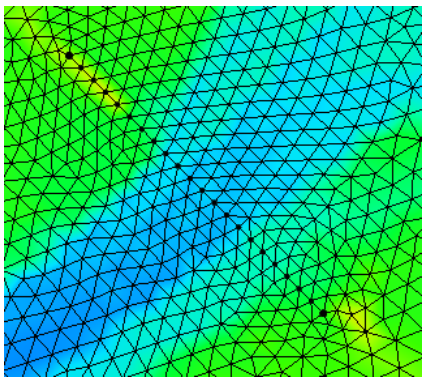


Figure 7.5 – Bridge polyline on mesh.

Note: There is no limit to the number of bridges that can be used.

7.1.1 Bridge Geometry Data File

The bridge geometry cross section file is necessary to define the bridge cross section. It is defined by four polylines and the fined in five columns as follows:

Line 1: Number of points defining polylines.

NP

NP lines with these entries:

STATION(1) BEDELEV(1) ZLOWER(1) LOWCHORD(1) DECKELEV(1)

...

STATION(NP) BEDELEV(NP) ZLOWER(NP) LOWCHORD(NP) DECKELEV(NP)
.

The relationship between the four polylines must be as follows:

- **For all stations, STATION(I) $\leq$ STATION(I+1)**
- **BEDELEV $\leq$ ZLOWER $\leq$ LOWCHORD $\leq$ DECKELEV**
- **In a given line all elevations correspond to the same station.**
- **The space between BEDELEV and ZLOWER is blocked to the flow.**
- **The space between ZLOWER and LOWCHORD is open to the flow.**
- **The space between LOWCHORD and DECKELEV is blocked to the flow.**

7.1.1.1 Example of the Bridge Cross Section File

The following table is an example one of the geometry file that schematically represents the bridge in 7.6.

NP 23

Station	BedElev	ZLower	LowChord	DeckElev
0.00	142.00	142.00	142.00	142.00
96.68	125.72	125.72	125.85	142.00
193.37	123.03	123.03	123.32	142.00
290.05	119.86	119.86	120.79	142.00
386.74	110.37	110.37	120.79	142.00
483.42	109.00	109.00	120.79	142.00
580.10	107.58	107.58	120.79	142.00
676.79	106.35	106.35	120.79	142.00
750.00	106.30	106.30	120.79	142.00
750.00	106.30	106.30	106.44	142.00
780.00	106.30	106.30	106.55	142.00
780.00	106.30	106.30	120.79	142.00
870.16	105.18	105.18	120.79	142.00
966.84	106.77	106.77	120.79	142.00
1063.52	107.30	107.30	120.79	142.00
1160.21	116.47	116.47	120.79	142.00
1256.89	116.02	116.02	120.79	142.00
1353.58	116.09	116.09	120.79	142.00
1450.26	119.61	119.61	120.79	142.00
1546.94	121.24	120.92	120.79	142.00
1643.63	124.74	124.74	124.67	142.00
1644.00	142.00	142.00	142.00	142.00

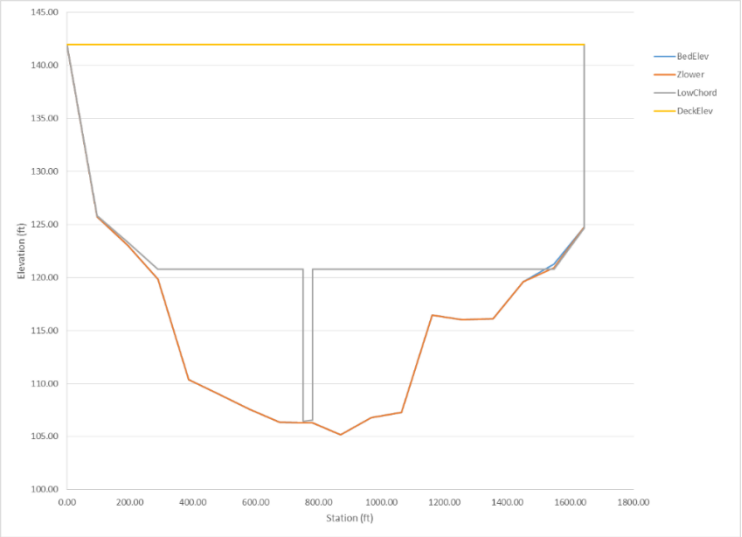


Figure 7.6 – Front view of a bridge cross section.

Table 7.1 – Variable Descriptions for the bridge cross section geometry file.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BEDELEV	R	-	m or ft	Bed elevation. Must be the lowest elevation for all polylines at a given point.
DECKELEV	R	-	m or ft	Elevation of the bridge deck. Must be the highest elevation for all polylines at a given point.
LOWCHORD	R	-	m or ft	Elevation of the lower bridge deck. LOWCHORD must be larger or equal to ZLOWER and smaller or equal to DECKELEV for a particular point. The space between LOWCHORD and DECELEV is a blocked area to the flow.
NP	I	-	> 1	Number of points defining cross section polylines.
STATION	R	-	m or ft	Distance from leftmost point defining cross section polyline. All polylines points must have a common station.
ZLOWER	R	-	m or ft	Elevation of lower polyline. ZLOWER must be larger or equal to BEDELEV and smaller or equal to LOWCHORD for a given point. The space between BEDELEV and ZLOWER is a blocked area to the flow. The space between ZLOWER and LOWCHORD is open space. If the bridge has no holes, ZLOWER must be identical to BEDELEV.

Note: R = Real variable. I = Integer variable. S = Text variable.

7.1.2 Bridge Calculations

To model bridges, the source term in the dynamic equation is split in three terms $\mathbf{S} = \mathbf{S}_z + \mathbf{S}_f + \mathbf{S}_b$. The term $\mathbf{S}_z$ defined as

$$\mathbf{S}_z = \left(0,\ -gh\frac{\partial z}{\partial x},\ -gh\frac{\partial z}{\partial y}\right)^T \tag{7.1}$$

expresses the variation of the pressure force along the bottom in the x and y direction respectively, formulated in terms of the bed slopes of the bottom level z. The term $\mathbf{S}_f$

$$\mathbf{S}_f = \left(0, -\frac{\tau_{f,x}}{\rho}, -\frac{\tau_{f,y}}{\rho}\right)^T \tag{7.2}$$

involves the the bed shear stresses $\tau_{f,x}, \tau_{f,y}$ in the x and y direction respectively, with ρ the density of the fluid. The last term, $\mathbf{S}_b$ stands for local energy losses due to other processes

$$\mathbf{S}_b = (0,\ -ghS_{b,x}, -ghS_{b,y})^T \tag{7.3}$$

and will be used to represent bridges.

The description of energy losses for the friction term associated to the bed stress in equation (7.2) is commonly formulated as an extension of a 1D formulation. The same approach is applied in RiverFlow2D deriving 1D closure relations for the definition of the bridge source term. Note that although the terms $S_{b,x}$ and $S_{b,y}$ represent energy losses in the presence of bridges, they are actually acting as a momentum sink. Empirical models for the energy loss caused by the bridge are described next.

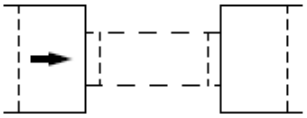


Figure 7.7 – Top view of a bridge showing the cross sections of interest. Only two piers are depicted for simplicity.

7.1.2.1 Energy dissipation in bridges

The formulation of Borda-Carnot for energy loss in sudden contractions or expansions in pipes can also be used for channels (Ratia, Murillo & García-Navarro 2014). This in turn can model bridges with free water surface. The energy loss will be expressed in terms of the total available head ΔH_{BC}, and represents the total mechanical energy of the flow. In a 1D framework the head loss ΔH_{BC} is expressed as follows

$$\Delta H_{BC} = (\Delta H_c + \Delta H_e) \tag{7.4}$$

where ΔH_c and ΔH_e are the contraction and expansion losses respectively

$$\begin{aligned} \Delta H_c &= \frac{\bar{v}_1^2}{2g}\left[\left(\frac{1}{m}-1\right)^2+\frac{1}{9}\right]\left(\frac{A_1}{A_2}\right)^2 \\ \Delta H_e &= \frac{\bar{v}_4^2}{2g}\left[\left(\frac{A_4}{A_3}-1\right)^2+\frac{1}{9}\right] \end{aligned} \tag{7.5}$$

where m is a typical value for the contraction coefficient, $m = 0.62$ (Ratia et al. 2014) and the areas A_1 to A_4 refer to effective cross sectional flow area. The numbering of areas is shown in Figures 7.7. Area 1 is a section upstream of the bridge while area 4 is a downstream section. Areas 2 and 3 are sections inside the bridge, near the entrance and exit respectively.

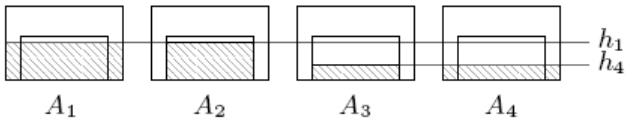


Figure 7.8 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in free surface bridges.

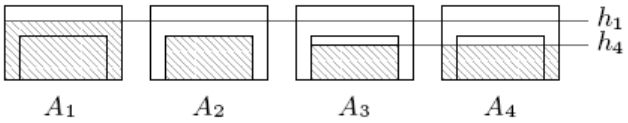


Figure 7.9 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in a partially submerged bridges.

The values $\bar{v}_1$ and $\bar{v}_4$ are the cross sectional averaged velocities

$$\bar{v}_1 = \frac{Q_1}{A_1(d_1)}, \qquad \bar{v}_4 = \frac{Q_4}{A_4(d_4)} \tag{7.6}$$

with Q_1 and Q_4 the total discharges in areas A_1 and A_4, expressed as a function of the water surface elevation, $d = h + z$. Different regimes can be described. Figure 7.8 shows a skecth of

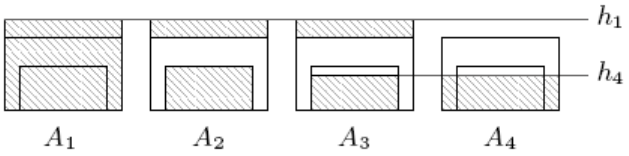


Figure 7.10 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in fully submerged bridges.

the areas considered in the free surface case, Figure 7.9 shows the equivalent areas for partially submerged bridges and Figure 7.10 for fully submerged bridges.

7.1.2.2 Integration of the energy losses associated to bridges

The unified formulation of the source terms accounting for energy losses associated to bridges also ensures the well balanced property in steady cases with velocity. In order to do that it is necessary to define $\mathbf{S}_{\mathbf{n}b}$ at the edge of the RP where the bridge exists. The source term $\mathbf{S}_{\mathbf{n}b}$ is formulated as

$$(\mathbf{S}_{\mathbf{n}b})_k = \begin{pmatrix} 0 \\ -g\widetilde{h}\,\delta H n_x \\ -g\widetilde{h}\,\delta H n_y \end{pmatrix}_k \tag{7.7}$$

with

$$\delta H = \Delta H \frac{\widetilde{\mathbf{u}}\mathbf{n}}{|\widetilde{\mathbf{u}} \cdot \mathbf{n}|} \tag{7.8}$$

where ΔH is the singular loss term used to represent bridges.

Computation of ΔH in a real mesh is done as follows. The bridge is defined on cell edges (bold line in Figure 7.11), and the cells on both sides of these edges are considered to form two cross sections Γ_L and Γ_R (hatched cells in Figure 7.11). Note that it is possible to define bridges in arbitrary orientations and in structured/unstructured meshes.

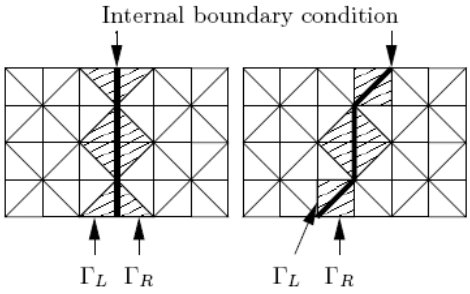


Figure 7.11 – Application of the scheme in triangular structured meshes. Normal bridge (left) and oblique bridge (right).

In each time step, the necessary variables for the calculation of the global bridge head loss are averaged from the cells in both upstream and downstream sections as illustrated in Figure 7.11. The discharge is computed as

$$Q_{\Gamma_L} = \sum_{k \in \Gamma_L} (\mathbf{qn})_k l_k \qquad Q_{\Gamma_R} = \sum_{k \in \Gamma_R} (\mathbf{qn})_k l_k \tag{7.9}$$

and the cross sectional average water level surface is estimated as

$$d_{\Gamma_L} = \frac{\sum_{k\in\Gamma_L} d_k l_k}{\sum_{k\in\Gamma_L} l_k} \qquad d_{\Gamma_R} = \frac{\sum_{k\in\Gamma_R} d_k l_k}{\sum_{k\in\Gamma_R} l_k} \tag{7.10}$$

involving cells with values of $h > 0$. The signs of Q_{Γ_L} and Q_{Γ_R} are used to determine which section is upstream and which downstream. If $Q_{\Gamma_L} \geq 0$, the discharge across the bridge is computed as $Q = Q_{\Gamma_L}$ and the areas are computed using $d_1 = d_{\Gamma_L}$ and $d_4 = d_{\Gamma_R}$. In case that $Q_{\Gamma_L} < 0$, the discharge across the bridge is computed as $Q = Q_{\Gamma_R}$ and the sections are reversed setting $d_1 = d_{\Gamma_R}$ and $d_4 = d_{\Gamma_L}$. Next, the different areas and the cross-sectional top width are calculated as a function of the average water level surface. From these values the total head loss Δ_H can be evaluated.

7.1.2.3 Influence of the Bridge Width

The computation algorithm used in the Bridges component neglects the effect of the structure width (distance perpendicular to the bridge alignment) on the head loss. According to (Yarnell 1934*b*) and (Yarnell 1934*a*), the bridge width has a small influence on the flow variables such as water surface elevation and energy loss. Yarnell performed experiments in a laboratory flume with bridges having rectangular piers with width-to-length ratios (w:l) of 1:4, 1:7 and 1:13, where w is the pier dimension perpendicular to the flow direction and l the pier length parallel to the flow. Yarnell noted that the energy loss increased less than 10% for the configuration with longest piers. (Ratia et al. 2014) performed numerical simulations to confirm Yarnell's experiments using piers with the same with-to-length ratios and a wide range of approach discharges (see Figure 7.12).

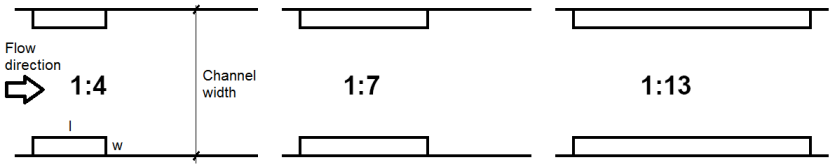


Figure 7.12 – Bridge pier proportions used to asses the influence of the structure width.

Numerical results indicate that the changes in total head loss across the structure are very similar for the three configurations (see Figure 7.13).

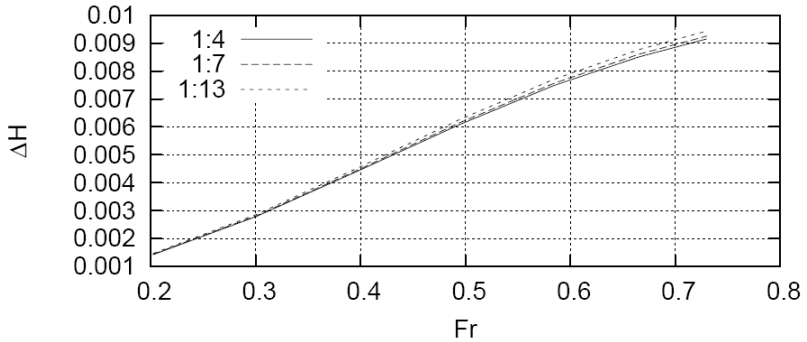


Figure 7.13 – Influence of the structure width on the total head change (ΔH) across the bridge as a function of the Froud number downstream.

7.2 Bridge Piers

The Bridge Piers component allows accounting for the losses caused by piers in the flow field in a simplified way, without requiring a refined mesh around the actual pier plan geometry.

To run a simulation with the Bridge Piers Component, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Program as shown in Figure 7.14.

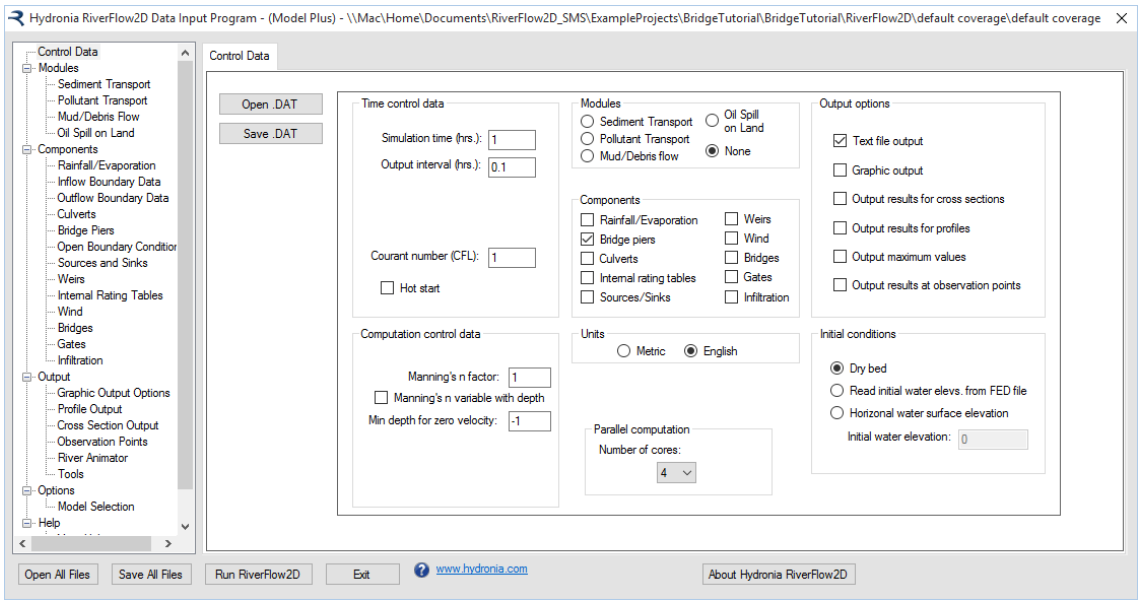
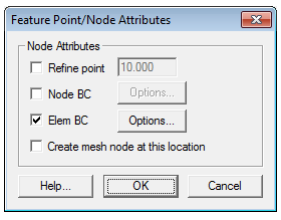


Figure 7.14 – RiverFlow2D *Control Panel* dialog with Bridge Piers component selected.

Bridge pier data is entered in the SMS conceptual model coverage. To create a bridge pier, first make sure that the Pier Component is active and then follow these steps:

- Click on the Create Feature Point icon and click on the location where you want to enter the pier
- Now click on the Select Feature Point icon and click on the point entered in the previous step
- On the Feature Objects menu, click Attributes...
- Select Elem BC and click Options... button:



- Select Pier, enter the Pier Name and other data as shown:

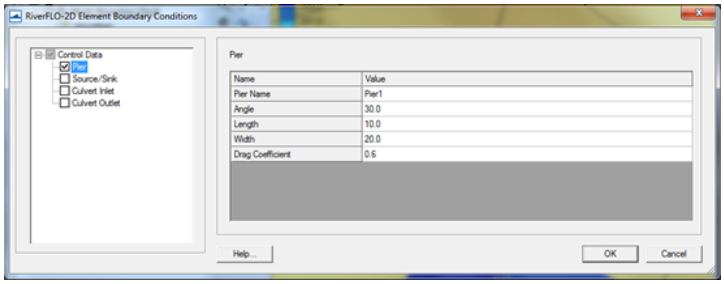


Figure 7.15 – Pier Data dialog.

Note: There is no limit to the number of Bridge Piers that can be used.

7.2.1 Bridge Pier Calculation

The Bridge Pier component can be used when the pier plan area is small compared to the element area and there is no need to determine the details of the flow field around the structure. In this component the model computes the drag force on each pier as a function of the drag coefficient, water density, flow velocity and wetted pier projected area as shown in Eq. 7.11:

$$F_D = \frac{1}{2} C_D \rho U^2 A_P \tag{7.11}$$

Where C_D is the pier drag coefficient, ρ is the water density, U is the water velocity, and A_P is the pier wetted area projected normal to the flow direction. Piers are assumed to be located on elements that not necessarily conform to the pier geometry as shown on the following figure.

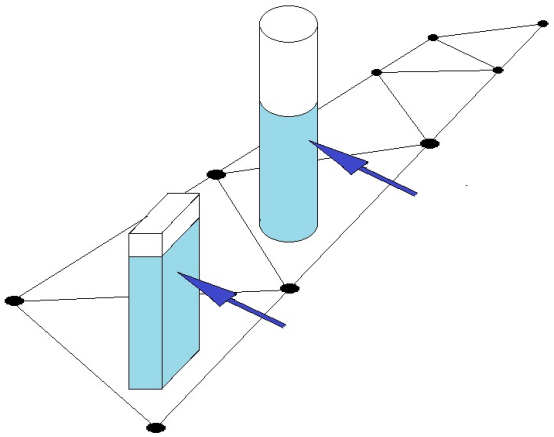


Figure 7.16 – Piers inside cells.

Piers can be circular or rectangular in plan. Rectangular piers are located on elements based on the pier center coordinates and the angle between the axis along the largest dimension and the X-axis as shown in the following figure.

Velocity vector magnitude and approach angle usually varies in time during unsteady flow computations and is used to calculate the projected area. To account for the resistance force that

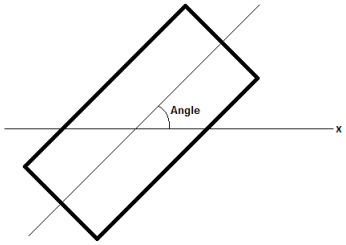


Figure 7.17 – Schematic view of a rectangular pier.

the pier exerts on the flow, RiverFlow2D converts it to the distributed shear stress on the element where the pier centroid coordinate is located. The resulting pier shear stress expressions in x and y directions are as follows:

$$\tau_{px} = \frac{1}{2} C_D \rho U \sqrt{U^2 + V^2} \frac{A_P}{A_e} \tag{7.12}$$

$$\tau_{py} = \frac{1}{2} C_D \rho V \sqrt{U^2 + V^2} \frac{A_P}{A_e} \tag{7.13}$$

where A_e is the element area.

7.3 Culverts Component

The culvert component in RiverFlow2D allows incorporating 1D hydraulic structures that convey water between two locations on the mesh, or between a point on the mesh an another outside.

To run a simulation with the Culverts Component, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Program dialog as shown in Figure 7.18.

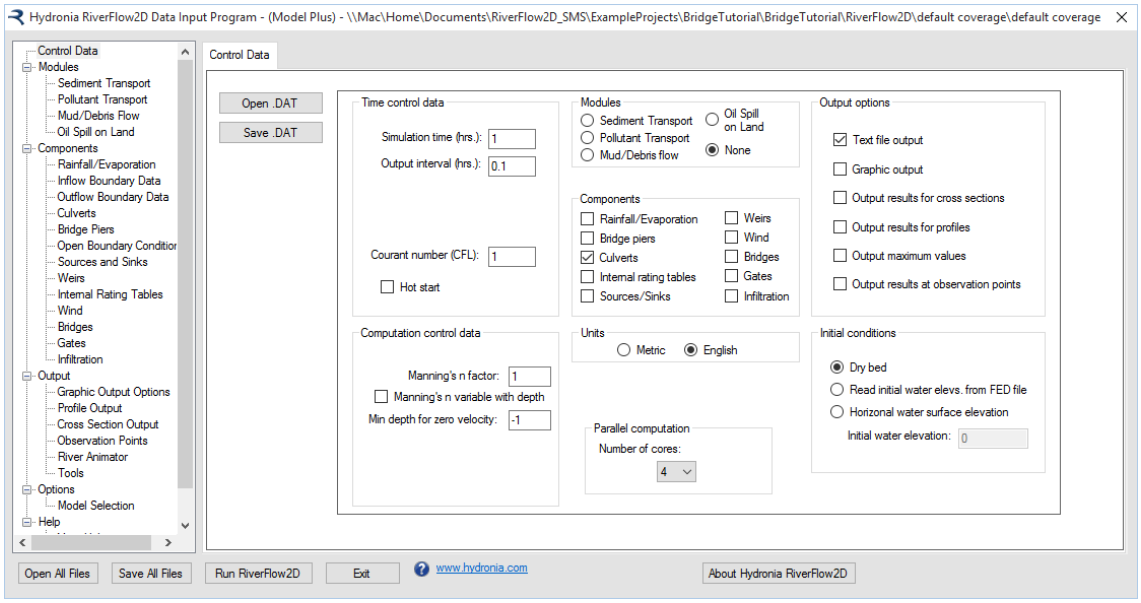


Figure 7.18 – RiverFlow2D *Global Parameters* dialog with Culverts Component selected.

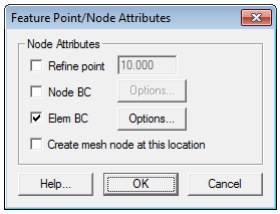
The culvert data is entered in the SMS conceptual model coverage:

1. Click on the *Create Feature Point* icon and click on the location where you want to enter the inlet point.

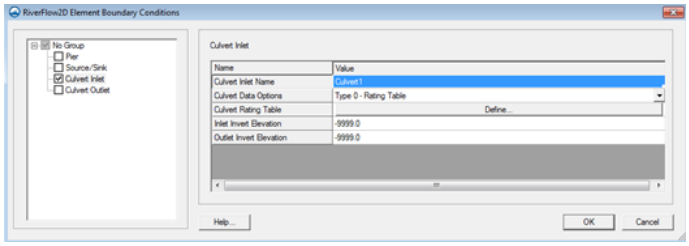
2. Now click on the Select *Feature Point* icon and right click on the point entered in the previous step.

3. On the *Feature Objects* menu, click *Attributes...*

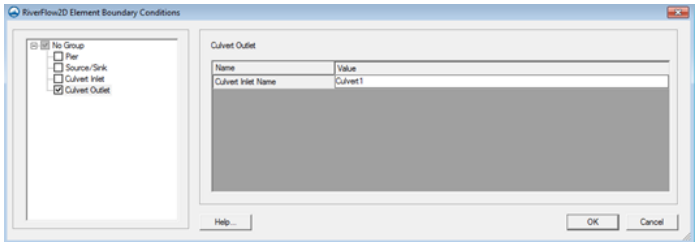
4. Select *Elem BC* and click *Options...* button:



5. Select *Culvert*, enter the Culvert ID name (e.g. Culvert1) and enter additional data depending on the culvert formulation desired.



6. Then select the Culvert Outlet Element and enter the Culvert Inlet Name:



7. Before running the model, make sure that Culverts it is selected in the *Global Parameters* dialog.

There are two options to compute culvert discharge in RiverFlow2D. When the user selects Rating Table calculation and provides a rating table on the associated file, the model determines the discharge by interpolation as a function of the depth upstream. If the user enters Culvert calculation using culvert characteristics, the model will calculate the discharge based on the culvert geometric characteristics given in the file. Both procedures are described in more detail below.

Note: There is no limit to the number of culverts that can be used.

7.3.1 Culvert Calculation using a Rating Table (CulvertType = 0)

When the user provides a rating table, the culvert calculation algorithm is as follows:

1. If at least one of the culvert ends is wet, determine the flow direction based on the water surface elevations at each culvert end,
2. Interpolate flow discharge from the rating table using the depth at the culvert inlet,
3. If depth at the culvert inlet is lower than minimum value in the rating table, then the discharge is assumed to be zero.
4. If depth at entrance is higher than maximum value in the rating table, then the discharge is assumed to be equal to that of the maximum depth.
5. The computed discharge is subtracted from the inlet element and added to the outlet element assuming instantaneous water volume transmission.

7.3.2 Culvert Calculation using a Culvert Characteristics (CulvertType = 1,2)

For CulvertType's 1 and 2, the model will calculate culvert discharge for inlet and outlet control using the FHWA procedure (Norman et al. 1985). Later Froehlich (2003) restated the algorithm in dimensionless form. The resulting formula is expressed as follows:

$$Q = N_b C_c A_c \sqrt{2gH_c} \tag{7.14}$$

where N_b is the number of identical barrels, C_c is a discharge coefficient that depends on the flow control and culvert geometric characteristics, A_c is the culvert area at full section, g is the gravitational acceleration, $H_c = WSEL_h - Z_{bi}$ for inlet control and $H_c = WSEL_h - WSE_{tw}$ for outlet control, WSE_h is the water surface elevation at the culvert inlet, Z_{bi} is the inlet invert elevation, WSE_{tw} is the water elevation downstream (tailwater). For inlet control calculation,

$$C_c = \begin{cases} \sqrt{\frac{1-\frac{D_c}{H_h}(Y+mS_0)}{2c'}} & for \quad H_h > 1.2D_c \\ \frac{1}{\sqrt{2}K'(1/M)}\left(\frac{H_h}{D_c}\right)^{\left(\frac{1}{M}-0.5\right)} & for \quad H_h \leq 1.2D_c \end{cases} \tag{7.15}$$

where $H_h = WSEL_h - Z_{bi}$ is the headwater depth. D_c is the culvert diameter for circular culverts and the base dimension for box culverts, $m = 0.7$ for mitered inlets and $m = -0.5$ for all other inlets. For outlet control, the following formula is used to determine C_c:

$$C_c = \left(1 + K_e + \frac{2gn_c^2 L_c}{R_c^{4/3}}\right)^{-0.5} \tag{7.16}$$

where R_c is the culvert hydraulic radius, K_e is the entrance loss coefficient that can be obtained from Table 7.3, n_c is the Manning's n obtained from Table 7.2, L_c is the culvert length, and Y, K', M', C' are inlet control coefficients (see Table 7.4).

Table 7.2 – Manning's n roughness coefficients for various culvert materials. Adapted from Froehlich (2003).

CULVERT BARREL MATERIAL	ENTRANCE DESCRIPTION	MANNING'S N N_c
Concrete	Good joints, smooth walls	0.012
	Projecting from fill, square-cut end	0.015
	Poor joints, rough walls	0.017
Corrugated metal	2-2/3 inch × 1/2 inch corrugations	0.025
	6 inch × 1 inch corrugations	0.024
	5 inch × 1 inch corrugations	0.026
	3 inch × 1 inch corrugations	0.028
	6 inch × 2 inch corrugations	0.034
	9 inch × 2-1/2 inch corrugations	0.035

Table 7.3 – Entrance loss coefficients K_e. Adapted from Froehlich (2003).

TYPE OF CULVERT	ENTRANCE DESCRIPTION*	ENTRANCE LOSS COEFFICIENT K_e
Concrete pipe	Projecting from fill, grooved end	0.2
	Projecting from fill, square-cut end	0.5
	Headwall or headwall with wingwalls (concrete or cement sandbags)	
	Grooved pipe end	0.2
	Square-cut pipe end	0.5
	Rounded pipe end	0.1
	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7
Corrugated metal pipe or pipe-arch	Projecting from embankment (no headwall)	0.9
	Headwall with or without wingwalls (concrete or cement sandbags)	0.5
	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7
Reinforced concrete box	Headwall parallel to embankment (no wingwalls)	
	Square-edged on three sides	0.5
	Rounded on three sides to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 30° to 75° to barrel	
	Square-edged at crown	0.4
	Crown edge rounded to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 10° to 30° to barrel	
	Square-edged at crown	0.5
	Wingwalls parallel to embankment	
	Square-edged at crown	0.7

*See Table 7.5 for inlet configurations.

Table 7.4 – Culvert inlet control formula coefficients. Adapted from Froehlich (2003).

BARREL MATERIAL	BARREL SHAPE	INLET DESCRIPTION*	K'	M	C'	Y
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45°bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7°bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30°to 75°flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90°and 15°flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0°flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45°flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18°to 33.7°flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; 3/4 inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45°bevels	1.5752	0.6670	1.0101	0.8200
Concrete	Rectangular	Headwall; 33.7°bevels	1.5466	0.6670	0.8107	0.8650
Concrete	Rectangular	Headwall; 45°skew; 3/4 in chamfers	1.6611	0.6670	1.2932	0.7300
Concrete	Rectangular	Headwall; 30°skew; 3/4 in chamfers	1.6961	0.6670	1.3672	0.7050
Concrete	Rectangular	Headwall; 15°skew; 3/4 in chamfers	.7343	0.6670	1.4493	0.6800
Concrete	Rectangular	Headwall; 10-45° skew; 45°bevels	1.5848	0.6670	1.0520	0.7500
Concrete	Rectangular	Wingwalls; non-offset 45°/flares;	1.5816	0.6670	1.0906	0.8030
Concrete	Rectangular	Wingwalls; non-offset 18.4°/flares; 3/4 in chamfers	1.5689	0.6670	1.1613	0.8060
Concrete	Rectangular	Wingwalls; non-offset 18.4°/flares; 30°/skewed barrel	1.5752	0.6670	1.2418	0.7100
Concrete	Rectangular	Wingwalls; offset 45°/flares; beveled top edge	1.5816	0.6670	0.9715	0.8350
Concrete	Rectangular	Wingwalls; offset 33.7°/flares; beveled top edge	1.5752	0.6670	0.8107	0.8810
Concrete	Rectangular	Wingwalls; offset 18.4°/flares; top edge bevel	1.5689	0.6670	0.7303	0.8870
Cor. metal	Rectangular	Headwall	0.2670	2.0000	1.2192	0.6900
Cor. metal	Rectangular	Projecting; thick wall	0.3023	1.7500	1.3479	0.6400
Cor. metal	Rectangular	Projecting; thin wall	0.4593	1.5000	1.5956	0.5700
Concrete	Circular	Tapered throat	1.3991	0.5550	0.6305	0.8900
Cor. metal	Circular	Tapered throat	1.5760	0.6400	0.9297	0.9000
Concrete	Rectangular	Tapered throat	1.5116	0.6670	0.5758	0.9700
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700

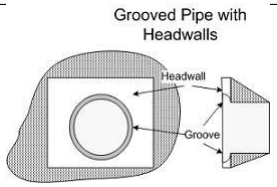
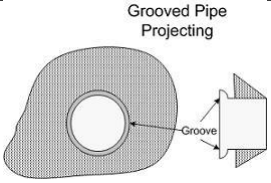
Continued on next page

Table 7.4 – continued from previous page

BARREL MATERIAL	BARREL SHAPE	INLET DESCRIPTION*	K'	M	C'	Y
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45°bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7°bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30°to75°flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90°and 15°flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0°flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45°flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18°to 33.7°flare; beveled edge 1.5466	0.6670	0.8010	0.8300	
Concrete	Rectangular	Headwall; 3/4 inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45°bevels	1.5752	0.6670	1.0101	0.8200

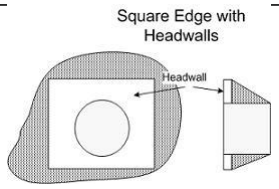
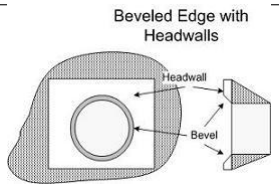
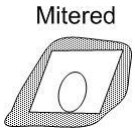
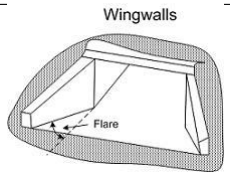
*See Table 7.5 for inlet configurations.

Table 7.5 – Culvert inlet configurations. Adapted from *www.xmswiki.com/xms/*.

INLET CONFIGURATION	DESCRIPTION
	End of the culvert barrel projects out of the embankment.
	Grooved pipe for concrete culverts decreases energy losses through the culvert entrance.
	This option is for concrete pipe culverts.

Continued on next page

Table 7.5 – continued from previous page

INLET CONFIGURATION	DESCRIPTION
	Square edge with headwall is an entrance condition where the culvert entrance is flush with the headwall.
	'Beveled edges' is a tapered inlet edge that decreases head loss as flow enters the culvert barrel.
	Mitered entrance is when the culvert barrel is cut so it is flush with the embankment slope.
	Wingwalls are used when the culvert is shorter than the embankment and prevents embankment material from falling into the culvert

The culvert computation algorithm works as follows:

1. If at least one of the culvert ends is wet, Determine the flow direction based on the water surface elevations at each culvert end.

2. Compute the culvert discharge using inlet control formulas.

3. Compute the culvert discharge using outlet control formulas.

4. Select the minimum discharge from the inlet and outlet control discharges.

5. If depth at the culvert inlet is lower than minimum value in the rating table, then the discharge is assumed to be zero.

6. The computed discharge is subtracted from the inlet element and added to the outlet element assuming instantaneous water volume transmission.

When using CulvertType 1 or 2, both ends of the culvert must be inside the mesh.

7.3.3 Assumptions of Culvert Calculations

1. The same rating table will be used to interpolate discharge regardless of the flow direction. In other words, if the flow is from element A to element B at some point during the simulation, depth at A will be used to interpolate discharge from A to B, but if at some other time flow changes from B to A, discharge will be interpolated using depth at B.
2. There is no outlet control on the rating table discharge calculation.
3. When using CulvertTypes 1 and 2, both ends of the culvert must be inside the mesh. It is not allowed to extract flow from the modeling domain when using these options.
4. Discharge calculation with CulverTypes 1 and 2 is only available for circular or box (rectangular) cross section culverts.
5. The entrance to a culvert is regarded as submerged when the head water depth, H, 1.2D, where D is the diameter of the circular culvert or the height of box culverts.

7.4 Gates Component

The GATES components allows integrating gates inside the modeling region. Each gate needs to be defined in terms of its plan alignment, crest elevation (Zc), gate height ($Hgate$) and the time history of apertures (Ha) given as a table in a file associated to each structure (see Figure 7.19). Figure 7.20 shows the flow modes that can be calculated through gates that include submergence and overtopping.

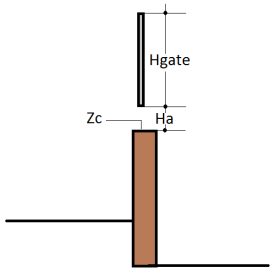


Figure 7.19 – Schematic cut view perpendicular to a gate structure.

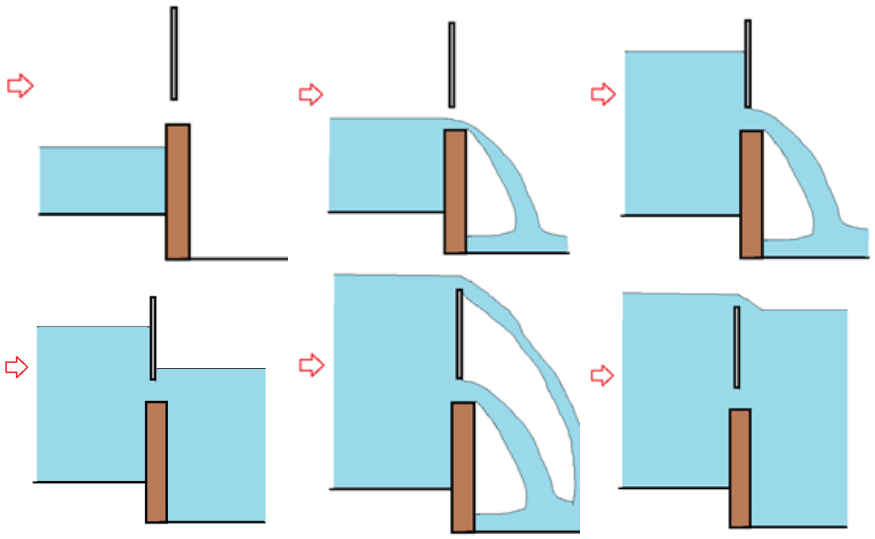


Figure 7.20 – Flow modes across gates.

To run a simulation with the gates component, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Program as shown in Figure 7.21.

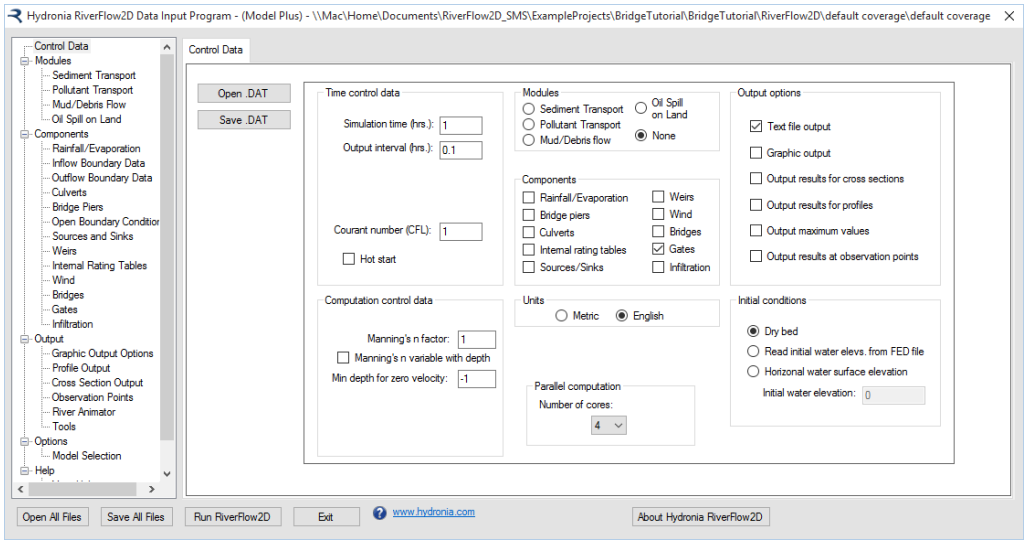


Figure 7.21 – RiverFlow2D *Control Data* panel with Gates Component selected.

The gate plan data is entered in the SMS conceptual model coverage. First create an arc that will define the gate geometry in plan. Draw a polyline clicking on each vertex until you reach the desired end when you can double click to complete the polyline.

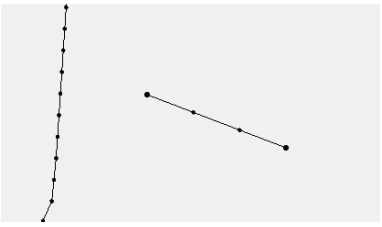


Figure 7.22 – Gate polyline.

Then select the polyline using the *Select Feature Arch* icon, and right click to redistribute the vertices on the polyline.

While the polyline is selected, right-click on the arc and select *Attributes...* and on the *Feature arc Attributes* dialog, select *Boundary conditions* and click Options...

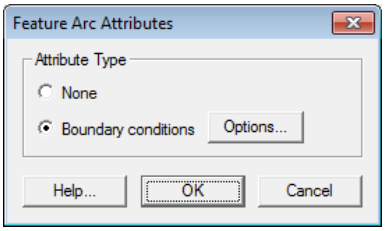


Figure 7.23 – Feature arc attributes.

On the *Nodestring Boundary Conditions* dialog, select the *Gates* component and enter the desired Gate ID (any text), and gate file name that contains the aperture time series, the crest elevation, crest height and gate *Cd*.

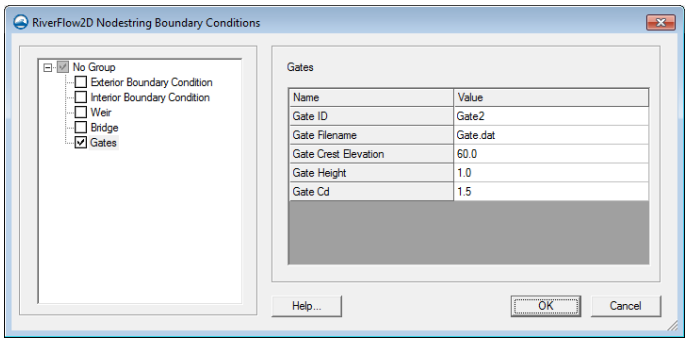


Figure 7.24 – Gates dialog.

Since the gate polyline must pass through nodes, it is essential that the mesh generation engine creates nodes along the polyline. This is easily done recreating the mesh using the *Feature Objects/Map → 2D Mesh* command.

Note: There is no limit to the number of gates that can be used.

7.4.1 Gate Calculations

The gate is simulated by assuming that the discharge per unit breadth q crossing the gate is governed by the difference between the water surface level ($d = h + z$) on both sides of the gate, referred to as d_l upstream of the gate and d_r downstream of the gate, and by the allowable gate opening, G_o. Several situations are envisaged. In the case that $G_o = 0$ the gate behaves as a solid wall and no flow crosses the gate. When the gate opening is larger than the surface water level on both sides, it no longer influences the flow. In any other case, assuming that $d_l < d_r$, without loss of generality, two different flow situations can occur depending on the relative values of G_o, z_l, z_r, d_l and d_r. When $G_o + max(z_l, z_r) < min(d_l, d_r)$, Figure 7.25, the discharge is given by

$$q = G_o K_1 (d_r - d_l)^{1/2} \tag{7.17}$$

with K_1 an energy loss coefficient. In RiverFlow2D K_1=3.33 (Henderson 1966).

When $G_o + max(z_l, z_r) > min(d_l, d_r)$, Figure 7.26, the discharge is given by

$$q = G_o K_2 \left[d_r - max(z_l, z_r)\right]^{1/2} \tag{7.18}$$

with K_2 another energy loss coefficient. In RiverFlow2D K_2=2.25 (Henderson 1966).



Figure 7.25 – Water levels for discharge under a gate in submerged conditions formulated as in (G1).

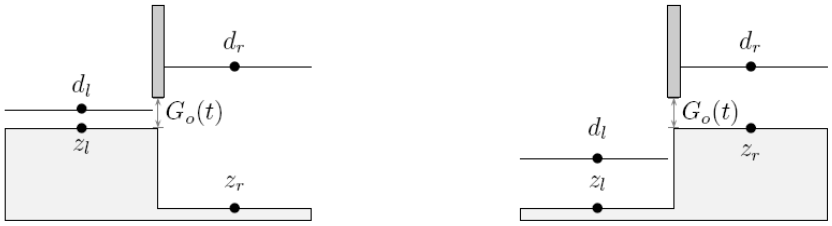


Figure 7.26 – Water levels for discharge under a gate in submerged conditions formulated as in (G2).

7.5 Rainfall, Evaporation and Infiltration

This section describes the Hydrologic computations implemented in the RiverFlow2D model. The component includes capabilities to integrate rainfall, evaporation and infiltration in the model simulations.

7.5.1 Rainfall

Rainfall is treated as a simple source term. It represents an additional input to the cell water depth in the previous step of flow calculation. You can set up local rainfall events for several regions of the watershed. This allows to simulate more realistic cases, in which data from several rain gauges are available.

Notice that the rainfall is set before the infiltration calculation. This is an important detail because the infiltration capacity strongly depends on the rainfall intensity, as we will see in the next section.

7.5.2 Infiltration

Infiltration represents another component of the hydrological budget and it can be defined as the process by which surface water enters the soil. In the RiverFlow2D model, infiltration is treated as a loss. This process is mainly governed by two forces: gravity and capillarity action. The model includes three methods to compute the infiltration losses: Horton, Green & Ampt and SCS-Curve Number.

7.5.2.1 Horton Infiltration Model

Horton's infiltration model (Horton 1933) suggests an exponential equation (7.19) for modelling the soil infiltration capacity f_p:

$$f_p = f_c + (f_0 - f_c)\, e^{-kt} \tag{7.19}$$

where f_0 and f_c are the initial and final infiltration capacities, both measured in m/s or in/s and k represents the rate of decrease in the capacity $(1/s)$.

The parameters f_0 and k have no physical basis, so they must be determinated from experimental data. A good source for experimental values of these parameters for different types of soils can be found in (Rawls, Yates & Asmussen 1976) and summarized in (ASCE 1996). Table 7.6 and Table 7.7 show the parameters for some general types of soil, as presented in (Akan 1993). Note that no k values are shown. A value of $k = 4.14\ hr^{-1}$ is recommended in the absence of any field data (Akan 1993).

Table 7.6 – Horton initial infiltration for different soils. Source: Akan(1993).

SOIL TYPE	f_0 **(mm/hr)**
Dry sandy soils with few to no vegetation	127
Dry loam soils with few to no vegetation	76.2
Dry clay soils with few to no vegetation	25.4
Dry sandy soils with dense vegetation	254
Dry loam soils with dense vegetation	152.4
Dry clay soils with dense vegetation	50.8
Moist sandy soils with few to no vegetation	43.18
Moist loam soils with few to no vegetation	25.4
Moist clay soils with few to no vegetation	7.62
Moist sandy soils with dense vegetation	83.82
Moist loam soils with dense vegetation	50.8
Moist clay soils with dense vegetation	17.78

Table 7.7 – Horton final infiltration for different soils. Source: Akan(1993).

SOIL TYPE	f_c **(mm/hr)**
Clay loam, silty clay loams	127
Sandy clay loam	1.3 - 3.8
Silt loam, loam	3.8 - 7.6
Sand, loamy sand, sandy loams	7.6 - 11.4

The equation (7.19) has to be applied after the surface ponding. In other words, we are assuming conditions of unlimited water supply at the surface. Under this consideration, the cumulative infiltration up to time t can be calculated by integrating the infiltration capacity:

$$F = \int_0^t f_p(t)dt = f_c t + \frac{f_0 - f_c}{k}\left(1 - e^{-kt}\right) \tag{7.20}$$

It is important to highlight the difference between the infiltration capacity f_p and the infiltration rate f. If we consider a rain event starting with a weak rainfall intensity ($R \leq f_p$), then all the rain will be infiltrated into the soil. On the other hand, if the rain exceeds the soil infiltration capacity or if the surface becomes ponded, this magnitude will determine the infiltration rate:

$$R \leq f_p \Rightarrow f = R \qquad\qquad R > f_p, t > t_p \Rightarrow f = f_p \tag{7.21}$$

where t_p represents the ponding time.

Following (Gupta 1995), for the two first rain intervals, the rainfall intensity is less than the infiltration capacity, so the real infiltration rate is equal to the rainfall rate. Because of this fact, the actual infiltration capacity does not decay as given by Horton's equation. The reason, as indicated above, is the Horton's model assumption of water supply always exceeding the infiltration capacity from the beginning. Hence, the soil has more infiltration capacity and we have to compute the real infiltration at t=20 min, so we need to determine the ponding time t_p by solving (7.22):

$$F = \int_0^{t_p} R(t)dt = f_c t_p + \frac{f_0 - fc}{k}\left(1 - e^{-kt_p}\right) \tag{7.22}$$

where F stands for the cumulative infiltration (that is equal to the rainfall volume) until this ponding time.

The above equation needs to be solved by an iterative procedure, for instance the Newton-Raphson method. Thus, the infiltration capacity is now a function of the actual infiltrated water, not just a function of time. Finally, the real infiltration capacity at t=20 min is calculated by evaluating (7.19) at t_p:

$$f_p = f_c + (f_0 - f_c)\, e^{-kt_p} \tag{7.23}$$

When rainfall intensity exceeds the soil infiltration capacity, the real infiltration rate is equal to this capacity and decays following Horton's equation by replacing $fc = fp$ and $t = t - t'$, being t' at which the rainfall intensity exceeds the soil infiltration capacity:

$$f = f_c + (f_p - f_c)\, e^{-k\left(t - t'\right)} \tag{7.24}$$

An additional consideration must be taken into account. It is possible that the recalculated infiltration capacity will be greater than the rainfall intensity. This implies a non-physical situation with negative storage or run-off. The reason for this behaviour is that the soil cannot infiltrate more than the rainfall rate, so a limit in the recalculated infiltration capacity must be imposed:

$$f_p \leq R \tag{7.25}$$

7.5.2.2 Green-Ampt Infiltration Model

The infiltration Green-Ampt model is a simple model with a theoretical base on Darcy's law, so it is not strictly empirical. Moreover, its parameters have physical meaning and they can be computed from soil properties. The most common soil parameters are shown in Table 7.8, as presented in (Rawls & Brakensiek 1983).

Table 7.8 – Mean values and standard deviation for Green-Ampt model parameters. Source: Rawls et al. (1983).

SOIL TYPE	θ_s	S_f **(mm/hr)**	K_s **(mm/hr)**
Sand	0.437(0.374-0.500)	4.95(0.97-25.36)	the78
Loamy sand	0.437(0.363-0.506)	6.13(1.35-27.94)	2.99
Sandy loam	0.453(0.351-0.565)	11.01(2.67-45.47)	1.09
Loam	0.463(0.375-0.551)	8.89(1.33-59.38)	0.66
Silt loam	0.501(0.420-0.582)	16.68(2.92-95.39)	0.34
Sandy clay loam	0.398(0.332-0.464)	21.85(4.42-108.0)	0.15
Clay loam	0.464(0.409-0.519)	20.88(4.79-91.10)	0.10
Silty clay loam	0.471(0.418-0.524)	27.30(5.67-131.50)	0.10
Sandy clay	0.430(0.370-0.490)	23.90(4.08-140.2)	0.06
Silty clay	0.479(0.425-0.533)	29.22(6.13-139.4)	0.05

The original Green-Ampt model starts from the assumption that a ponding depth h is maintained over the surface. The Green-Ampt method approximates the soil infiltration capacity as follows:

$$f_p = K_s + \frac{K_s\left(\theta_s - \theta_i\right) S_f}{F} \tag{7.26}$$

being K_s the effective hydraulic conductivity, S_f the suction head at the wetting front, θ_i the initial uniform water content and θ_s the porosity. The integration of (7.26) provides the cumulative infiltration:

$$f_p = \frac{dF}{dt} \Longrightarrow K_s t = F - \left(\theta_s - \theta_i\right) S_f ln\left[1 + \frac{F}{\left(\theta_s - \theta_i\right) S_f}\right] \tag{7.27}$$

Solving for the cumulative infiltration F in equation (7.27) requires an iteration procedure (e.g. Picard iterations or Newton-Rhapson method). The effective suction head can be replaced by the average value Ψ (Mein & Larson 1973).

Equations (7.26) and (7.27) assume that the soil is ponded from the beginning. Additional considerations should be taken into account in order to model an unsteady storm pattern (Te Chow, Maidment & Mays 1988). Three posibilities can occur in every timestep: 1) ponding occurs at the beginning of the interval; 2) there is no ponding within the interval; 3) ponding occurs within the interval. The first step consists of computing the actual infiltration capacity f_p from the known value of the cumulative infiltration F at time t. From (7.26):

$$f_p = K_s\left(\frac{\Psi\Delta\theta}{F} + 1\right) \tag{7.28}$$

The result from eq. (7.28) is compared with the rainfall intensity i. If $f_p \leq i$, case 1 occurs and the cumulative infiltration at the end of the interval is given by (7.29). Moreover, the real infiltration f rate will be equal to the potential one $f_p \leq i$:

$$F_{t+\Delta t} - F - \Psi\Delta\theta ln\left(\frac{F_{t+\Delta t} + \Psi\Delta}{F + \Psi\Delta}\right) = K\Delta\theta \tag{7.29}$$

If $f_p > i$, there is no ponding at the beginning of the interval. We assume that there is no ponding during the entire interval, so the real infiltration rate is equal to the rain rate and a tentative value for the cumulative infiltration at the end of the period can be computed as:

$$F'_{t+\Delta t} = F + i\Delta t. \tag{7.30}$$

From equations (7.28) and (7.30) a tentative infiltration capacity $f'_{p,t+\Delta t}$ can be calculated. If $f'_{p,t+\Delta t} > i$, there is no ponding during the interval, the assumption is correct and the problem corresponds to situation number 2, so $F'_{t+\Delta t} = F_{t+\Delta t}$. If $f'_{p,t+\Delta t} \leq i$, there are ponding condition within the interval (case 3). The cumulative infiltration at ponding time F_p is found by taking $f_p = i$ and $F = F_p$ at (7.28):

$$F_p = \frac{K_s \Psi \Delta\theta}{i - K_s} \tag{7.31}$$

Then, the ponding time is computed as $t + \Delta t'$, where:

$$\Delta t' = \frac{F_p - F}{i} \tag{7.32}$$

Finally, the cumulative infiltration can be found by replacing $F = F_p$ and $\Delta t = \Delta t - \Delta t'$ in equation (7.29).

7.5.2.3 SCS-CN Runoff Model

The Soil Conservation Service-Curve Number (SCS-CN) runoff model was originally developed by the USDA Natural Resources Conservation Service for estimating runoff from rainfall events on agricultural watersheds (USDA 1986). Nowadays it is also used for urban hydrology. The main parameter of the method is the Curve Number (CN) which is essentially a coefficient for reducing the total precipitation to runoff or surface water potential, by taking into account the losses (evaporation, absorption, transpiration and surface storage). In general terms, the higher the CN value the higher the runoff potential. The main equation of the SCS-CN method is:

$$RO = \begin{cases} \frac{(RV - I_a)^2}{RV - I_a + S} & (RV > I_a) \\ 0 & (RV \leq I_a) \end{cases} \tag{7.33}$$

where RV=Runoff of effective precipitation, RV=Rain Volume, I_a=Initial abstraction which infiltrates before runoff begins, and S=Potential maximum retention, estimated by means of the Curve Number:

$$S = \frac{25400}{CN} - 254 \tag{7.34}$$

As seen on Eq. 7.33, runoff cannot begin until initial abstraction has been met. Initial abstraction is written in terms of its ratio α and S:

$$I_a = \alpha S \tag{7.35}$$

The initial abstraction ratio was traditionally taken as $\alpha = 0.2$ for every watershed (USDA 1986) but recent studies suggest that there is a wide range of values that work better than this value, depending on the soil properties. The influence of this parameter was widely studied in Caviedes et al. (Caviedes-Voullième, García-Navarro & Murillo 2012).

It is important to remark that SCS-CN method was not designed to consider time. Following (Caviedes-Voullième et al. 2012), when the method is implemented in a complex simulator, a time-advancing methodology is used. The method is not applied to the entire catchment. Runoff is

calculated for every cell in every time step, using the cumulative rainfall since the beginning of the storm.

Follow the guidelines provided in (USDA 1986) to determine appropriate Curve Numbers.

Another important fact that needs to be considered is the Antecedent Moisture Content (AMC) or Antecedent Runoff Condition (ARC), which represents the preceding relative moisture of the soil prior to the storm event (Te Chow et al. 1988). These parameters result useful in order to attempt to account for the CN variation among storm events. Three posible assumptions can be considered: dry conditions (AMC I), average conditions (AMC II) or wet conditions (AMC III) (see Table 7.9).

Table 7.9 – Antecedent Moisture Content groups (adaptation from Mishra et al. (2003))

SOIL AMC	Total 5-day rainfall (dormant season)	Total 5-day rainfall (growing season)
I	Less than 13 mm	Less than 36 mm
II	13 mm to 28 mm	36 mm to 53 mm
III	More than 28 mm	More than 53 mm

Traditionally (Te Chow et al. 1988), the Curve Number for dry or wet conditions has been recalculated in terms of the standard conditions according to Eqs. 7.36 and 7.37:

$$CN(I) = \frac{4.2CN(II)}{10 - 0.058CN(II)} \tag{7.36}$$

$$CN(III) = \frac{23CN(II)}{10 - 0.13CN(II)} \tag{7.37}$$

On the other hand, some newer references (USDA 2004, Mishra & Singh 2003) recommend to use an empirical data table to compute both values.

7.5.3 Entering Rainfall, Evaporation and Infiltration Data

To enter data for a simulation considering rainfall, evaporation and infiltration simulation use the *Rainfall/Infiltration* panel in the RiverFlow2D Data Input Program.

You may use this component to make hydrologic simulations with the following options:

- Rainfall and evaporation and impermeable soil (no infiltration)
- Infiltration (no rainfall)
- Rainfall, evaporation and infiltration.

In the infiltration without rainfall option only the Horton or the Green and Amp methods can be used since they take into account the water depth to determine the infiltration rates. The SCS-CN method calculates infiltration as a function of the given rainfall and does not consider the pounded water.

Please follow the Rainfall, Evaporation and Infiltration Tutorial that explains how to setup a hydrologic simulation from start to finish.

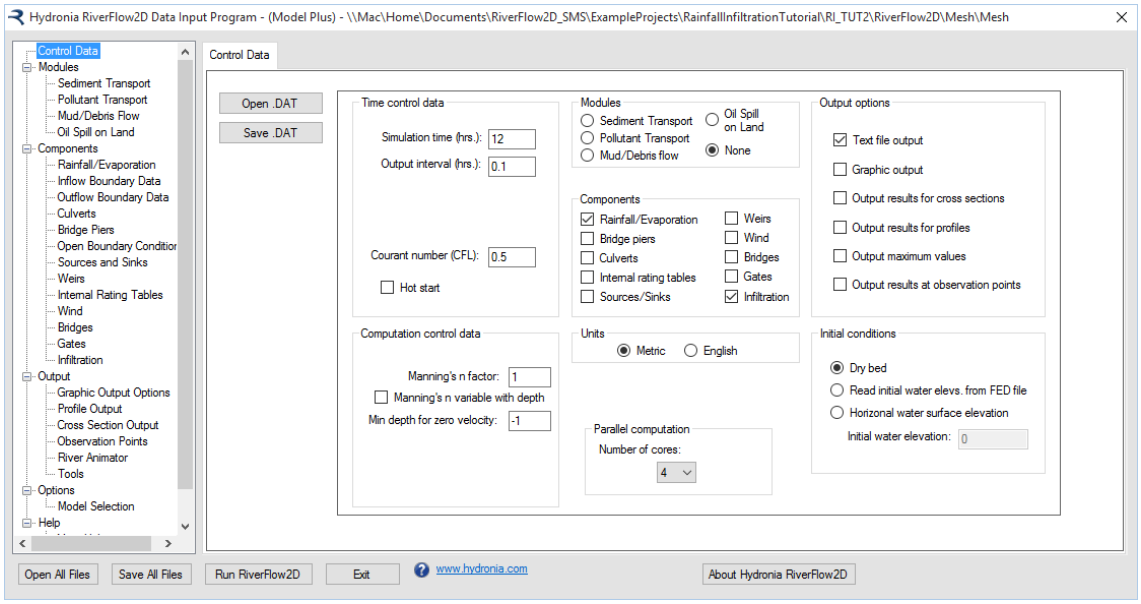


Figure 7.27 – RiverFlow2D Rainfall/Infiltration panel.

7.6 Wind Component

The wind stress is implemented by means of an extra source term included in the full system of equations that only affects the momentum components. Therefore the vector of source terms (7.2) is rewritten as:

$$\mathbf{S} = (0,\ gh(S_{0x} - S_{fx}) + S_{wx},\ gh(S_{0y} - S_{fy}) + S_{wy})^T \tag{7.38}$$

where

$$S_{wx} = C_a \frac{\rho_a}{\rho_w} \mathcal{U}|\boldsymbol{\mathcal{U}}| \qquad S_{wy} = C_a \frac{\rho_a}{\rho_w} \mathcal{V}|\boldsymbol{\mathcal{U}}| \tag{7.39}$$

being $\boldsymbol{\mathcal{U}} = (\mathcal{U}, \mathcal{V})$ the wind velocity, ρ_a and ρ_w the air and water densities respectively, and C_d is the wind stress coefficient. This source term is discretized using the same upwind technique as the friction and bed slope source terms.

7.7 Internal Rating Tables

Internal Rating Tables is an internal condition along a polyline where the model imposes the interpolated water elevation from the calculated discharge from a user provided rating table.

If the rating table is not fully compatible with the computed 2D flow, results can be erroneous. It is suggested to use this condition with care to avoid inconsistencies.

To run a simulation with Internal Rating Tables, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Programas shown in Figure 7.28.

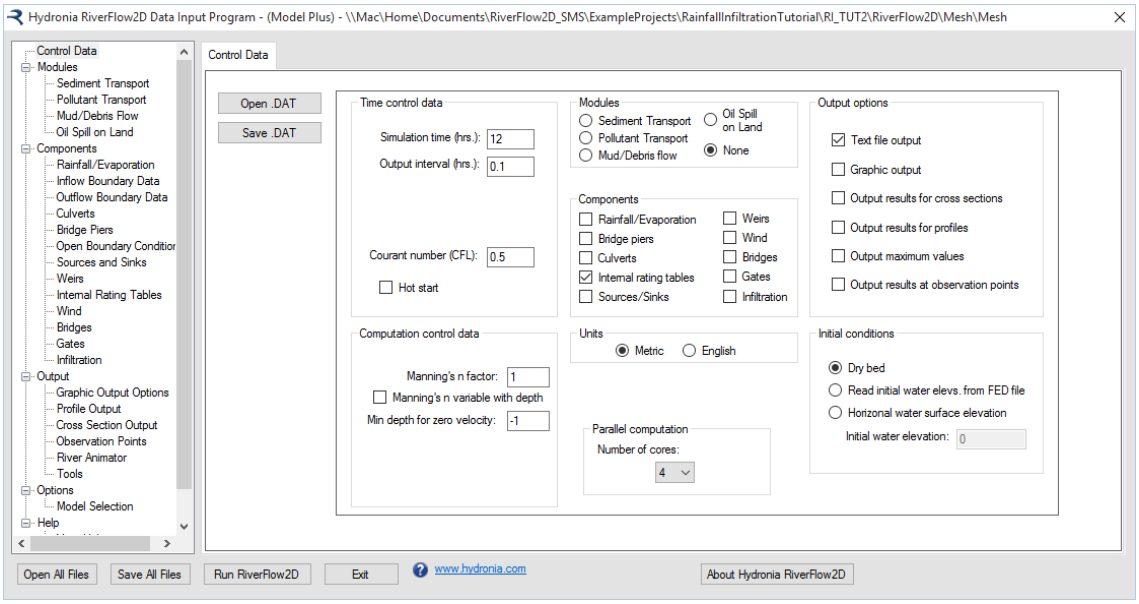


Figure 7.28 – RiverFlow2D *Control Data* panel with IRT component selected.

Internal Rating Table (IRT) plan data is entered in the SMS conceptual model coverage. To create a IRT polyline, draw a polyline clicking on each vertex until you reach the desired end when you can double click to complete the polyline.

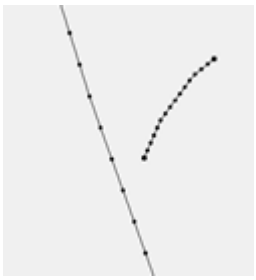


Figure 7.29 – Internal Rating Table polyline.

Then select the polyline using the Select Feature Arch icon, and right click to redistribute the vertices on the polyline. While the polyline is selected, right click on the arc and select Attributes... and on the *Feature arc Attributes* dialog, select Boundary conditions and click Options...

On the Nodestring *Boundary Conditions* dialog, select the Interior Boundary Condition component and enter the desired IRT ID (any text), and IRT file name that contains of water elevation vs discharge table.

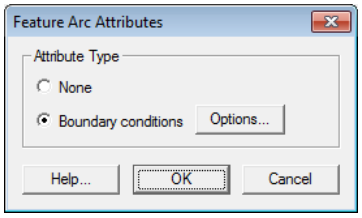


Figure 7.30 – Feature arc attributes.

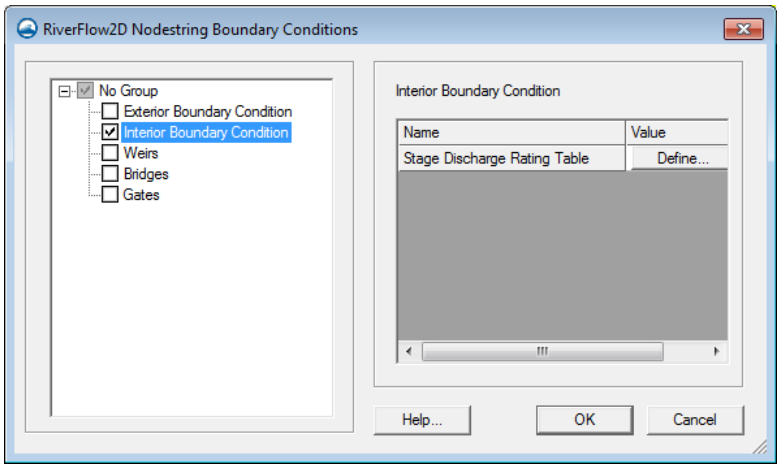


Figure 7.31 – *Internal Rating Table* dialog.

Enter the desired IRT (any text), and click Define... to enter the rating table. The following dialog pops up and the Internal Rating Data (Discharge vs. Water Surface Elevation) can be entered.

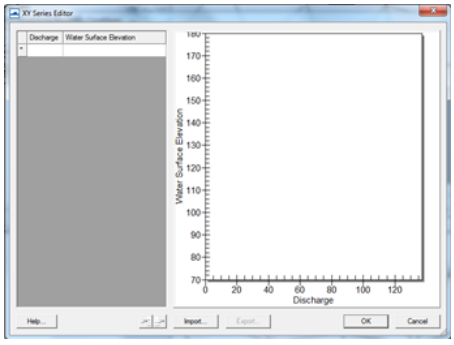


Figure 7.32 – Window to enter data pairs (discharge-WSE) for an Internal Rating Table.

Since the IRT must pass through nodes, it is essential that mesh generation engine creates nodes along the IRT nodestring. This is easily done recreating the mesh using the *Feature Objects/Map → 2D Mesh* command. The resulting mesh nodes line along the IRT polyline as shown in Figure 7.33.

Note: There is no limit to the number of Internal Rating Tables that can be used.

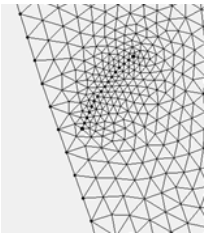


Figure 7.33 – *Internal Rating Table* dialog.

7.7.1 Internal Rating Table Calculations

An internal rating table is implemented as a set of values of total discharge in terms of the water surface level $Q = Q(h + z)$. This table is defined along a polyline in the mesh. First, a common average water surface level is computed considering all the upstream cells along the polyline. Then, the discharge is imposed at the cells sharing the edges on both sides in the polyline according to the common upstream water surface level and following the internal rating table.
The IRT calculation algorithm works as follows:

1. For each calculation time interval, estimate an average water surface level at each side of the IRT polyline.
2. Compute the discharge passing through the IRT polyline from the average water levels in 1 using the rating table.
3. Define an average velocity from the discharge and the cross sectional wetted area.
4. Assign a common unit discharge to every pair of cells sharing a polyline segment.

Some inappropriate IRT polyline configurations or very long polylines can over-constrain the model and should be avoided.

7.7.2 Assumptions of Internal Rating Table Calculations

- The rating table does not account for outlet control.

7.8 Sources and Sinks

Sources and Sinks component allows accounting point inflows (source) or outflows (sink) of water on the mesh. This allows simulating for example water intakes at any location on the mesh.

To run a simulation with Sources or Sinks, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Program as shown in Figure 7.34.

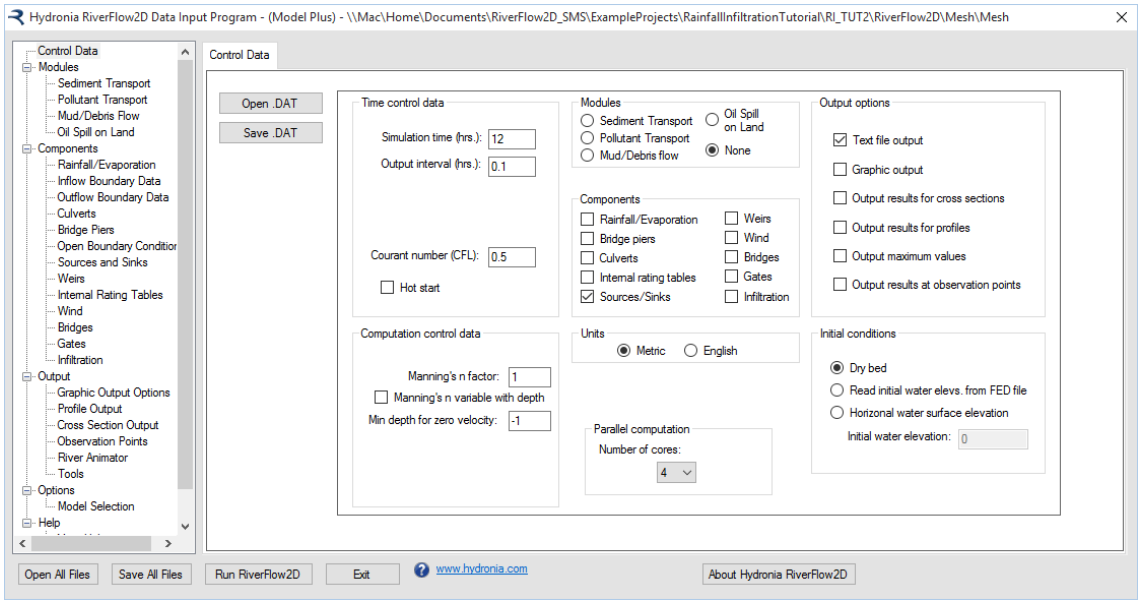
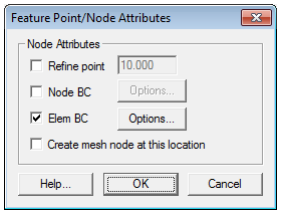


Figure 7.34 – RiverFlow2D *Control Data* panel with the Sources and Sinks component selected.

Sources and Sinks data is entered in the SMS conceptual model coverage. To create a Source/Sink follow these steps:

- Then, click on the *Create Feature Point* icon and click on the location where you want to enter the source
- Now click on the *Select Feature Point* icon and click on the point entered in the previous step
- On the *Feature Objects* menu, click *Attributes...*
- Select Elem BC and click *Options...* button:



- Select Source/Sink, enter the Source/Sink ID name (e.g. SOURCE1) and click Define...

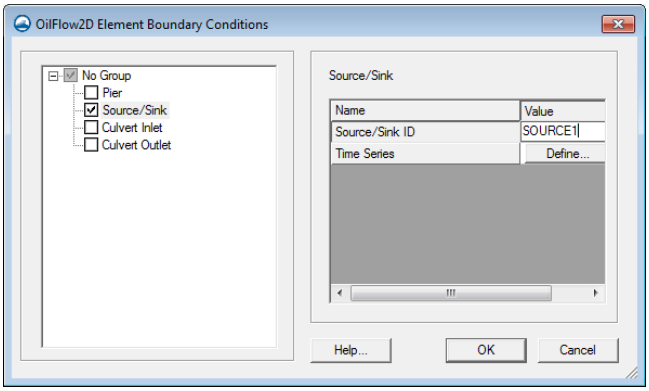


Figure 7.35 – Source/Sink dialog.

- Enter the discharge time series (unsteady) or a single discharge (steady flow).

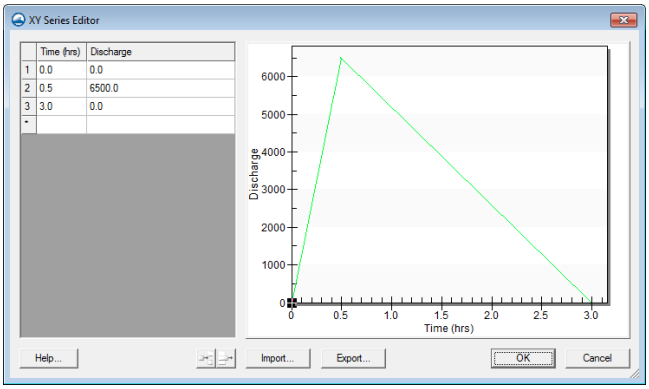


Figure 7.36 – Time series editor to enter discharge for a Source/Sink.

Note: There is no limit to the number of sources and sinks that can be used.

7.9 Weirs

RiverFlow2D Weirs component may be convenient when trying to simulate levee or road overtopping. The tool allows defining a polyline representing the structure alignment and assigning crest elevations that can vary along the polyline.

To run a simulation with weirs, you need to select the option in the *Control Data* panel of the RiverFlow2D Data Input Program as shown in Figure 7.37.

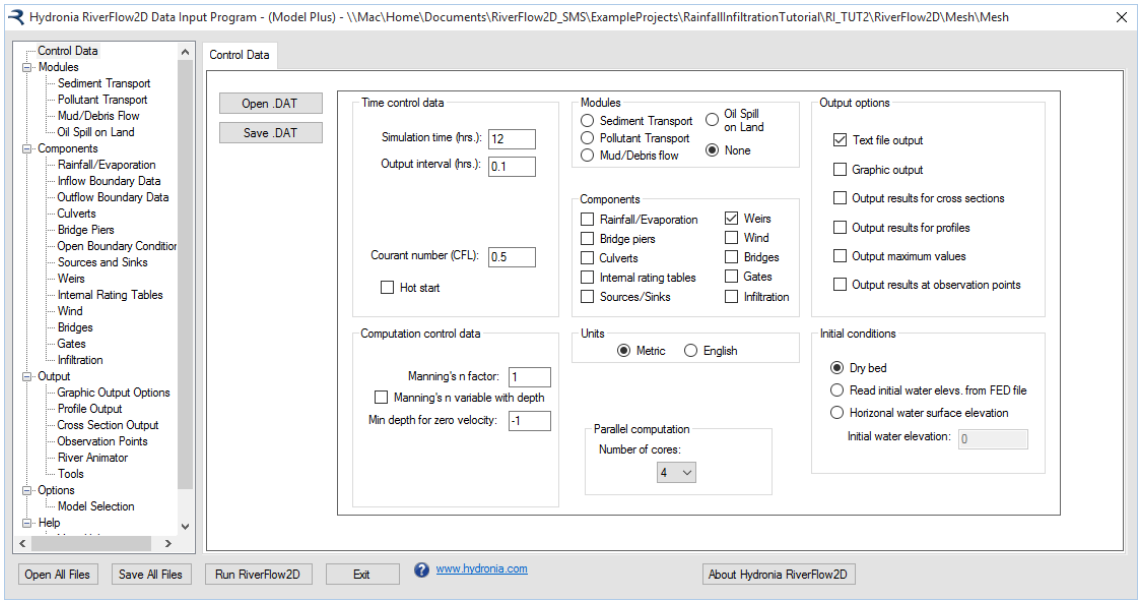


Figure 7.37 – RiverFlow2D Control Data panel with Weirs component selected.

Weir plan data is entered in the SMS conceptual model coverage. To create a weir, first make sure that the conceptual model coverage where you defined the domain outline is active. Then, create an arc that will define the weir geometry in plan. Now, draw a polyline clicking on each vertex until you reach the desired end when you can double click to complete the polyline.

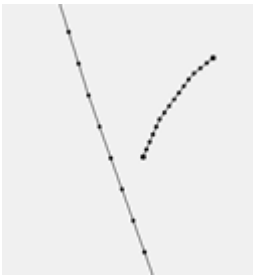


Figure 7.38 – Weir polyline.

Then select the polyline using the Select Feature Arch icon, and right click to redistribute the vertices on the polyline. While the polyline is selected, right click on the arc and select Attributes... and on the *Feature arc Attributes* dialog, select *Boundary conditions* and click *Options...*

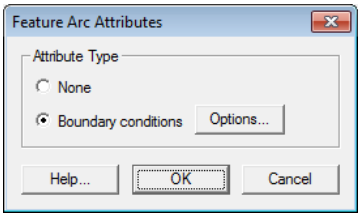


Figure 7.39 – Feature arc attributes.

On the Nodestring *Boundary Conditions* dialog, select the Interior Boundary Condition component and enter the desired Weir name, crest elevation and weir coefficient C_f. This version of the software considers variable crest elevation along the weir.

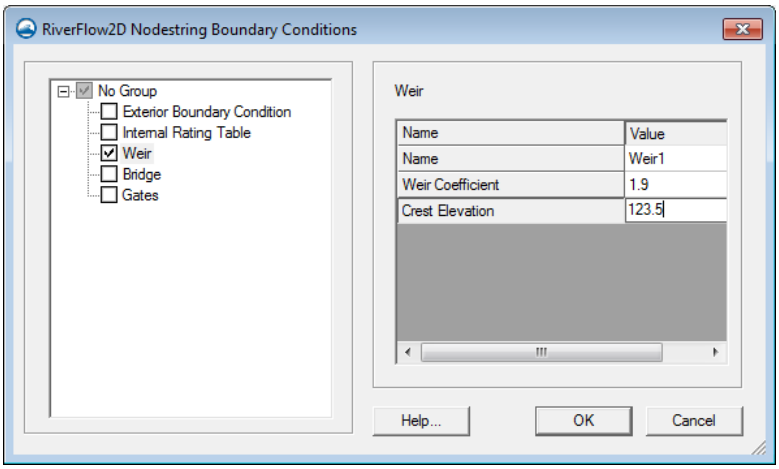


Figure 7.40 – Weirs data dialog.

Since RiverFlow2D requires that the weir passes through nodes, it is essential that mesh generation engine creates nodes along the weir polyline. To achieve this it is necessary that each weir polyline coincides exactly with corresponding arcs on SMS Conceptual Model.

Note: There is no limit to the number of weirs that can be used.

7.9.1 Weir Calculations

The weir calculation algorithm works as follows:

1. For each calculation time interval, the model checks for each segment defined by two pair of opposing cells (L, R) along the weir that at least one of the opposite cells is wet and that its water surface elevation is above the crest elevation.

2. Then the model calculates the water elevation at each weir segment as:

$$d_w = h_{crest} + MAX\left(z_L, z_R\right) \tag{7.40}$$

where h_{crest} is the crest elevation and d_w the segment water elevation.

3. When the water surface levels on both sides is below the weir level, $MAX\,(d_L, d_R) \leq d_w$, the velocity component normal to the weir segment direction is set to zero.

4. Otherwise the model calculates the normal discharge for the segment according to the water levels on both sides.

5. The discharge is imposed on both the and cells.

The weir is simulated by assuming that the discharge per unit breadth q crossing the weir is governed by the difference between the water surface level ($d = h + z$) on both sides of the weir, referred to as d_l upstream and d_r downstream of the weir, and by the weir crest elevation, H_w. Several situations are accounted for. In the case that both water elevations are below the weir crest elevation the weir behaves as a solid wall and no flow crosses it. When $d_l < d_r$, without loss of generality, two different flow situations can occur depending on the relative values of H_w, z_l, z_r, d_l and d_r. When $H_w + max\,(z_l, z_r) < min\,(d_l, d_r)$, the discharge is given by

$$q = C_f\,(d_r - d_l)^{3/2} \tag{7.41}$$

with C_f the discharge coefficient.

When $H_w + max\,(z_l, z_r) > min\,(d_l, d_r)$, the discharge is given by

$$q = C_f\,(d_r - H_w)^{3/2} \tag{7.42}$$

7.9.2 Assumptions of Weir Calculations

1. The weir crest elevation may vary along the weir but must be higher than both cells opposing each weir segment.

7.10 Dam Breach Modeling

The RiverFlow2D Dam Breach component provides a way to simulate a gradual breach of internal linear obstructions such as dams, levees, etc. The obstruction is entered as an arbitrary polyline and is considered a barrier to the flowing water that restricts, directs or slows down the flow, often creating water ponding upstream.

In RiverFlow2D, the dam is defined as an internal boundary condition and modeled as a progressive trapezoid. For a complete parametrization of the breach, the next parameters and variables are used (see Figure 7.41):

- Coordinates (x,y) of the center of the breach, assuming $z = z_{crest}$, where z_{crest} is the initial dam z-coordinate.
- Value of material angle α (assumed constant).
- Table (t, $b(t)$, $H_b(t)$), being t =time, b =lower breach width, H_b =breach height.

As interenting particular cases, $b(t) = 0$ means the generation of a triangular breach. and $\alpha = 0$ means the generation of a rectangular breach.

In general, the total discharge through the breach will be calculated with a law of the type:

$$Q_b = KB(H)^{3/2} \tag{7.43}$$

where $H = h + z - H_v$, $H_v = z_{crest} - H_b$ and $B(H) = b(t) + 2\frac{H(t)}{tg\alpha}$.

The discharge computed in (7.43) will be distributed among the cells included in the breach top lenght $B(H_b)$:

$$B(H_b) = b(t) + 2\frac{H_b(t)}{tg\alpha} \tag{7.44}$$

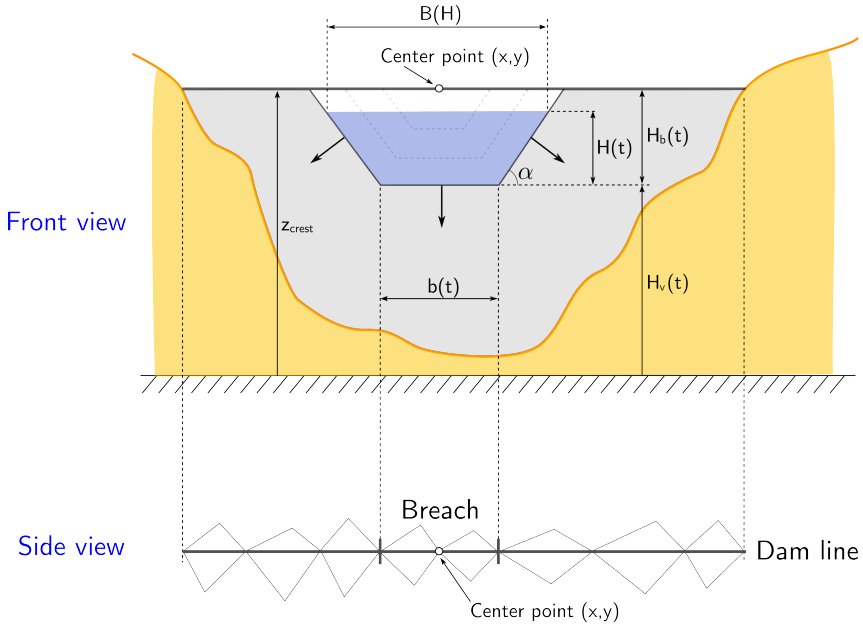


Figure 7.41 – Coordinate system for the dam breach representation.

8

RiverFlow2D Data Input Program (DIP)

Data Input Program (DIP) user interface provides the tools to enter non-spatial data, and run RiverFlow2D model. All changes introduced on the DIP will be saved to the native RiverFlow2D data files not to the `.2dm` file.

The RiverFlow2D DIP appears when you use the *Run RiverFlow2D* command from SMS *RiverFlow2D* menu. You can also access the DIP double-clicking on the RiverFlow2D icon on the desktop. In that case, the program will give you a list of previous projects and let you open any one of them:

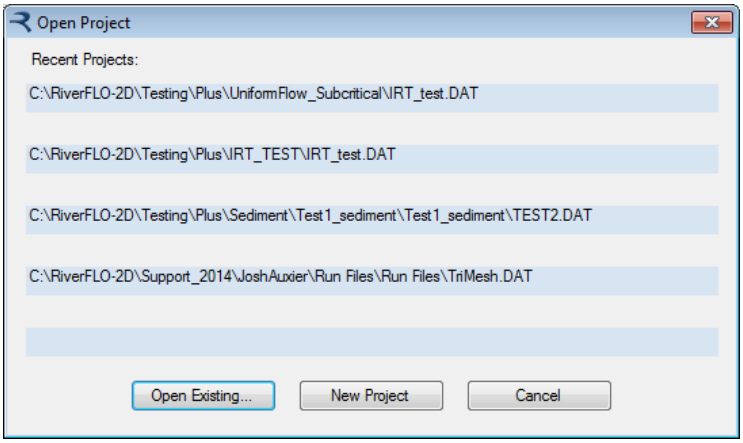


Figure 8.1 – RiverFlow2D DIP Open Project Dialog.

RiverFlow2D DIP provides a data input environment with panels that include all the non-spatial options to run the RiverFlow2D model. The left column on the main window allows you select modules, components, output options, etc. When you click on one of the elements, the appropriate right side panel is activated. Each panel contains the data corresponding to each of the RiverFlow2D data files. For example, the Control Data Panel has all the data of the `.DAT` file.

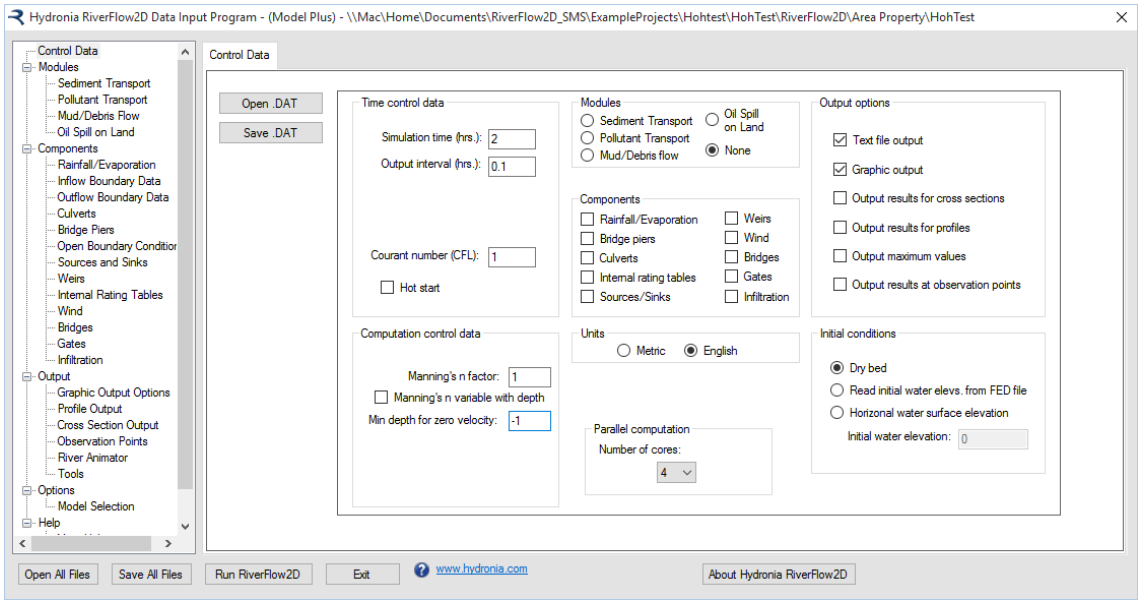


Figure 8.2 – Main RiverFlow2D Data Input Program window.

RiverFlow2D DIP lets you select different model engines. Clicking on Model Selection opens the dialog where you can select between RiverFlow2D Plus GPU, RiverFlow2D Plus or RiverFlow2D FE. Note that to run the GPU version you need a special license as well as the appropriate GPU hardware.

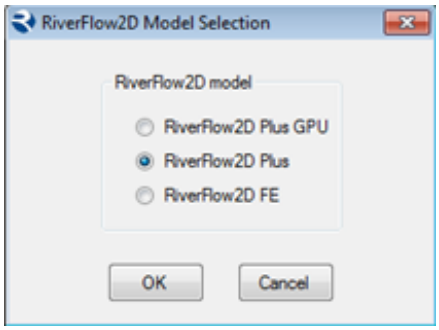


Figure 8.3 – RiverFlow2D Model Selection Dialog.

The following sections describe the panel dialogs of RiverFlow2D DIP.

8.1 Control Data Panel (.DAT files)

This panel determines the general run options like time step control parameters, metric or English units, physical process (components), graphical outputs, and initial conditions. It also provides buttons to open and saving files, and running the RiverFlow2D model. The program will launch with the Control Data panel visible.

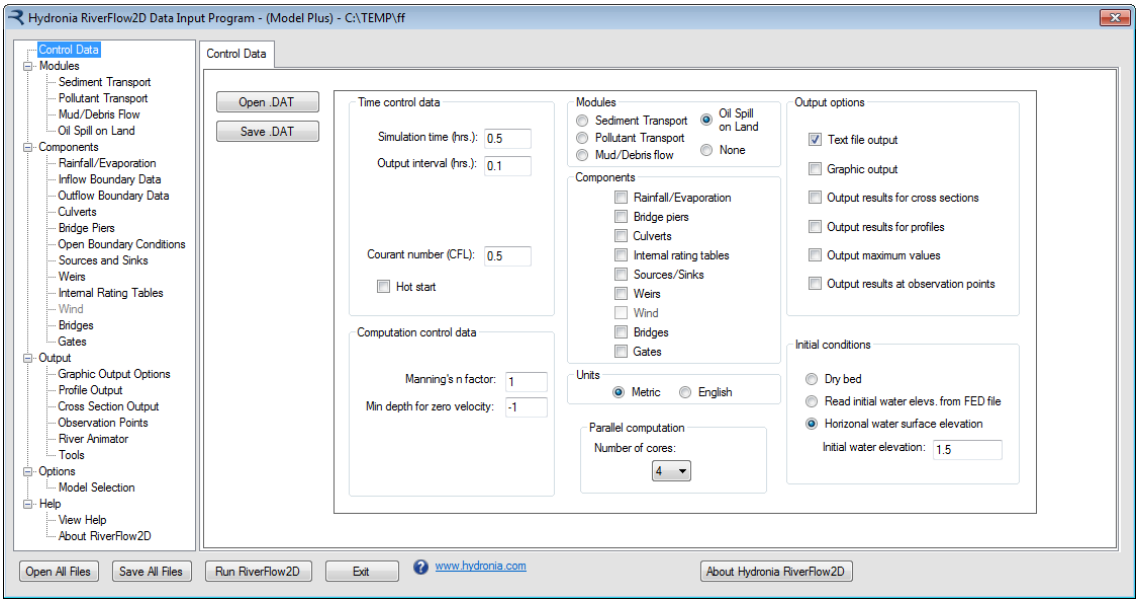


Figure 8.4 – Control Data panel.

Table 8.1 – Buttons in the Control Data Panel.

BUTTON NAME	DESCRIPTION
Open .DAT	Opens an existing `.DAT` file.
Save .DAT	Saves a `.DAT` file with the data shown on the Panel.
Buttons on Bottom Row Duplicated on Every Panel.	
Open All Files	Saves data from all enabled Tabs. *Note: This operation does not alter the node coordinates and elevations, triangular mesh topology, Manning roughness coefficients, and other mesh related parameters in the `.FED` file.*
Run RiverFlow2D	Runs the RiverFlow2D model.
Exit	Closes the RiverFlow2D Data Input Program.
About Hydronia RiverFlow2D	Shows a concise description of RiverFlow2D.
?	Opens the RiverFlow2D User's Guide.
www.hydronia.com	Opens browser to Hydronia home page.

Table 8.2 – Time Control Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Simulation time (hrs.)	Total simulation time in hours.
Output Interval (hrs.)	Time interval for output reporting.
CFL	Courant-Friederich-Lewy condition (CFL). Set this number to a value in the (0,1] interval. By default CFL is set to 1.0 which is the recommended value for maximum performance. A few rare applications may require reducing CFL to 0.5 or so avoid model oscillations in the model results.
Hot start	Use this option to restart the model from a previously simulation.

Table 8.3 – Units Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Metric	Select this option to work in metric units. Coordinates are given in meters, velocities in m/s, discharge in m^3/s, etc. Text output is provided in metric units.
English	Select this option to work in English units. Coordinates are given in feet, velocities in ft/s, discharge in ft^3/s, etc. Text output is provided in English units.

*Note: Changing the units on this panel does **not** change the contents of the data files.*

Table 8.4 – Computation Control Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Manning's n factor	Use this factor (XNMAN) to test the sensitivity of results to the Manning's n. Each element Manning's n-value will be multiplied by XNMAN. Over several simulations, using this factor may speed up the model calibration. Default is XNMAN = 1.
Manning's n variable with depth	Select this option to set Manning's n as a function of depth. The user must enter polygons over the mesh and each polygon should have an associated file containing the depth vs Manning's n table.
Minimum depth for dry bed	This parameter indicates the depth below which cell velocity will be assumed 0. By default it is set to -1 which will allow the model to dynamically set the dry cell conditions for depths smaller than 10^{-6}m.

Table 8.5 – Closed Boundary Conditions Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION

Table 8.6 – Output Options Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Text file output	Reports results to ASCII text output files.

Continued on next page

Table 8.6 – continued from previous page

CONTROL NAME	DESCRIPTION
Graphic output	Activates plot simulation results while program runs and reporting to graphic files.
Output results for cross sections	Use this option to generate results for user defined cross sections. The cross section can be edited in the Cross Section Output Panel. This data goes in a `.XSECS` file.
Output results for profiles	Use this option to generate results along a user defined polyline. The polyline data can be edited in the Profile Cut Output Panel. This data goes in a `.PROFILES` file.
Output maximum values	Switch to allow reporting maximum values throughout the simulation to `.MAXI`, `.MAXE`, and maximum value `.EXP` output files. In the RiverFlow2D Plus model maximum values will only be generated using the post processing Plot RiverFlow2D results button on the Graphic Output Options panel.
Output results at observation points	Switch to allow reporting time series of results at specified locations defined by coordinates in the Observation Points Panel.

Table 8.7 – Components Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Rainfall/Evaporation	Option to activate rainfall and/or evaporation. The required data has to be entered in the Rainfall /Evaporation Panel. This data is written to `.LRAIN` file.
Bridge piers	Switch to allow accounting for pier drag force. The Bridge piers data can be edited in the Bridge Piers Panel. The data is written to `.PIERS` file.
Bridges	Switch to model Bridges using the bridge cross section geometry and accounting for energy losses. The data can be edited in the Bridges Panel. The data is written to `.BRIDGES` file.
Culverts	Switch indicating if one dimensional culverts will be used. The Culverts data can be edited in the Culverts Panel. The data is written to `.CULVERTS` file.
Gates	Switch to model gates. The data can be edited in the Gates Panel. The data is written to `.GATES` file.
Infiltration	Option to activate Infiltration loss calculations. The required data has to be entered in the Infiltration Panel. This data is written to `.LINF` file.
Internal rating tables	Switch to allow using internal rating tables. The data can be edited in the Internal Rating Tables Panel. The data is written to `.IRT` file.
Mud/Debris Flow	Option to activate the Mud and Debris Flow modeling. The data can be edited in the Mud/Debris Flow Panel. The data is written to the `.MUD` file.
Oil Spill on Land	Option to activate the Overland Oil Spill modeling. The data can be edited in the Oil Spill on Land Panel. The data is written to the `.OILP` file.
Pollutant Transport	Option to activate pollutant transport modeling. The pollutant transport data can be edited in the Pollutant Transport Panel. The data is written to the `.SOLUTES` file.
Sources/Sinks	Switch to indicate existence of sources of sinks. The sources/sinks data can be edited in the Sources/Sinks Panel. The data is written to `.SOURCES` file.
Weirs	Switch to indicate the existence of Weir computation. The Weir data can be edited in the Weirs Panel. The data is written to `.WEIRS` file.

Continued on next page

Table 8.7 – continued from previous page

CONTROL NAME	DESCRIPTION
Wind	This option activates the calculation of wind stress on the water surface. The data can be edited in the Wind Panel. The data is written to `.WIND` file.
Sediment transport	Option to activate sediment transport modeling with erosion and deposition for a mobile bed. The sediment transport data can be edited in the Sediment Transport Panel. This data is written to the `.SEDS` and `.SEDB` files.

Table 8.8 – Components Data Frame on the Control Data Panel.

CONTROL NAME	DESCRIPTION
Dry bed	The simulation will start with a fully dry bed. For discharge boundary conditions, an arbitrary depth (> 0.0) is assigned to calculate the inflow for the first time-step. Subsequently the flow depth at the boundary will be determined by the model.
Read initial water elevations from `.FED` file	Initial water surface elevations will be read from the `.FED` file. It is possible to assign a spatially variable initial water surface elevation in the Initial Conditions Layer.
Horizontal water surface elevation	Use this option to start a simulation with a user provided initial horizontal water surface elevation.
Initial water elevation	Initial water surface elevation on the whole mesh. If initial water elevation is set to -9999, the program will assign a constant water elevation equal to the highest bed elevation on the mesh.

8.2 Sediment Transport Data Panel for the RiverFlow2D Plus model (.SEDS and .SEDB Files)

This panel allows entering sediment transport data. To activate this panel, first select *Sediment Transport* on the *Components Frame* of the *Control Data Panel*.

Table 8.9 – Parameters on the Sediment transport Mode frame and buttons of the Sediment Transport Panel (RiverFlow2D Plus model).

CONTROL NAME	DESCRIPTION
Suspended sediment	When this check box is selected, the model will compute sediment concentrations using the suspended sediment transport component. See comment 1.
Bed load Sediment	Selecting this check box will activate the bed load sediment transport component. See comment 1.
Buttons	
Open `.SED*`	Opens an existing `.SEDS` or `.SEDB` files.
Save `.SED*`	Saves the sediment data to `.SEDS` and `.SEDB` files.

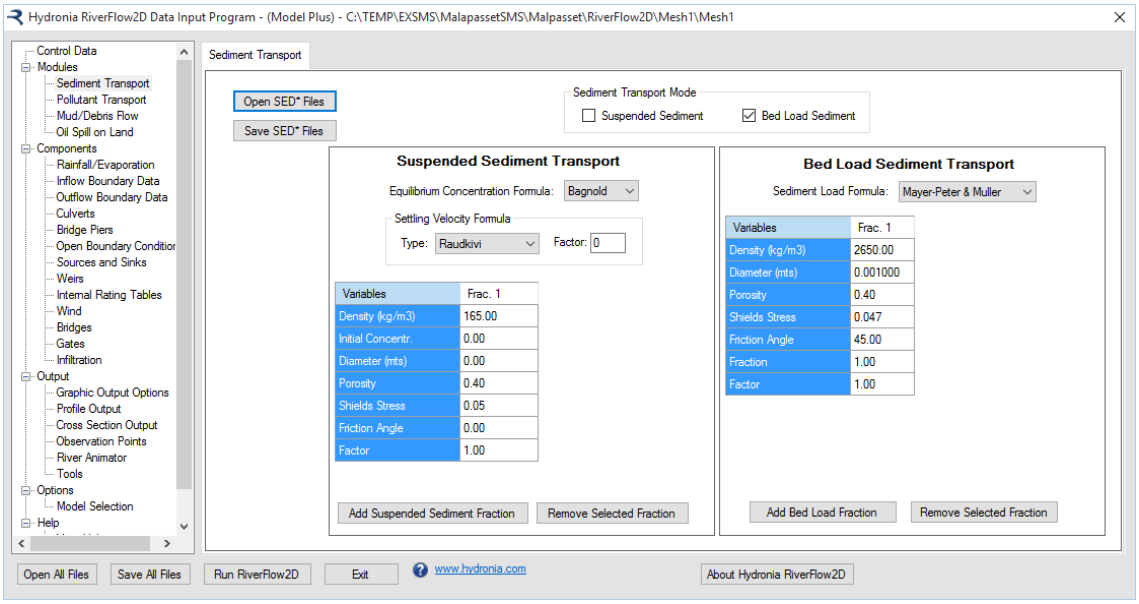


Figure 8.5 – Sediment Transport Panel (RiverFlow2D Plus model).

Table 8.10 – Parameters on the Suspended Sediment transport frame of the Sediment Transport Panel (RiverFlow2D Plus model).

CONTROL NAME	DESCRIPTION
Equilibrium Concentration formula	When this check box is selected, the model will compute sediment concentrations using one of the following suspended sediment transport formulas: 1. Bagnold (1966)lb 2. Van Rijn (1984a)
Settling Velocity Formula	It is a unique formula that applies for all fractions. This drop-down list includes the following formulas: 1. Rubey (1933) 2. Zhang (1961) 3. Zanke (1977) 4. Van Rijn (1984a) 5. Raudkivi (1990) 6. Julien (1998) 7. Cheng (1997) 8. Jimenez-Madsen (2003) 9. Wu-Wong (2006)
Factor	This factor multiplies the settling velocity calculated by the selected formula.
Table	
Density	Suspended sediment density (kg/m^3 or lb/ft^3).
Initial Concentration	Initial volumetric sediment concentration. See comment 2.
Diameter	Characteristic sediment size for this fraction (m or ft).
Porosity	Bed porosity.
Shields Stress	Critical Shield stress.
Friction Angle	Sediment friction angle (degrees).

Continued on next page

Table 8.10 – continued from previous page

CONTROL NAME	DESCRIPTION
Factor	Equilibrium concentration formula factor for each fraction. This factor multiplies the equilibrium concentration calculated by the selected formula.
Buttons	
Add Suspended Sediment Fraction	Used to add a new fraction. Up to 10 fractions may be used.
Remove Selected Fraction	Deletes the selected fraction.

Table 8.11 – Parameters on the Bed Load Sediment transport frame of the Sediment Transport Panel (RiverFlow2D Plus model).

CONTROL NAME	DESCRIPTION
Sediment load formula	Allows selection of one of the following sediment transport formulas: 1. Meyer-Peter & Muller (1948) 2. Ashida (1972) 3. Engelund (1976) 4. Fernandez (1976) 5. Parker fit to Einstein (1979) 6. Smart (1984) 7. Nielsen (1992) 8. Wong 1 (2003) 9. Wong 2 (2003) 10. Camenen-Larson (1966)
Table	
Density	Sediment density (lb/ft^3 or kg/m^3).
Diameter D30	Sediment D30 size (m). 30% of the sediment is finer than D30. Only used for Smart Formula.
Diameter	Characteristic sediment size for this fraction (m).
Diameter D90	Sediment D90 size (m). 90% of the sediment is finer than D90. Only used for Smart Formula.
Porosity	Sediment porosity.
Shields Stress	Critical Shield stress.
Friction Angle	Sediment friction angle (degrees).
Fraction	Fraction of material in bed. All fractions must add up to 1.
Factor	Transport formula factor for each fraction. This factor multiplies the result of the transport formula selected.
Buttons	
Add Bed Load Fraction	Used to add a new fraction. Up to 10 fractions may be used.
Remove Selected Fraction	Deletes the selected fraction.

8.2.0.1 Comments for the .SEDS and .SEDB Files

1. You can select either one or both options. When the suspended sediment transport options all inflow data files should contain time series of volumetric concentrations for each fraction entered.

2. Volumetric concentration should be provided as a fraction of 1. Note that the typically total maximum suspended load concentration do not exceed 0.08. Concentrations greater than 0.08 is generally considered hyperconcentrated flow which falls beyond the validity of the sediment transport algorithms. Therefore, the sum of all initial concentrations should also not exceed 0.08.

8.3 Mud and Debris Flow Data Panel for the RiverFlow2D Plus model (.MUD File)

This panel allows entering mud and debris flow data for the RiverFlow2D model. To activate this panel, first select *Mud* on the *Components Frame* of the *Control Data Panel*.

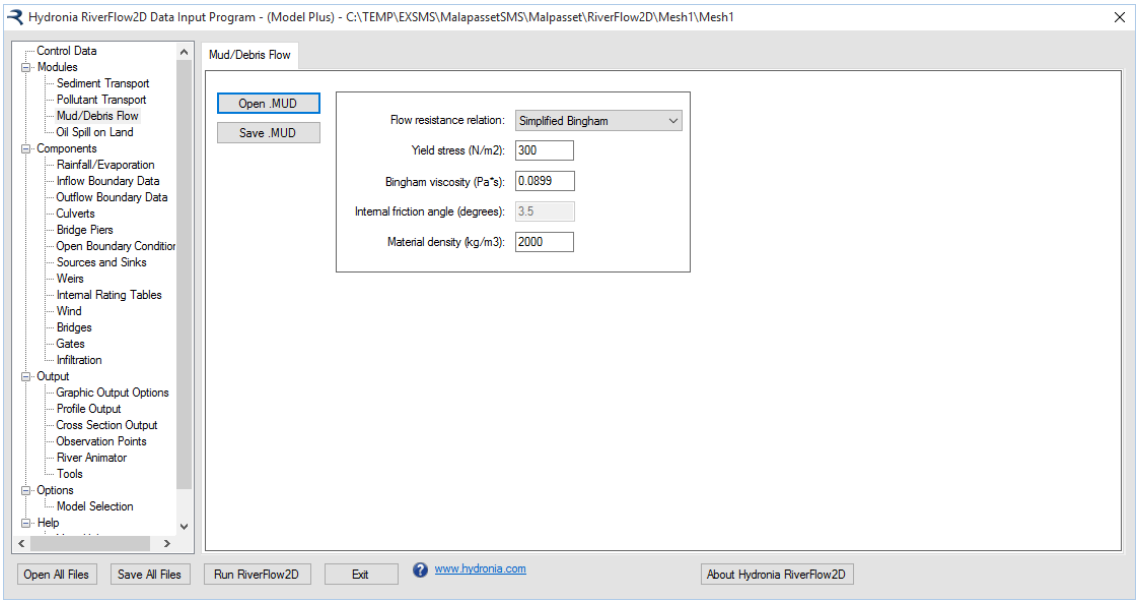


Figure 8.6 – Mud/Debris Flow Panel (RiverFlow2D Plus model).

Table 8.12 – Parameters on the Mud/Debris Flow Panel (RiverFlow2D Plus model).

CONTROL NAME	DESCRIPTION
Flow resistance relation	1. Turbulent flow 2. Full Bingham 3. Simplified Bingham 4. Turbulent and Coulomb 5. Turbulent and Yield 6. Turbulent, Coulomb and Yield 7. Quadratic 8. Granular flow
Yield stress	Yield stress (Pa or lb/in^2).
Bingham viscosity	Bingham viscosity (Poise or lb/in^2).
Internal friction angle	Internal friction angle (degrees).
Material density	Material density (kg/m^3 or lb/ft^3).

8.4 Oil Spill on Land for the OilFlow2D model (.OILP File)

This panel allows entering viscous fluid properties for the OilFlow2D model. To activate this panel, first select *Oil Spill* on Land on the *Components Frame* of the *Control Data Panel*.

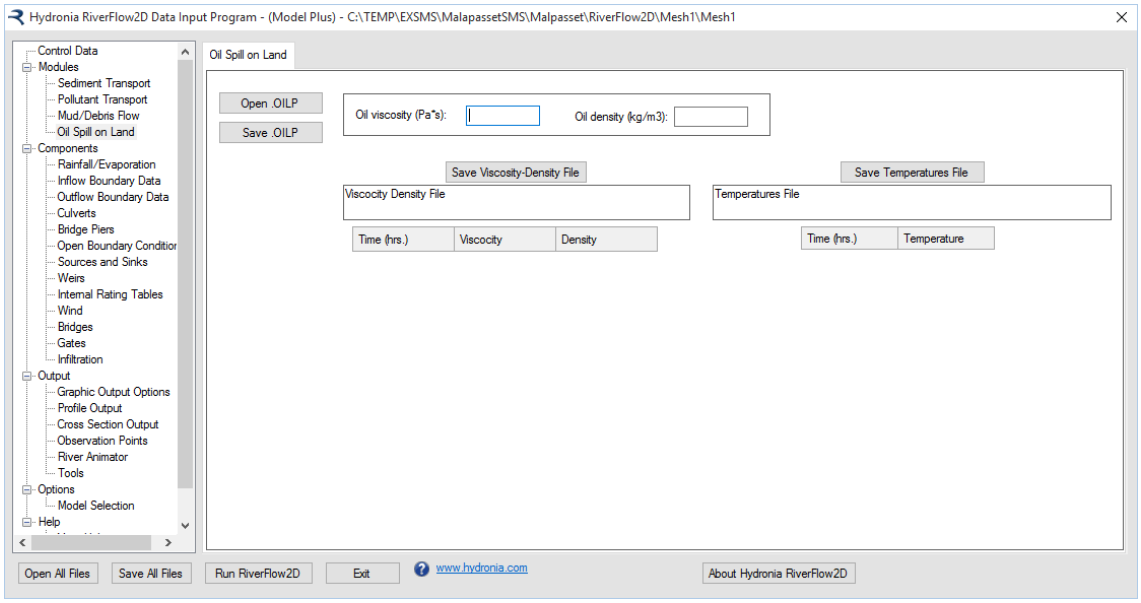


Figure 8.7 – Oil Spill on Land Panel (OilFlow2D model).

Table 8.13 – Parameters on the Oil Spill on Land Panel (OilFlow2D Plus model).

CONTROL NAME	DESCRIPTION
Fluid viscosity	Viscosity (Pascal s or lbs/in^2).
Fluid density	Density (kg/m^3 or lb/ft^3).

8.5 Graphic Output Options Tab (.PLT File)

This panel allows entering options to control RiverFlow2D output. To activate this panel, first select *Graphic Output Options* from the *Output* group on the left panel of the RiverFlow2D DIP.

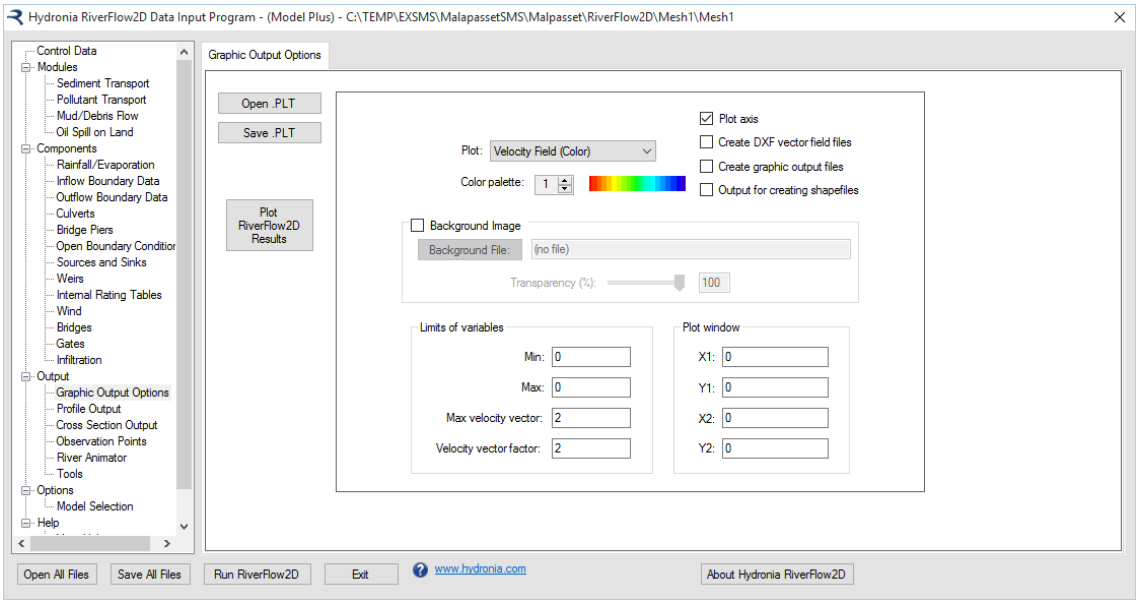


Figure 8.8 – Graphic Output Panel.

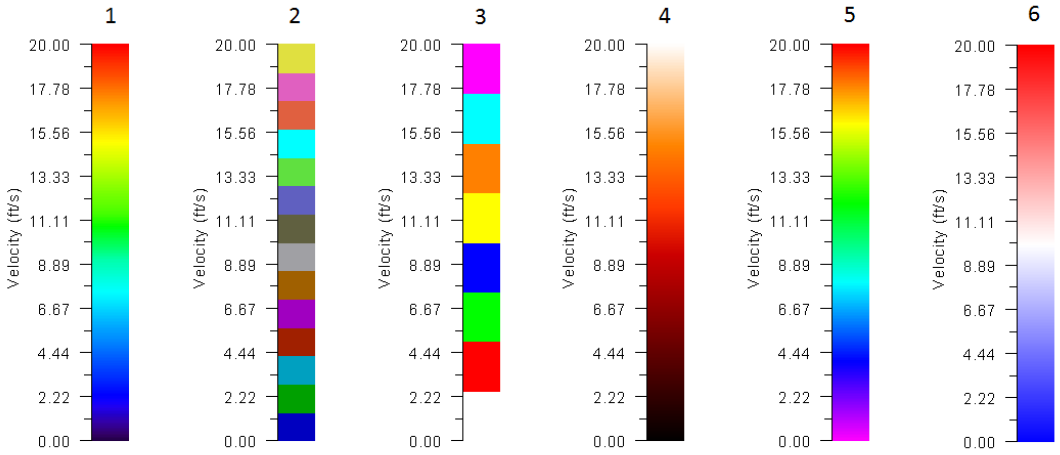


Figure 8.9 – Color Palettes.

Table 8.14 – Parameters on the Graphic Output Option Panel.

CONTROL NAME	DESCRIPTION
Buttons	
Open .PLT	Opens an existing .PLT output data file.
Save .PLT	Saves only the graphic output data to a .PLT file.
Plot RiverFlow2D Results	Use this button to create output files and visualize RiverFlow2D results from existing simulations. The output files for the simulation must be available.
Plot Frame	

Continued on next page

Table 8.14 – continued from previous page

CONTROL NAME	DESCRIPTION
Plot	Chose the desired plot from the list: • Velocity field using black arrows. • Velocity field using colored arrows based on velocity magnitude. • Velocities in black over colored depths. • Velocities in black over colored bed elevations. • Flow depth. • Bed elevation. • Water elevations. • Velocities in black over colored water elevations. • Erosion and deposition. • Concentration
Color palette	Choose between 6 color schemes (see Figure 8.9).
Plot axis	Display axes on plot.
Create DXF vector field files	Generate velocity vector DXF (CAD) files. This option will also export the mesh in DXF format to the file: `<ProjectName>_MESH.DXF`.
Create graphic output files	Generates a raster image in GIF format for each report time step.
Output for creating shapefiles	Using this option RiverFlow2D will generate support files that can be used to create ESRI Shapefiles for velocity fields, depth and water surface elevations.
Background Image Frame	
Background image	Check this if you want to import a background aerial to for dynamic plots.
Background file	Selects images for background. Supported formats include: `.BMP`, `.GIF`, `.PNG` and `.TIF`.
Transparency	Transparency of plots to allow visualizing the background image under dynamic plots. 100=full transparency, 0=full opacity.
List of variables Frame	
Min	Minimum value to plot of the current variable chosen in Plot option.
Max	Maximum value to plot of the current variable chosen in Plot option.
Max velocity vector	Maximum velocity vector to plot.
Velocity vector factor	Multiplication factor to amplify arrows in vector plots.
Plot Window Frame	Use to define zoom or area to visualize. If all values are 0, then maximum extent is plotted.
X1	X coordinates of lower left corner of rectangle.
Y1	Y coordinates of lower left corner of rectangle.
X2	X coordinate of upper right corner of rectangle.
Y2	Y coordinate of upper right corner of rectangle.

8.6 Profile Output Panel (.PROFILES File)

Use this panel to enter polyline coordinates where the model results are to be generated. RiverFlow2D will generate output `.PRFI` and `.PRFE` files. To activate this panel, first select the *Profile Output* in *Output* from the *Output* group on the left panel of the RiverFlow2D DIP.

See output file section (9.4.3) for output file content description.

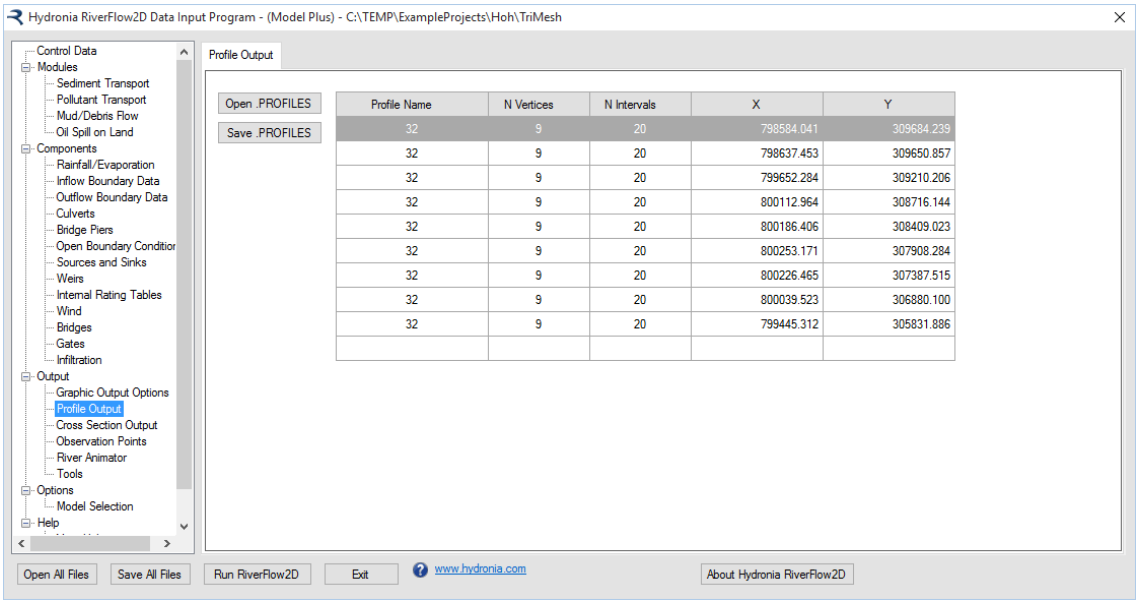


Figure 8.10 – Profile Output File.

Table 8.15 – Parameters on the Profile Output Panel.

COLUMN NAME	DESCRIPTION
Profile data table	
Profile name	Profile name. Should not contain spaces and must have less than 26 characters.
N Vertices	Number of vertices in each profile.
N Intervals	Intervals to divide each profile sub-segment between vertices. Results will be reported at each interval.
X, Y	Coordinates for each vertex in polyline.
Buttons	
Open `.PROFILES`	Opens an existing `.PROFILES` file.
Save `.PROFILES`	Saves only the profile data to a `.PROFILES` file.

8.7 Cross Section Output Panel (.XSECS File)

Use this panel to enter coordinates for cross sections that intersect the triangular element mesh where you want to output model results. RiverFlow2D will generate output `.XSECI` and `.XSECE` files. To activate this panel, first select the *Cross Section Output* from the *Output* group on the left panel of the RiverFlow2D DIP.

See output file section (4.4.1) for output file content description.

Table 8.16 – Parameters on the Inflow Boundary Data Panel.

COLUMN NAME	DESCRIPTION
XS data table	
XS Name	Cross section name. Should not contain spaces and must have less than 26 characters.
N Intervals	Intervals to divide each section. Results will be extracted and reported at each interval.
X1 Y1 X2 Y2	Each row corresponds to the coordinates of the initial (X1,Y1) and ending (X2,Y2) of one cross section.
Buttons	
Open `.XSECS`	Opens an existing `.XSECS` file.
Save `.XSECS`	Saves only the cross section data to a `.XSECS` file.

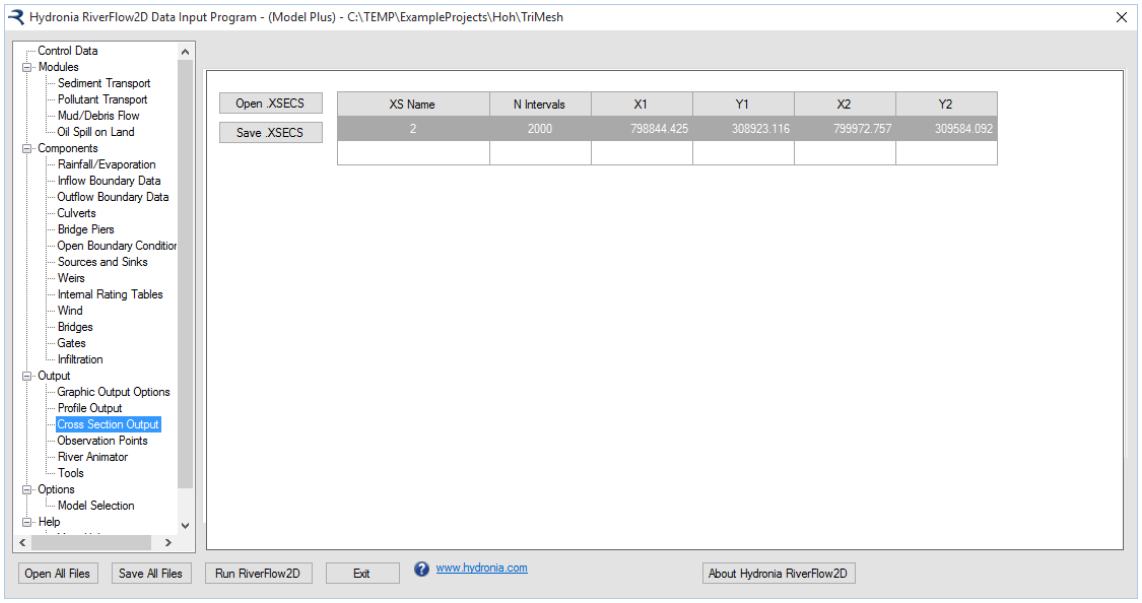


Figure 8.11 – Cross Section Output Panel.

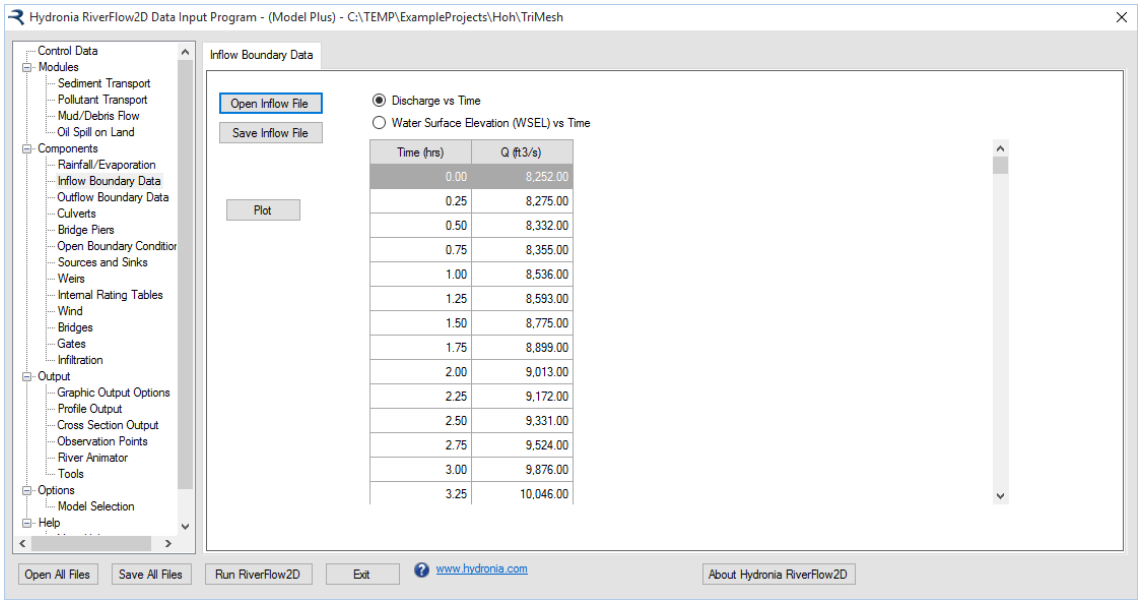


Figure 8.12 – Inflow Boundary Data Panel.

Use the Plot button to generate a graph of the time series as shown in Figure 8.13.

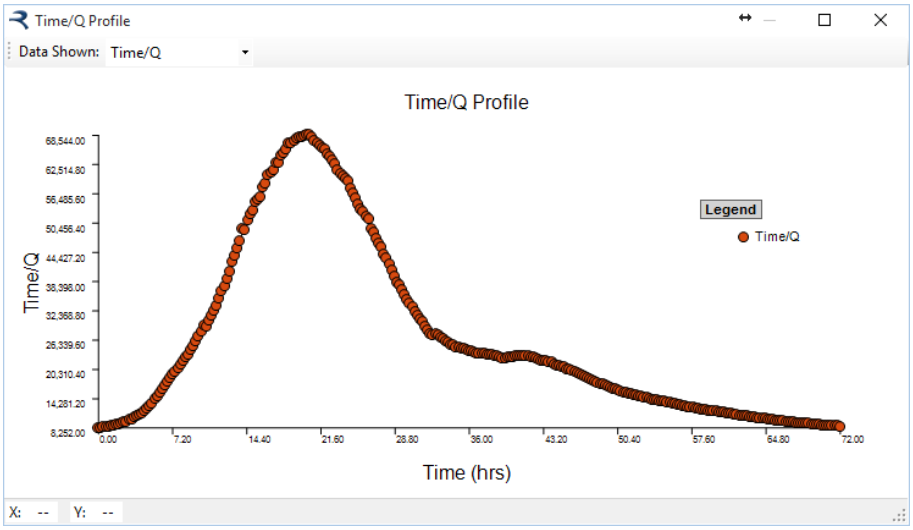


Figure 8.13 – Inflow hydrograph plot.

Table 8.17 – Parameters on the Inflow Boundary Data Panel.

COLUMN NAME	DESCRIPTION
Select boundary condition type	
Discharge vs. Time	To select a discharge vs. time hydrograph.
Water Surface Elevation (WSE) vs. Time	To select a water surface elevation vs. time data series.
Data table	
Time (hrs.)	Time in hours.
Q; WSEL*	Water discharge (m^3/s or ft^3/s); Water surface elevation (m or ft).
Buttons	
Open Inflow File	Opens an existing file with inflow data.
Save Inflow File	Saves only the inflow table to a file.
Plot	Generates a graph of the time series.

**Varies according to boundary condition type.*

8.9 Outflow Boundary Data Panel

Use this panel to create, edit, and plot hydrographs (Discharge vs. Time, Water Surface Elevation vs. Time, or Discharge Rating Curve tables) that can be assigned as outflow boundary conditions. To activate this panel, first select the *Outflow Boundary Data Output* from the *Components* group on the left panel of the RiverFlow2D DIP.

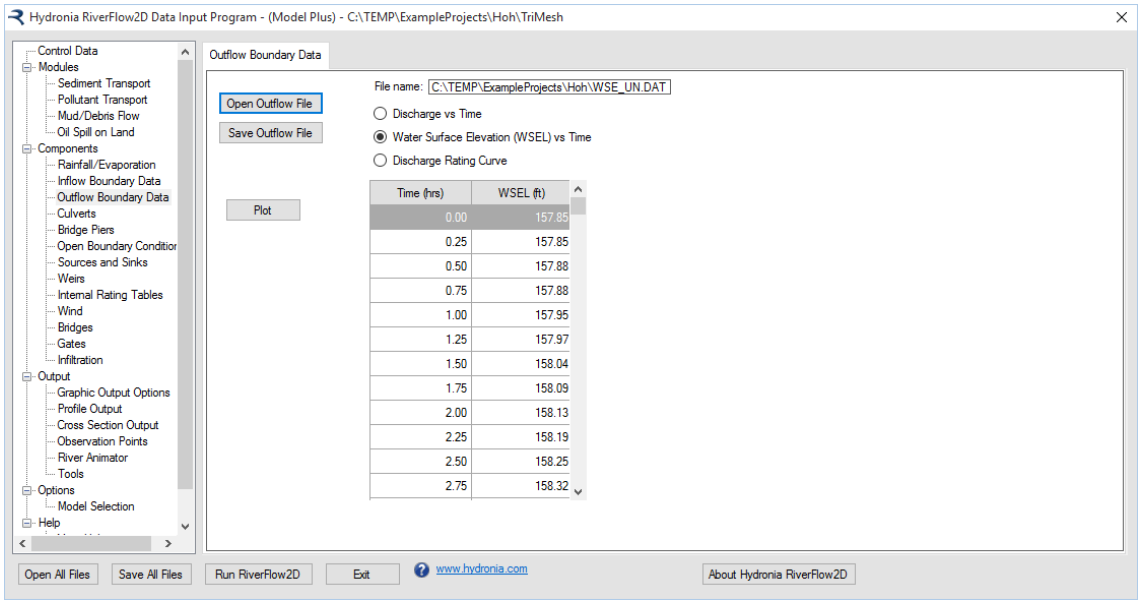


Figure 8.14 – Outflow Boundary Data Panel.

Use the Plot button to generate a graph of the time series.

Table 8.18 – Parameters on the Outflow Boundary Data Panel.

COLUMN NAME	DESCRIPTION
Select boundary condition type	
Discharge vs. Time	To select a discharge vs. time hydrograph.
Water Surface Elevation (WSE) vs. Time	To select a water surface elevation vs. time data series.
Discharge Rating Curve	To select a discharge rating curve.
Data table	
Time; Q*	Time in hours; Q (m^3/s or ft^3/s).
Q; WSE*	Water discharge (m^3/s or ft^3/s); Water surface elevation (m or ft).
Buttons	
Open Outflow File	Opens an existing file with outflow data.
Save Outflow File	Saves only the outflow table to a file.
Plot	Generates a graph of the time series.

**Varies according to boundary condition type.*

8.10 Culverts Panel (.CULVERTS File)

This panel is used to display the content of the `.CULVERTS` file and enter data for culverts. Figure 8.15 shows the *Culvert* panel with a three culverts. Selecting Culvert1 on the first row shows the associated rating table. To activate this panel, first select the *Culverts* from the *Components* group on the left panel of the RiverFlow2D DIP.

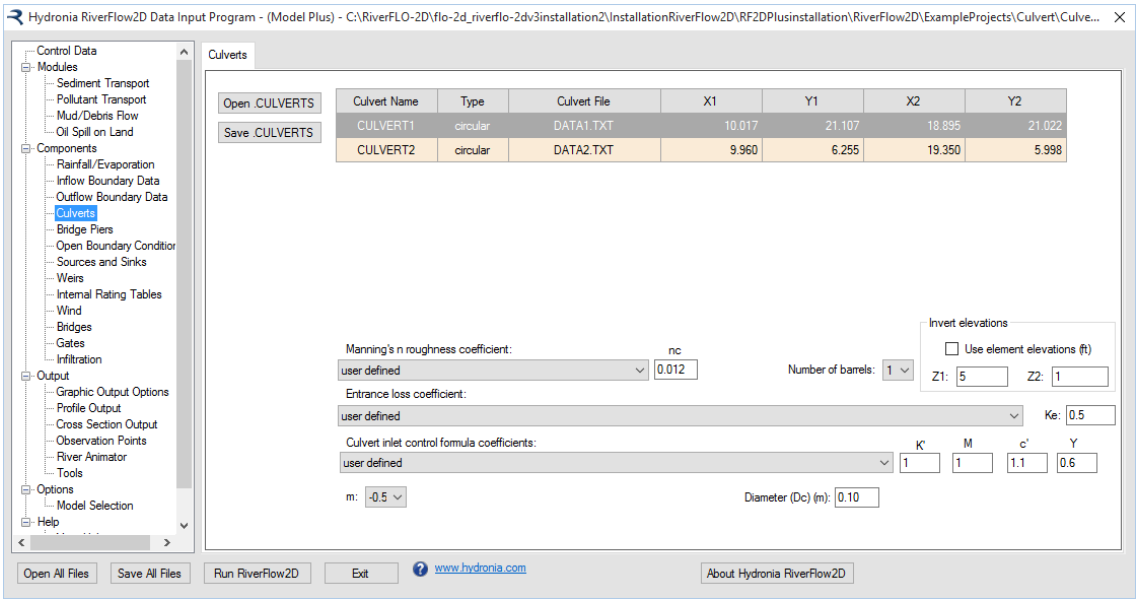


Figure 8.15 – Culverts Panel showing data in rating curve.

Figure 8.15 shows the corresponding data entry controls that appear when selecting the first row for Culvert1 that is a circular culvert.

Table 8.19 – Parameters on the Culverts Panel.

COLUMN NAME	DESCRIPTION
Culvert data	
Culvert Name	Culvert name. Should not contain spaces and must have less than 26 characters.
Type	Type of culvert. For Type = 0, culvert discharge is computed from a user given rating table on the Culvert File. For Types = 1 and 2, discharge is computed using culvert equations based on culvert characteristics provided in the Culvert File.
Culvert File	Culvert rating table file name or culvert characteristic data. Name Should not contain spaces and must have less than 26 characters.
X1, Y1, X2, Y2	Coordinates of vertices defining each culvert line.
Manning's roughness coefficient	Culvert Manning's n Coefficient given by Table 7.2.
Entrance loss coefficient	Culvert entrance loss coefficient given by Table 7.3.
Culvert inlet control formula coefficients	Culvert inlet control formula coefficients given by Table 7.4.
m	Inlet form coefficient. m=0.7 for mitered inlets, m=-0.5 for all other inlets.
Barrel height (Hb)	Barrel height for box culverts (ft or m). Only for box culverts: CulvertType = 1.

Continued on next page

Table 8.19 – continued from previous page

COLUMN NAME	DESCRIPTION
Barrel width (Base)	Barrel width for box culverts (m or ft). Only for box culverts: CulvertType = 1.
Diameter (Dc)	Barrel diameter for circular culverts (m or ft). Only for circular culverts: CulvertType = 2.
Number of barrels	Number of identical barrels.
Use element elevations	When this check box is selected the model will extract the inlet and outlet invert elevations from the element or cell elevations of the culvert ending points. If the check box is not selected, the user can enter the inlet invert elevation (Z1) and outlet invert elevations (Z2) that may be different from the element/cell elevations.
Buttoms	
Open `.CULVERTS`	Opens an existing `.CULVERTS` file.
Save `.CULVERTS`	Saves only the culvert data to a `.CULVERTS` file.

8.11 Internal Rating Tables Panel (.IRT File)

This panel is used to display the content of the `.IRT` file and enter data for culverts. In this Panel can also edit Internal Rating Table polylines, type, and data file name. To activate this panel, first select the *Internal Rating Table* from the *Components* group on the left panel of the RiverFlow2D DIP.

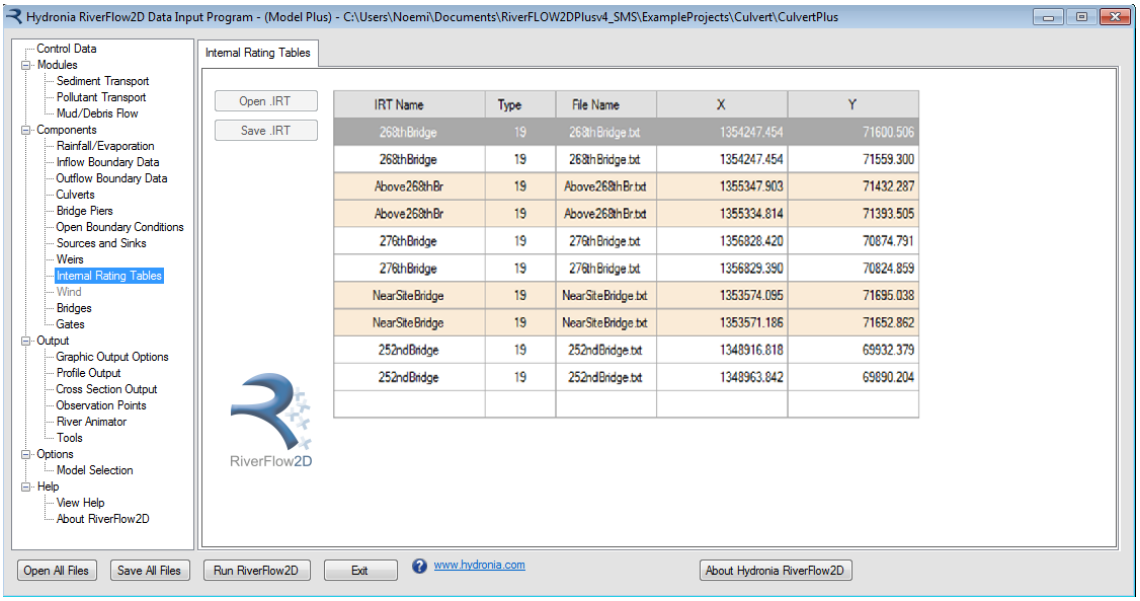


Figure 8.16 – Internal Rating Tables Panel.

Table 8.20 – Parameters on the Internal Rating Tables Panel.

COLUMN NAME	DESCRIPTION
Data table	
IRT Name	Name of internal rating table. Should not contain spaces and must have less than 26 characters.
Type	Boundary condition is always equal to 19 in this version, corresponding to discharge vs. water surface elevation tables.
File Name	Name of file containing internal rating table data in the format described as a stage-discharge data file.
X, Y	Coordinates of vertices defining each IRT polyline.
Buttoms	
Open `.IRT`	Opens an existing `.IRT` file.
Save `.IRT`	Saves only the internal rating table data to a `.IRT` file.

8.12 Weirs Panel (.WEIRS File)

This panel is used to display the content of the `.WEIRS` file. In this Panel can also create weir polyline data. To activate this panel, first select the *Weirs* from the *Components* group on the left panel of the RiverFlow2D DIP.

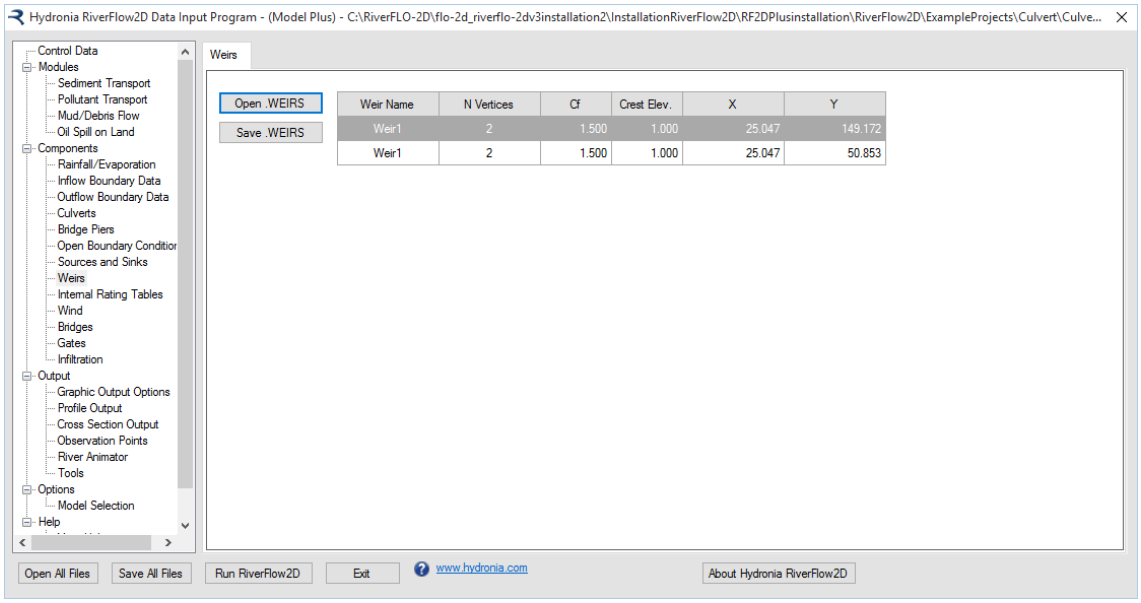


Figure 8.17 – Weirs Panel.

Table 8.21 – Parameters on the Weirs Panel.

COLUMN NAME	DESCRIPTION
Data table	
Weir Name	Name of weir. Should not contain spaces and must have less than 26 characters.
N Vertices	Number of points defining each weir polyline.
Cf	Weir coefficient.
X, Y	Coordinates of vertices defining each weir polyline (m or ft).
Buttoms	
Open `.WEIRS`	Opens an existing `.WEIRS` file.
Save `.WEIRS`	Saves only the weir data to a `.WEIRS` file.

8.13 Sources/Sinks Panel (.SOURCES File)

This panel is used to display the content of the `.SOURCES` file. Use this Panel to also create sources and sinks location data, type, and sources/sink data file. To activate this panel, first select the *Sources and Sinks* from the *Components* group on the left panel of the RiverFlow2D DIP.

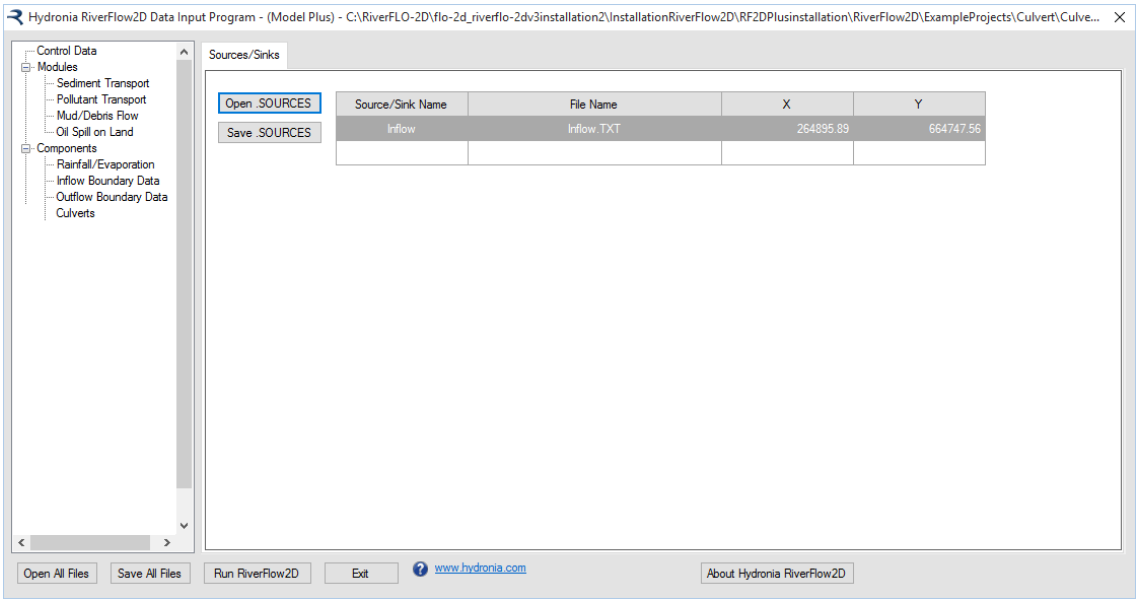


Figure 8.18 – Sources/Sinks Panel.

Table 8.22 – Parameters on the Sources/Sinks Panel.

COLUMN NAME	DESCRIPTION
Data table	
Source/Sink Name	Name of point source or sink. Should not contain spaces and must have less than 26 characters.
File Name	Name of file containing the time series of each point source or sink.
X, Y	Coordinates of point.
Buttoms	
Open `.SOURCES`	Opens an existing `.SOURCES` file.
Save `.SOURCES`	Saves only the sources and sinks data to a `.SOURCES` file.

8.14 Open Boundary Conditions Panel (.IFL File)

This panel is used to display the content of the `.IFL` file. Since this file is automatically generated by the RiverFlow2D model based on the user input in the SMS interfaces, any change made will be disregarded when the model runs. To activate this panel, first select the *Open Boundary Conditions* from the Components group on the left panel of the RiverFlow2D DIP.

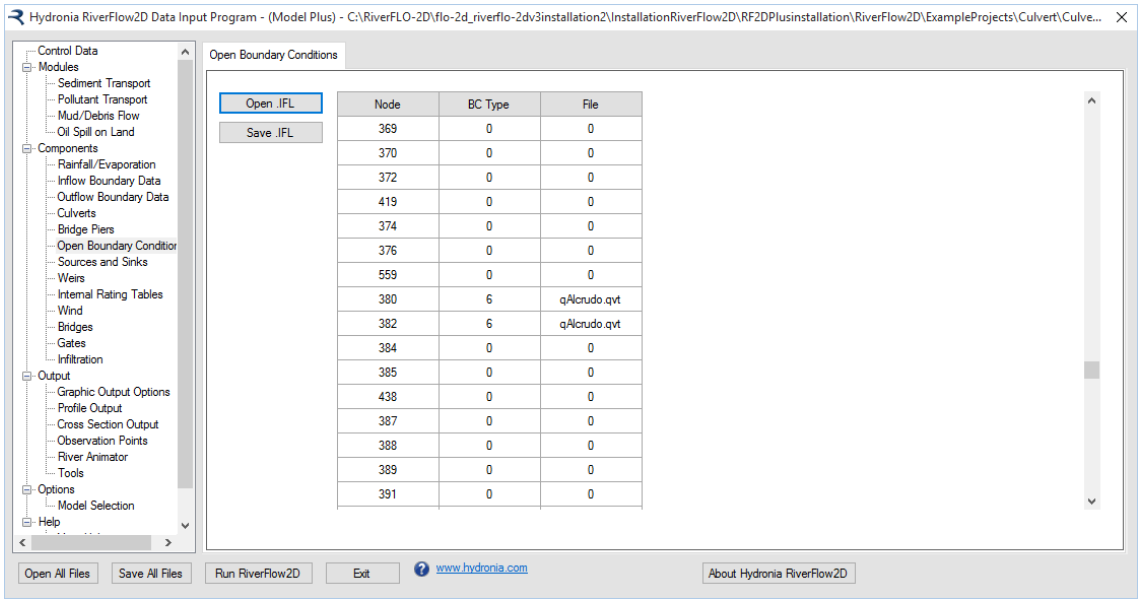


Figure 8.19 – Open Boundary Conditions Panel.

Table 8.23 – Parameters on the Open Boundary Condition Panel.

COLUMN NAME	DESCRIPTION
Data table	
Node	Node number.
BC Type	Code to indicate type of open boundary. See boundary condition options in the `.IFL` file description section.
File	Boundary condition file name. Should not contain spaces and must have less than 26 characters.
Buttoms	
Open `.IFL`	Opens an existing `.IFL` file.
Save `.IFL`	Saves only the open boundary conditions data to a `.IFL` file.

8.15 Bridge Piers Panel (.PIERS File)

This panel is used to display the content of the `.PIERS` file. In this Panel can also create bridge pier location and pier characteristic data. To activate this panel, first select the *Bridges Piers* from the *Components* group on the left panel of the RiverFlow2D DIP.

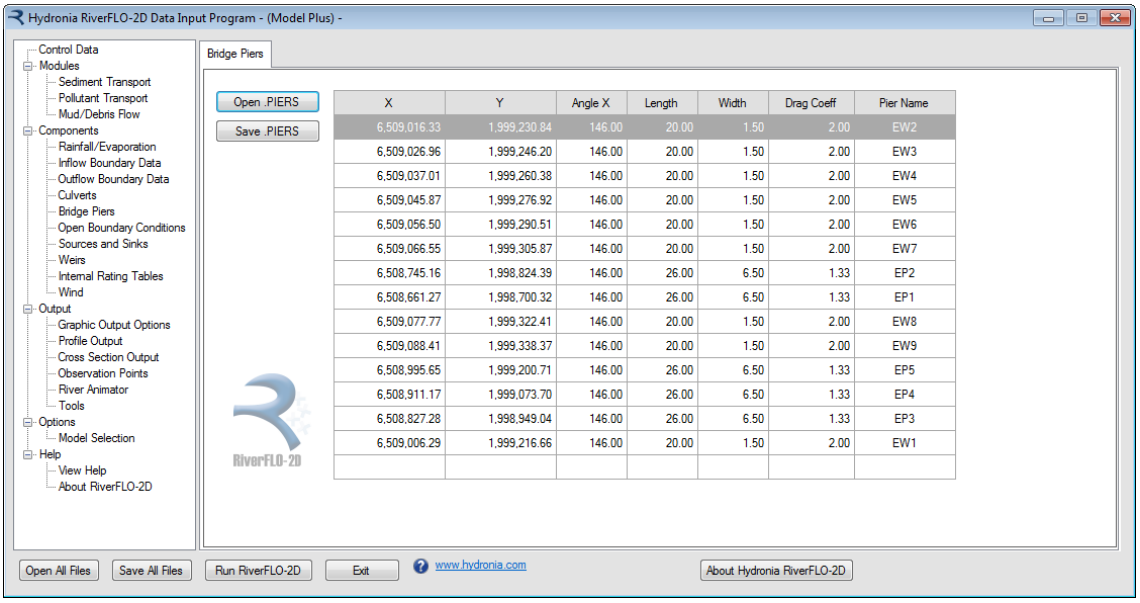


Figure 8.20 – Bridge Piers Panel.

Table 8.24 – Parameters on the Bridge Piers Panel.

COLUMN NAME	DESCRIPTION
Data table	
X,Y	Coordinates of pier centroid.
Angle X	Pier angle with respect to X axis.
Length	Pier length (m or ft).
Width	Pier width (m or ft).
Drag Coeff.	Drag coefficient of the pier.
Pier Name	Name of pier. Should not contain spaces and must have less than 26 characters.
Buttoms	
Open `.PIERS`	Opens an existing `.PIERS` file.
Save `.PIERS`	Saves only the bridge piers data to a `.PIERS` file.

8.16 Observation Points Panel (.OBS File)

Use this panel to create, edit and display the content of the `.OBS` file. To activate this panel, first select the *Observation Points* from the *Output* group on the left panel of the RiverFlow2D DIP.

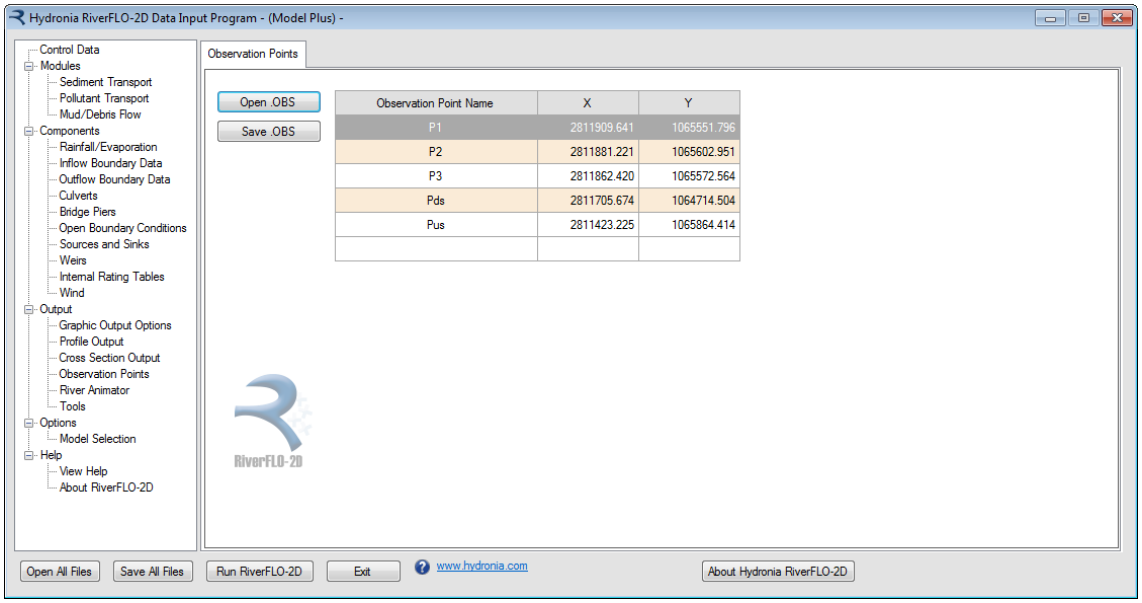


Figure 8.21 – Observation Points Panel.

Table 8.25 – Parameters on the Observation Points Panel.

COLUMN NAME	DESCRIPTION
Data table	
Observation Point Name	Name of observation point. Should not contain spaces and must have less than 26 characters.
X,Y	Coordinates of point.
Buttoms	
Open `.OBS`	Opens an existing `.OBS` file.
Save `.OBS`	Saves only the observation point data to a `.OBS` file.

8.17 Tools Panel

This section describes various utilities that are available through the RiverFlow2D Data Input Program.

8.17.1 Two-Dimensional Model Result Interpreter: 2dMRI

2dMRI is a RiverFlow2D utility developed by Northwest Hydraulic Consultants (`http://www.nhc-sea.com`). The program gathers the data from RiverFlow2D output files and generates plots and animations for cross sections, profiles and observation points.

To access the program use first select *Tools* from the *Output* group on the left panel of the RiverFlow2D DIP.

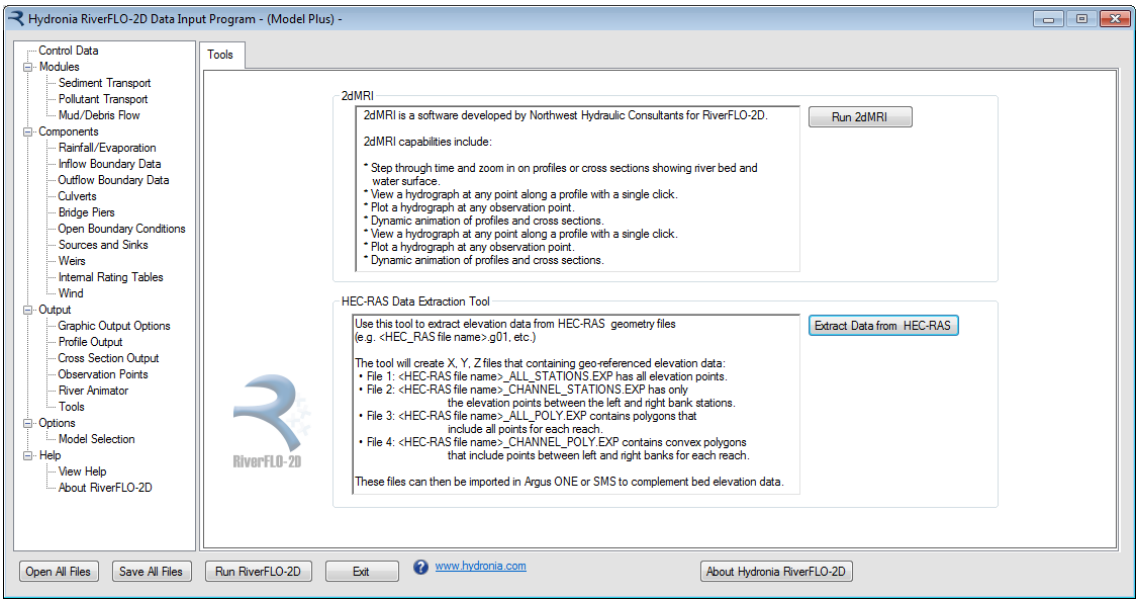


Figure 8.22 – Two-dimensional Model Result Interpreter: 2dMRI.

Explanation and guidelines on how to use the 2dMRI utility is described in the 2dMRI manual included in the documentation folder:

...\Documents\RiverFlow2Dv4\Documentation.

8.17.2 HEC-RAS Data Extraction Tool

The purpose of this tool is to facilitate migrating existing HEC-RAS projects to RiverFlow2D. The program allows extraction of point elevation data from geo-referenced cross-section from the HEC-RAS one-dimensional model developed by the USACE. The tool reads HEC-RAS geometry files with extension `.g01`, `.g02`, etc., and creates X Y Z files that can be readily imported in SMS. The utility discriminates the elevations in the channel between the left and right bank on each cross section and exports the files as detailed in the following table.

Table 8.26 – Files generated by the HEC-RAS Data Extraction Tool.

FILE NAME	DESCRIPTION
`<HEC-RASfilename>_ALL_STATIONS.EXP`	Contains all elevation points in all cross sections in the for all reaches and cross sections in the `<HEC-RASfilename>.g0?` file.
`<HEC-RASfilename>_ALL_POLY.EXP`	Contains polygons that include all elevation points in each reach.
`<HEC-RASfilename>h0_CHANNEL_STATIONS.EXP`	Contains only the elevation points between the left and right banks in all cross sections in the for all reaches in the `<HEC-RASfilename>.g0?` file.
`<HEC-RASfilename>_CHANNEL_POLY.EXP`	Contains polygons that include only the elevation in the main channel for each reach.

To activate this panel, first select the *Tools* from the *Output* group on the left panel of the RiverFlow2D DIP.

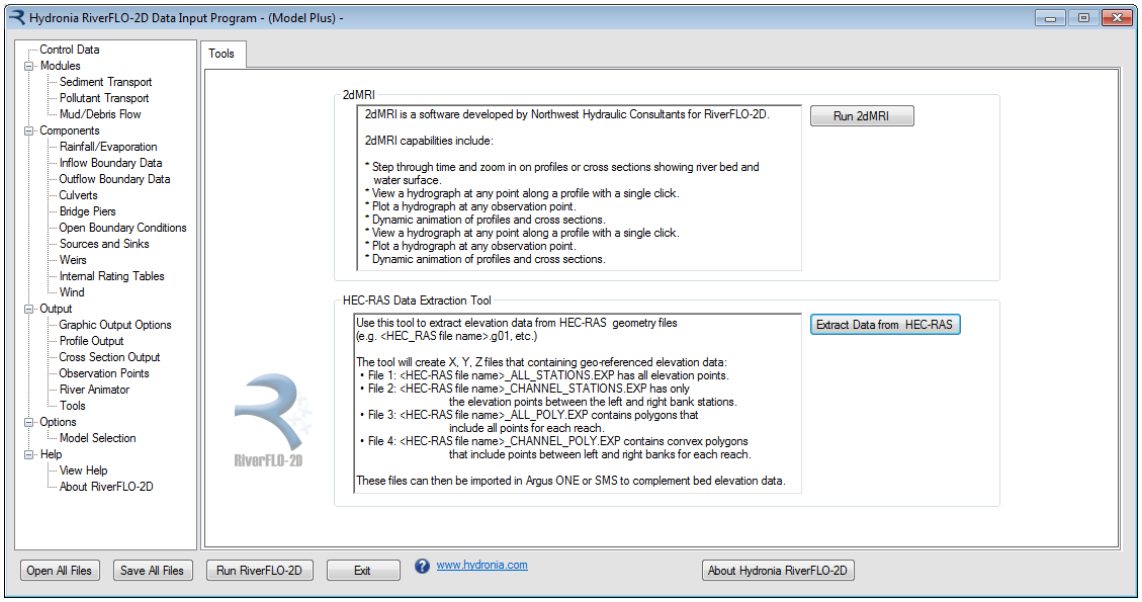


Figure 8.23 – Two-dimensional Model Result Interpreter: 2dMRI.

The tutorial: "Using HEC-RAS elevation data" in the RiverFlow2D User's Guide, describes in detail the procedure to use this tool.

9 Input Data File Reference

Data files for non-spatial information required to run the RiverFlow2D model are created by the Data Input Program and the spatial data is generated based on the `.2dm` file. All RiverFlow2D input data files are in ASCII free-form format, which can be opened using any text editor or spreadsheet program. In some instances it may be convenient to directly edit the data. However, it is recommended to edit files with extreme caution, and only after having gained a thorough understanding of RiverFlow2D file formats. This section explains the input data files, and the parameters included in each file.

The RiverFlow2D installation program creates a folder with several example projects that can be consulted to review the model data files. Depending on your operating system and settings, this folder can be found in:

.\Documents\RiverFlow2D_SMS\ExampleProjects.

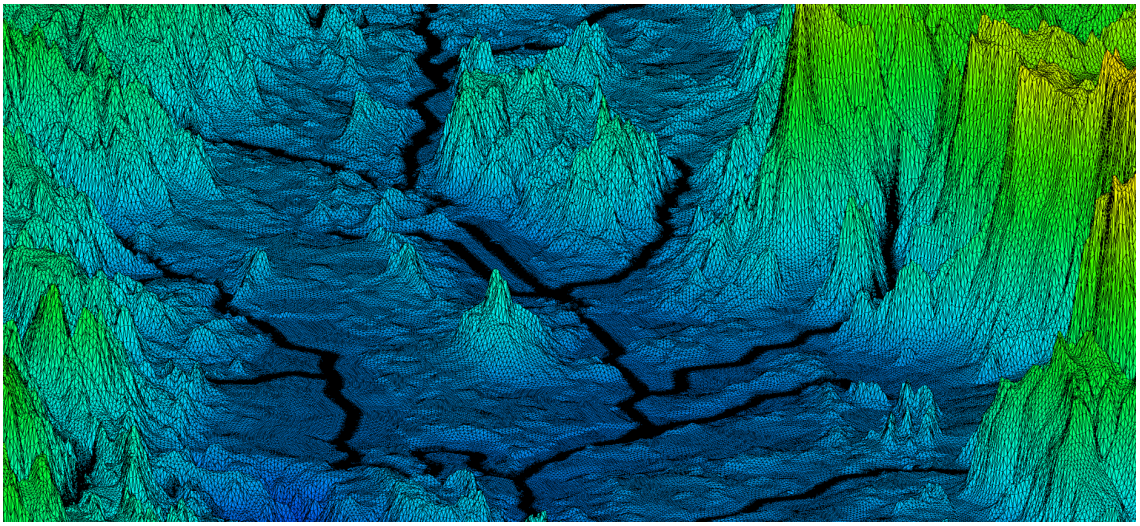


Figure 9.1 – Example of a RiverFlow2D Mesh.

RiverFlow2D data files will share the same name and will use the file extensions listed in the table below. For example a run named Run1 will have files as follows: `Run1.DAT`, `Run1.FED`, etc. The following table summarizes the data files used by RiverFlow2D model.

Table 9.1 – List of Input Data Files.

NAME	FILE EXTENSION	DEPENDENCIES	CONTENT
SMS			
SMS project file	`.SMS`	Required when using SMS GUI	This is the project file where SMS stores the triangular element mesh, the data and projection.
2DM file	`.2DM`	Required when using SMS GUI	This is the file where SMS stores all the data for a simulation using the generic mesh.
INDIVIDUAL DATA FILES CREATED FROM THE *2DM			
SPATIAL DATA FILES			
Elevation data	any	Required	Scattered elevation data points.
Triangle-element mesh data	`.FED`	Required	Node coordinates and elevations, triangular mesh topology, boundary condition type and file names, initial water elevations, and Manning's n coefficients.
Mesh boundary nodes	`.TBA`	Internal file	List of external and island boundary nodes. This file is automatically generated by RiverFlow2D.
I/O boundary conditions	`.IFL`	Internal file	List of external boundary nodes, inflow and outflow conditions. This file is automatically generated by RiverFlow2D.
Boundary condition nodal file	`.OBCP`	Required	List of external boundary conditions. For each boundary, it contains the list of nodes and the associated data file. This file is now automatically generated by RiverFlow2D, based on the information in the `.FED` file.
CONTROL FILES			
Run control data	`.DAT`	Required	General run control options, including time step, simulation time, metric or English units, graphical output options, initial conditions, components, etc.
Plot results options	`.PLT`	Optional	Graphical output options.
Observation points data	`.OBS`	Optional	Location of observation points where the model will report time series of results.
Cross section output	`.XSECS`	Optional	List of cross sections where the model will output results. Each cross section is defined by coordinates of its two ending points.
Profile output	`.PROFILES`	Optional	Mesh profile cut where results are desired.
BOUNDARY CONDITIONS			
Time series or rating table files for inflow or outflow boundary conditions	user defined	Required	Hydrograph, water surface elevations vs. time, etc. The model requires one file for each open boundary condition, except "free" boundary condition types.
Initial concentration of each pollutant	`.CINITIAL`	The Pollutant Transport component is selected	Defines the initial concentrations over the mesh.
COMPONENT FILES			
Bridges	`.BRIDGES`	The Bridges component is selected	Bridge cross section geometry file is used to compute energy losses.
Culverts	`.CULVERTS`	The Culvert component is selected	Culvert location and associated culvert data files.

Continued on next page

Table 9.1 – continued from previous page

NAME	FILE EXTENSION	DEPENDENCIES	CONTENT
Gates	.GATES	The Gates component is selected	Gate location and associated gate aperture data files.
Infiltration	.LRAIN	The Infiltration component is selected	Infiltration parameters data file.
Internal rating tables	.IRT	The Internal Rating Table component is selected	Data to impose discharge rating tables along internal boundaries.
Manning's n variable with depth	.MANNN	Created from polygons on the ManningsNz coverage	Provides the parameters necessary to account for Manning's n roughness coefficient that vary with depth according to a user provided table.
Mud	.MUD	The Mud/Debris Flow component is selected	Provides the parameters necessary to model mud and debris flow.
Oil Spills	.OILP	The Overland Oil Spill component is selected	Provides the parameters necessary to model overland oil spills.
Bridge piers	.PIERS	The Piers component is selected	Bridge pier data used to calculate pier drag forces.
Pollutant transport	.SOLUTE	The Pollutant Transport component is selected	Data for passive or reactive pollutants.
Rainfall/Evaporation.	LRAIN	The Rainfall/Evaporation component is selected	Time series for rainfall and evaporation.
Bed load sediment transport	.SEDB	Sediment transport model	Bed load sediment transport data.
Suspended sediment transport	.SEDS	Sediment transport model	Suspended sediment transport data.
Sources and sinks	.SOURCES	The Sources component is selected	This file contains location of input discharge sources or output discharge sinks and associated time series of discharge data files.
Weirs	.WEIRS	The Weir component is selected	This file contains weirs polylines and associated weir data.
Wind	.WIND	The Wind component is selected	This file contains wind specific density and velocity data.

9.1 Run Control Data

9.1.1 Run Control Data File: .DAT

This file contains parameters to control the model run including time step, simulation time, metric or English units, physical processes or component switches, and graphical output and initial conditions options.

Line 1: Internal program version number.
RELEASE

Line 2: Model selector switch.
IMS

Line 3: Physical processes or component switches.
IRAIN ISED IPIERS IWEIRS ICULVERTS ISOURCES IINTRC IBRIDGES IGATES IDAMS FUTUREUSE

Line 4: Wet-dry bed method switch.
IWETDRY

Line 5: Output control switches.
IEXTREMES IXSEC IPROFILE DUMMY IOBS

Line 6: Time control data.
DUMMY CFL DUMMY TOUT TLIMT

Line 7: Initial conditions and hot start control switches.
IINITIAL IHOTSTART

Line 8: Manning's n variable with depth switch.
IMANN

Line 9: Manning's n value global multiplication factor.
XNMAN

Line 10: Dummy parameter.
DUMMY

Line 11: Unit system definition switch.
NUNITS

Line 12: Surface detention or minimum value of flow depth for dry areas.
HMIN

Line 13: Initial water surface elevation.
INITIAL_WSE

Line 14: Pollutant transport model switch.
IADDISP

Line 15: Wind stress switch.
IWIND

Line 16: Mud/Debris flow model switch.
IMUD

Line 17: Number of cores or GPU ID.
IDGPU

Line 18: Graphical User Interface that created the files.
IGUI

9.1.1.1 Example of the .DAT file

201301
1
0 0 0 0 1 1 0 0 0 0 0

```
2
0 0 0 0 0
0 0.5 0.25 0.25 8
1 0
1
1
0.9
1
-1
0
0
0
0
4
1
```

Table 9.2 – Variable Descriptions for the .DAT File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CFL	R	$(0, 1]$	-	Applies to RiverFlow2D Plus and RiverFlow2D Plus GPU models. Courant number. Default value is set to 0.7. CFL may need to be set to lower values if results show signs of unexpected oscillations or instability. CFL may often be increased to 1 with stable results.
DUMMY	R	-	-	Dummy parameter for future use. Ignored in RiverFlow2D Plus.
HMIN	R	-1 or > 0	m/ft	In RiverFlow2D Plus HMIN is the depth limit for dry-wet calculation. If depth is less than HMIN, cell velocity will be set to 0. If HMIN = -1, all cells with depth less than 10^{-6} m will be considered dry.
IADDISP	I	0,1	-	Switch to activate the pollutant transport model. 0: Turn off pollutant transport computations. 1: Apply pollutant transport.
IBRIDGES	I	0,1	-	Switch to activate the Bridges component. 0: Turn off Bridges component. 1: Apply Bridges component. Requires `.BRIDGES` file. See details on the *Bridges* Section of this manual.

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ICULVERTS	I	0,1	-	Switch indicating if one-dimensional culverts will be used. 0: No culverts will be used. 1: Use culverts. Requires `.CULVERTS` file. See details on the *Culverts* Section of this manual.
IDAMS	I	0,1	-	Switch to activate the Dam Breach component. 0: Turn off Dam Breach component. 1: Apply Dam Breach component. Requires `.DAMBREACH` file. See details on the *Dam Breach* Section of this manual.
IDGPU	I	≥ 0	-	RiverFlow2D Plus: This parameter indicates how many processors or cores will be used in the parallel computation. The maximum number will depend on the processor capabilities. RiverFlow2D Plus GPU: If your computer has multiple GPU cards, this parameter allows selecting which card will be used for the run. Since the model allows only one concurrent run per cards, this option allows running simultaneous simulations in different cards.
IEXTREMES	I	0,1	-	Switch to reporting maximum values throughout the simulation. 0: Do not report maximum values. 1: Report maximum values.
IGATES	I	0,1	-	Switch to activate the Gates component. 0: Turn off Gates component. 1: Apply Gates component. Requires `.GATES` file. See details on the *Gates* Section of this manual.
IGUI	I	1, 2	-	This parameter indicates what Graphical User Interface was used to create the RiverFlow2D files. 1: Aquaveo SMS 2: QGIS

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IHOTSTART	I	0,1	-	Switch to start run from scratch or continue a previous simulation. 0: Start simulation from initial time. 1: Start simulation from previous run.
IINTRC	I	0,1	-	Switch for internal rating tables. 0: Do not use internal rating table component. 1: Use internal rating tables. See details on *Internal Rating Tables* Section of this manual.
IINITIAL	I	0,1,2,-9999	-	Initial condition switch for water surface elevations. 0: Prescribed horizontal water surface elevation 1: Initial dry bed on whole mesh. 2: Initial water surface elevations read from `.FED` file -9999: Assigns a horizontal water elevation equal to the maximum bed elevation plus 0.5 m. (1.64 ft.). See comment 3.
INITIAL_WSE	R	-	m/ft	Initial water surface elevation on the whole meshes. This will be the initial water surface if IINITIAL is 0. See comment 3.
IMANN	I	1,2	-	Variable Manning's n with depth switch. 0: Manning's n is constant for all depths. 1: Manning's n may vary with depths as defined in the `.MANNN` file.
IMS	I	1,2	-	Model switch used to select the hydrodynamic model engine. 1: RiverFlow2D Plus. 2: RiverFlow2D Plus GPU.
IMUD	I	0,1	-	Switch to select mud/debris flow model. 0: Do not run mud/debris flow model. 1: Run mud/debris flow model. Requires `.MUD` file. See details on the Mud/Debris Flow Model section of this manual.

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IOBS	I	0,1	-	Switch to report time series of results at specified locations defined by coordinates. 0: Do not report on observation points. 1: Report on observation points. Requires `.OBS` file. See details on the *Observation Points* section of this manual.
IPIERS	I	0,1	-	Switch to allow accounting for pier drag force. 0: Do not use pier drag force option. 1: Use pier drag force option. Requires `.PIERS` file. This option may be used if the mesh does not account for the pier geometry. See details on *Bridge Piers Section* of this manual.
IPOLLUTANT	I	0,1	-	Switch to select pollutant transport model. 0: Do not run pollutant transport model. 1: Run pollutant transport model. Requires `.AD` file. See details on the *Pollutant Transport Model* section of this manual.
IPROFILE	I	0,1	-	Switch to control profile output. 0: No profile results output. 1: Results will be output along a prescribed profile. Requires `.PROFILES` file. See comment 4.
IRAIN	I	0-4	-	Switch for rainfall and evaporation input. 0: No rainfall modeling. 1: Not used. 2: Rainfall/evaporation. 3: Infiltration. 4: Rainfall/evaporation and Infiltration.
ISED	I	0,1	-	Sediment transport switch. 0: No sediment transport modeling. 1: Sediment transport, mobile bed erosion, and deposition will be simulated. Requires `.SED` file. See details on the *Sediment Transport* section of this manual.

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ISOURCES	I	0,1	-	Switch for sources and sinks. 0: No sources or sinks are present. 1: Sources or sinks are present. Requires `.SOURCES` file. See details on the *Sources* section of this manual.
IWEIRS	I	0,1	-	Weir computation on internal boundary switch. 0: Do not use weir computation on internal boundaries. 1: Use weir computation on internal boundaries. See details on the *Weirs* section of this manual.
IWIND	I	0,1	-	Switch to account for wind stress on the water surface. 0: Do not consider wind stress. 1: Consider wind stress. Requires `.WIND` file. See details on the *Wind Stress* section of this manual.
IXSEC	I	0,1	-	Cross section output switch. 0: No cross section result output. 1: Cross section results will be output to file. Requires `.XSECS` file. See comment 5.
NUNITS	I	0,1	-	Variable to indicate unit system: 0: Metric units. 1: English units.
RELEASE	I	-	-	Release number ID used internally for reference. Should not be modified.
TLIMT	R	> 0	hrs.	Total simulation time.
TOUT	R	$\leq TLIMT$	hrs.	Output time interval for reporting results.
XNMAN	R	[0.1-2]	-	Manning's n coefficient multiplier. See comment 6.

Note: I = Integer variable. R = Real variable.

9.1.1.2 Comments for the .DAT file

1. Setting the CFL (Courant Friederich-Lewy) or Courant number is critical for adequate stability and ensure mass conservation. RiverFlow2D explicit time scheme is conditionally stable, meaning that there is a maximum time step above which the simulations will become unstable. This threshold can be theoretically approximated by a Courant-Frederick-Lewy

condition defined as follows:

$$CFL = \frac{\Delta t\sqrt{gh}}{\Delta x} \leq 1 \tag{9.1}$$

where Δt = DT is the time-step, Δx is a measure of the minimum triangular element or cell size, g is the acceleration of gravity, and h is the flow depth. It may occur that during the initial stages of a hydrograph, velocities are small and the selected time step is adequate. During the simulation, however, velocities and flow depth may increase causing the stability condition to be exceeded. In those cases it will be necessary to rerun the model with a smaller CFL. Alternatively, the variable time step option may be used.

2. For variable time step simulations, RiverFlow2D estimates the maximum DT using the theoretical Courant-Frederick-Lewy (CFL) condition. Sometimes, the estimated DT may be too high, leading to instabilities, and it may be necessary to reduce CFL to with a value less than one to adjust it. Typical CFL values range from 0.3 to 1, but may vary project to project.

3. There are three initial conditions options. If IINITIAL = 0, the initial water elevation will be a constant horizontal surface at the elevation given as INITIAL_WSE. If INITIAL_WSE is = -9999 then the program will assign a constant water elevation equal to the highest bed elevation on the mesh. If IINITIAL = 1, the whole computational mesh will be initially dry, except at open boundaries where discharge is prescribed and depth > 0 is assumed for the first time step. If IINITIAL = 2, initial water surface elevations are read from the `.FED` data file for each node in the mesh. These initial elevations can be set creating a dataset file (`.h5`) on SMS layer, and can be arbitrary. This last option allows modeling cases where part of the domain is flooded (e.g. a reservoir) and other parts of the mesh are dry. The initial velocities are always assumed to be equal to 0.

4. Use the IPROFILE option to allow RiverFlow2D to generate results along a polyline. The polyline and other required data should be given in the Profiles file (`.PROFILES`), which is defined later in this document.

5. Use this option to allow RiverFlow2D to generate results along prescribed cross sections. The cross sections and other required data should be given in Cross Section file (`.XSECS`) which is defined later in this document.

6. Use the XNMAN option to test the Manning's n value sensitivity on the results. The prescribed Manning's coefficient assigned to each element will be multiplied by XNMAN. This option is useful to test model sensitivity to Manning's n during model calibration.

7. The model will create output files with maximum values of each output variable.

8. The user can specify an initial water surface elevation setting IINITIAL = 0 and entering INITIAL_WSE.

9.2 Mesh Data

9.2.1 Mesh Data File: .FED

This file has the data that defines the triangular element mesh, and includes node coordinates, connectivity for each triangular element, node elevations, Manning's n coefficients and other parameters. This file is created with SMS GUI. The templates for RiverFlow2D provided with

SMS assures that the `.FED` file will be created error free and consistent with the boundary conditions and other mesh parameters. Manually editing this file may introduce unexpected errors.

Line 1: Number of elements and nodes.
NELEM NNODES DUMMY DUMMY

NNODES lines containing node coordinates and node parameters.
IN X(IN) Y(IN) ZB(IN) INITWSE(IN) MINERODELEV(IN) BCTYPE BCFILENAME

NELEM lines containing mesh connectivity and element/cell parameters.
IE NODE(IE,1) NODE(IE,2) NODE(IE,3) MANNINGN(IE) ELZB(IE) ELINITWSE(IE) ELMINERODELEV(IE)

9.2.1.1 Example of a .FED file

```
1773 954 5 5
1 798427.180 305984.903 160.800 0.000 -9999.000 0 0
2 798375.129 305901.823 160.400 0.000 -9999.000 11 0
3 798456.481 305851.866 160.600 0.000 -9999.000 11 0
4 800145.687 304890.733 166.800 0.000 -9999.000 11 0
5 800164.855 304802.764 162.400 0.000 -9999.000 11 0
6 800219.908 304878.337 171.200 0.000 -9999.000 0 0
7 800083.504 304852.721 161.800 0.000 -9999.000 11 0
8 798619.614 309561.972 161.834 0.000 -9999.000 0 0
9 798519.268 309583.373 161.736 0.000 -9999.000 6 qin_UN.dat
10 798566.714 309501.694 162.334 0.000 -9999.000 0 0
11 798579.326 309637.425 160.892 0.000 -9999.000 6 qin_UN.dat
-
-
-
1 1 2 3 0.040 160.400 0.000 -9999.000 0.000
2 4 5 6 0.040 163.200 0.000 -9999.000 0.000
3 4 7 5 0.040 161.350 0.000 -9999.000 0.000
4 8 9 10 0.040 162.322 0.000 -9999.000 0.000
5 8 11 9 0.040 161.238 0.000 -9999.000 0.000
6 12 11 8 0.040 160.944 0.000 -9999.000 0.000
7 12 13 11 0.040 160.634 0.000 -9999.000 0.000
8 14 13 12 0.040 159.446 0.000 -9999.000 0.000
9 14 15 13 0.040 159.544 0.000 -9999.000 0.000
-
-
```

This mesh has 1773 elements, 954 nodes.

Table 9.3 – Variable Descriptions for the .FED File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCTYPE	I	-	-	Code to indicate type of open boundary. See further details about boundary conditions on the `.IFL` file description below.

Continued on next page

Table 9.3 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCFILENAME	S	< 26	-	Boundary condition file name. Should not contain spaces and must have less than 26 characters. See further details on the `.IFL` file description below.
DUMMY	I	-	-	Always equal to 2.
ELINITWSE(IE)	R	-	m or ft	Initial water surface elevation for element/cell EL. Used in RiverFlow2D Plus and RiverFlow2D Plus GPU.
ELMINERODELEV (IE)	R	≥ 0	-	Minimum erosion elevation allowed at each cell. Used in RiverFlow2D Plus and RiverFlow2D Plus GPU.
ELZB (IE)	R	-	m or ft	Initial bed elevation for element/Cell EL. Used in RiverFlow2D Plus and RiverFlow2D Plus GPU.
INITWSE(IN)	R	-	m or ft	Initial water surface elevation for node IN.
IE	I	> 0	-	Element index. Consecutive from 1 to NELEM.
IN	I	> 0	-	Node number. Consecutive from 1 to NNODES.
MANNINGN(IE)	R	> 0	-	Manning's n value for element IE.
MINERODELEV (IN)	R	≥ 0	m or ft	Minimum erosion elevation allowed at each node.
NELEM	I	1-5	-	Number of triangular elements.
NNODES	I	> 0	-	Number of triangular element nodes.
NODE(IE,1), NODE(IE,2), NODE(IE,3)	I	> 0	-	Node numbers for element IE given in counter clockwise direction.
X(IN)	R	-	m or ft	X coordinate for node IN.
Y(IN)	R	-	m or ft	Y coordinate for node IN.
ZB (IN)	R	-	m or ft	Initial bed elevation for node IN.

Note: I = Integer variable. R = Real variable. S = Text variable..
All variables are separated by at least one blank space.

9.2.2 Open Boundary Conditions Data Files: .IFL and .OBCP

These file contain boundary condition data used only internally by the model. Both files are automatically generated by RiverFlow2D. The format of the .IFL file is as follows

Line 1: Number of nodes on external boundary.
NNODESBOUNDARY

NNODESBOUNDARY lines containing the external boundary conditions data.
NODE BCTYPE BCFILENAME

9.2.2.1 Example of a .IFL file

```
1165
365     1     WSE97out.txt
```

```
367     1     WSE97out.txt
431     1     WSE97out.txt
...
```

This `.IFL` file has 1165 nodes on the boundary. Node 365 has a BCTYPE=1 (Water Surface Elevation) and the time series of water surface elevations vs. time is in file `WSE97out.txt`.

The format of the .OBCP file is as follows

Line 1: Number of open inflow and outflow boundaries.
NOB
NOB groups of lines containing the following data.
BCTYPE
BCFILENAME
NNODESBOUNDARYI

NNODESBOUNDARYI lines containing the list of nodes on this boundary.
NODE[I]

9.2.2.2 Example of a .OBCP file

```
2
12
UNIF1.DATP
24
2916
...
3299

6
INFLOW1.QVT
17
2
1
...
25
26
```

This `.OBCP` file has 2 open boundaries. The first open boundary is BCTYPE=12 corresponding to Uniform Flow outflow. The uniform flow WSE vs Discharge table is included in file `UNIF1.DATP`, and there are 24 nodes on the boundary. The second open boundary is BCTYPE = 6 corresponding to inflow hydrograph where the Discharge vs time table is given in file `INFLOW1.QVT`, and there are 17 nodes on the boundary.

Table 9.4 – Variable Descriptions for the .IFL and .OBCP Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCTYPE	I	-	-	Code to indicate type of open boundary. See Table 9.5 and comment 1.

Continued on next page

Table 9.4 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCFILENAME	S	< 26	-	Boundary condition file name. Should not contain spaces and must have less than 26 characters. See comments 2 and 3.
NOB	I	-	-	Number of open inflow or outflow boundaries.
NODE	I	-	-	Node number.
NNODESBOUNDARYI	I	-	-	Number of nodes on open boundary I.
NNODESBOUNDARY	I	-	-	Total number of nodes on boundary.

Note: I = Integer variable. S = Text variable.

Table 9.5 – Boundary Condition Types.

BCTYPE	DESCRIPTION
0	Closed impermeable boundary. Slip boundary condition (no normal flow) is imposed. See comment 5.
1	Imposes Water Surface Elevation. An associated boundary condition file must be provided. See comments 2 and 4.
2	This code is only applicable for RiverFlow2D FE. Imposes U velocity component. An associated boundary condition file must be provided. See comment 2.
3	This code is only applicable for RiverFlow2D FE. Imposes V velocity component. An associated boundary condition file must be provided. See comment 2.
4	This code is only applicable for RiverFlow2D FE. Imposes U and V velocity components. A two-variable boundary condition file must be provided. See comment 3.
5	Imposes water discharge and water surface elevation. A two-variable boundary condition file must be provided. See comment 3.
6	Imposes water discharge. An associated boundary condition file must be provided. See comment 2.
7	This code is only applicable for RiverFlow2D FE. Imposes U velocity and water surface elevation. A two-variable boundary condition file must be provided. See comment 3.
8	This code is only applicable for RiverFlow2D FE. Imposes V velocity and water surface elevation. An associated boundary condition file must be provided. See comment 3.
9	Imposes single-valued stage-discharge rating table. An associated boundary condition file must be provided. See comment 6.
10	"Free" inflow or outflow condition. Velocities and water surface elevations are calculated by the model. See comment 7.
11	"Free" outflow condition. Velocities and water surface elevations are calculated by the model. Only outward flow is allowed. See comment 7.
12	Uniform flow outflow condition. See comment 10.
13-18	For future use.
19	Imposes single-valued stage-discharge rating table along an internal polyline. An associated boundary condition file must be provided. See comment 8.
20-25	For future use.
26	Imposes water discharge and sediment discharge time series. An associated boundary condition file must be provided. See comment 9.
61	This code is only applicable for RiverFlow2D FE. Imposes water discharge creating sources on all inflow elements. An associated boundary condition file must be provided. See comment 11.

9.2.2.3 Comments for the .IFL and .OBCP files

1. RiverFlow2D allows having any number of inflow and outflow boundaries with various combinations of imposed conditions. Proper use of these conditions is a critical component of a successful RiverFlow2D simulation. Theoretically, for subcritical flow it is required to provide at least one condition at inflow boundaries and one for outflow boundaries. For supercritical flow all conditions must be imposed on the inflow boundaries and 'none' on outflow boundaries. Table 9.6 helps determining which conditions to use for most applications.

Table 9.6 – Supercritical Flow Regime Boundary Conditions.

FLOW REGIME AT BOUNDARY	INFLOW BOUNDARY CONDITION	OUTFLOW BOUNDARY CONDITION
Subcritical	Q or Velocity	Water Surface Elevation
Supercritical	Q and WSE	"Free"

NOTE: It is recommended to have at least one boundary where WSE or stage-discharge is prescribed. Having only discharge and no WSE may result in instabilities due to violation of the theoretical boundary condition requirements of the shallow water equations.

2. When imposing a single variable (water surface elevation, Q, U or V), the user must provide an ASCII file with the time series for the corresponding variable. See section Boundary Conditions Data Files for details on the format for one-variable boundary condition files.

3. When imposing two variables (water surface elevation and discharge, etc.), it is required to provide an ASCII file with the time series for the variables. See section Boundary Conditions Data Files for details on the format for two-variable boundary condition files.

4. When imposing water surface elevation it is important to check that the imposed value is higher than the bed elevation. Even though RiverFlow2D can run with that condition, it could lead to volume conservation errors.

5. A closed boundary condition is imposed by default on all boundary nodes. In this case, the model calculates velocities and water surface elevations for all nodes on the boundary depending on the value of the ISPLIPBC parameter. For example ISLIPBC = 1 will impose slip conditions setting zero-flow across the boundary. Tangential flow is free corresponding to a slip condition.

6. When using a single valued stage-discharge condition the model first computes the discharge on the boundary then interpolates the corresponding water surface elevation from the rating table and imposes that value for the next time step. In case the boundary is dry, it functions as a free condition boundary (see comment 7). Water surface elevations are imposed only on wet nodes. This condition requires providing an ASCII file with the table values entries. See section Boundary Conditions Data Files for details on the file format. In general it is preferable to use stage hydrograph rather than stage-discharge condition. In most small slope rivers, the stage-discharge relationship is affected by hysteresis. In other words, the stage-discharge curve is looped with higher discharges occurring on the rising limb than on the rescission limb of the hydrograph. This is mainly caused by the depth gradient in the flow direction that changes in sign throughout the hydrograph. In practice, this implies that there can be two possible stages for the same discharge. If the stage-discharge relationship is not well known or if it just computed assuming steady state uniform flow,

it may lead to considerable errors when used as downstream boundary condition. That it is why it is often preferred to use the stage hydrograph for that purpose. However, such hydrograph may not be available to study changes in the river and evaluating proposed conditions. For those cases, it is useful to use a stage-discharge relationship, preferably measured over an extensive range of discharges. When this relationship is not available, one option would be to assume steady state flow to determine a single-value rating curve. Since this condition may generate wave reflection that can propagate upstream, it is important to locate the downstream boundary on a reach sufficiently far from the area of interest, therefore minimizing artificial backwater effects. Unfortunately, there is no general way to select such place, but numerical experimenting with the actual model will be necessary to achieve a reasonable location.

NOTE: loop stage-discharge relationships are not implemented in this RiverFlow2D version.

7. On free outflow condition boundaries, the model calculates velocities and water surface elevations applying the full equations from the internal elements. No specific values for velocities or depths are imposed *per se* on these nodes. In practice this is equivalent to assuming that derivatives of water surface elevations and velocities are 0. In subcritical flow situations, it is advisable to use this condition when there is at least another open boundary where WSE or stage-discharge is imposed.

8. When using a single valued stage-discharge condition on internal sections, the model first computes the discharge across the boundary then interpolates the corresponding water surface elevation from the rating table, imposing that value for the next time step for all nodes on the internal boundary. This condition requires providing an ASCII file with the table values entries. See section *Boundary Conditions Data Files* for details on the file format.

9. When imposing a water and sediment discharge, it is required to provide an ASCII file with the time series for water discharge and volumetric sediment discharge for each of the fractions. Note that sediment discharge is always expected in volume per unit time. See section *Boundary Conditions Data Files* for details on the format for multiple-variable boundary condition files.

10. The user must provide a file with the energy slope S_0 for the corresponding boundary. This file will only contain a single value S_0. The model will use S_0, Manning's n, and discharge to create a rating table from which water surface elevations will be imposed as a function of the computed outflow discharge. The rating table is calculated every 0.05 m (0.16 ft.) starting from the lowest bed elevation in the outflow cross section up to 50 m (164 ft.) above the highest bed elevation in the section. If $S_0 = -999$, the model will calculate the average bed slope perpendicular to the boundary line. Please, note than when letting the model calculate the average bed slope, it uses the elevations on the elements adjacent to the boundary line, which may result in adverse slopes or slopes that do not capture the general trend the reach.

11. This boundary condition is similar to the BCTYPE = 6 for inflow water discharge. However, in this case, instead of converting the discharge into velocities that are imposed on all the inflow nodes; the model creates sources on all the elements adjacent to the boundary line. The condition then can be visualized as if the given discharge enters over the inflow elements. For each time, the model evenly divides the discharge between all the inflow elements. For example if there are Ne inflow elements and the imposed discharge is Qin, each element will receive a discharge equal to Qin/Ne. The water volume will naturally flow away from the inflow depending on the bed slopes, etc. Care must be taken when the inflow boundary

elements have lower bed elevations than the surrounding elements. When imposing this condition the user must provide an ASCII file with the discharge time series. See section *Boundary Conditions Data Files* for details on the format for one-variable boundary condition files.

9.2.3 Mesh Boundary Data File: .TBA

`.TBA` file is for internal use by the model and contains the list of boundary nodes in counterclockwise order for the external boundary polygon and in clockwise order for the internal boundaries. **This file is automatically generated by RiverFlow2D**.

Line 1: Start of boundary indicator.
IBOUNDARYID

Line 2: Number of nodes in external boundary of mesh.
NNODESBOUNDARY

NNODESBOUNDARY lines containing the list of boundary nodes in counter clockwise direction.
BOUNDARYNODE (1:NNODESBOUNDARY)

The next lines are only used if there are islands in the mesh.

For each island:

Start of boundary parameter indicator for each island or internal closed contour.
IBOUNDARYID

Number of nodes in island boundary.
NNODESISLANDBOUNDARY

NNODESISLANDBOUNDARY lines containing the list of boundary nodes in clockwise direction.
ISLANDBOUNDARYNODE (1:NNODESISLANDBOUNDARY)

9.2.3.1 Example of a .TBA file

```
-9999
132
1
2
3
173
...
224
175
1
-9999
34
5
...
5
```

In this example the external boundary has 132 nodes and there is one island with 34 nodes.

Table 9.7 – Variable Descriptions for the .DAT File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IBOUNDARYID	I	-9999	-	Always = -9999. This value is used to indicate the start of a new boundary.
NNODESBOUNDARY	I	>0	-	Number of nodes on the mesh external boundary.
BOUNDARYNODE	I	>0	-	Node number on external boundary. See comments 1 and 2.
NNODESISLANDBOUNDARY	I	>0	-	Number of nodes on island boundary.
ISLANDBOUNDARYNODE	I	>0	-	Node number on island boundary.

Note: I = Integer variable.

9.2.3.2 Comments for the .TBA file

1. There should be a single external boundary polygon and any number of internal islands or closed contours.

2. The external boundary should also be the first on the file. The first boundary must always be the external one. The internal boundaries as islands, piers, etc. should follow the external domain polygon.

9.3 Component data

RiverFlow2D provides components to model specific hydraulic problems, including bridge piers, culverts, internal rating tables, weirs, and sediment transport and pollutant transport. This section explains the data file formats used in these components.

9.3.1 Bridges

RiverFlow2D provides four options to account for bridge piers. The most common option is to create the pier plan geometry generating a 2D triangular-cell mesh that represents each pier as a solid obstacle. In that case, the model will compute the flow around the pier and account for the pier drag. This would be the preferred approach when the user needs to know the detailed flow around the piers and the flow does not overtop the bridge deck. However, the resulting mesh may have very small elements, leading to increasing computer times.

The second option (*Bridge Piers*) is a simplified formulation that does not require defining the mesh around the piers, but will compute the pier drag force based on geometric data. This would be the preferred approach when the flow does not overtop the bridge deck and the user does not need to have detailed depiction of the flow around the piers but needs to account for the general effect that the pier would have on the flow.

The third option represented in the *Bridges* component is a comprehensive bridge hydraulics computation tool that does not require capturing bridge pier plan geometry in detail, therefore allowing longer time steps, while allowing calculating the bridge hydraulics accounting for arbitrary plan alignment, complex bridge geometry, free surface flow, pressure flow, overtopping, combined

pressure flow and overtopping, and submergence all in 2D. This is the recommended option for most bridges.

There is a fourth option using the *Internal Rating Table* component, but for most applications it is recommended to use one of the above since they better represent the bridge hydraulics.

9.3.2 Bridges Data File: .BRIDGES

This component requires the `.BRIDGES` data file that is automatically generated by the model based on the geometrical representation entered in the SMS GUI. The `.BRIDGES` file has the following format:

Line 1: Number of bridges.
NUMBEROFBRIDGES

NUMBEROFBRIDGES lines containing the data for each bridge.

Bridge Id.
BRIDGE_ID

Bridge Cross Section Geometry file name.
BRIDGE_GEOMETRY_FILE

Number of cells pairs along bridge alignment.
NC

NUMBEROFCELLS lines containing pairs of cells numbers along bridge alignment.
CELL_A[1] CELL_B[1]
...
CELL_A[NC] CELL_B[NC]

9.3.2.1 Example of a .BRIDGES file

```
1
BRIDGE1
1894878.176 586966.254 1895274.636 586613.844
BRIDGEGEOM.DAT
9
133       1294
131       1296
129       1298
127       1300
125       1302
123       1304
121       1306
119       1308
94       1310
```

Table 9.8 – Variable Descriptions for the .BRIDGES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BRIDGE_GEOMETRY_FILE	S	< 26	-	Contains the geometry of the bridge cross section as explained below.
BRIDGE_ID	S	< 26	-	Bridge ID.
CELL_A[i] CELL_B[i]	I	-	-	Cell pair along bridge alignment.
NC	I	> 0	-	Number of cell pairs along the bridge alignment.
NUMBEROFBRIDGES	I	> 0	-	Number of bridges.

Note: I = Integer variable. S = Text variable.

9.3.3 Bridge Cross Section Geometry Data File

The bridge geometry cross section file is necessary to define the bridge cross section and is defined by four polylines and the fined in five columns as follows:

Line 1: Number of points defining polylines.
NP

NP lines with these entries:
STATION(1) BEDELEV(1) ZLOWER(1) LOWCHORD(1) DECKELEV(1)
...
STATION(NP) BEDELEV(NP) ZLOWER(NP) LOWCHORD(NP) DECKELEV(NP)

The relationship between the four polylines must be as follows:

- For all stations, STATION(I)≤STATION(I+1).
- BEDELEV≤ZLOWER≤LOWCHORD≤DECKELEV .
- In a given line all elevations correspond to the same station.
- The space between BEDELEV and ZLOWER is blocked to the flow.
- The space between ZLOWER and LOWCHORD is open to the flow.
- The space between LOWCHORD and DECKELEV is blocked to the flow.

9.3.3.1 Example of the Cross Section Geometry Data File

The following table is an example one of the geometry file that schematically represents the bridge in Figure 9.2.

NP ⇒ 23

Station ⇩	BedElev ⇩	ZLower ⇩	LowChord ⇩	DeckElev ⇩
0.00	142.00	142.00	142.00	142.00
96.68	125.72	125.72	125.85	142.00
193.37	123.03	123.03	123.32	142.00
290.05	119.86	119.86	120.79	142.00
386.74	110.37	110.37	120.79	142.00
483.42	109.00	109.00	120.79	142.00
580.10	107.58	107.58	120.79	142.00
676.79	106.35	106.35	120.79	142.00
750.00	106.30	106.30	120.79	142.00
750.00	106.30	106.30	106.44	142.00
780.00	106.30	106.30	106.55	142.00
780.00	106.30	106.30	120.79	142.00
870.16	105.18	105.18	120.79	142.00
966.84	106.77	106.77	120.79	142.00
1063.52	107.30	107.30	120.79	142.00
1160.21	116.47	116.47	120.79	142.00
1256.89	116.02	116.02	120.79	142.00
1353.58	116.09	116.09	120.79	142.00
1450.26	119.61	119.61	120.79	142.00
1546.94	121.24	120.92	120.79	142.00
1643.63	124.74	124.74	124.67	142.00
1644.00	142.00	142.00	142.00	142.00

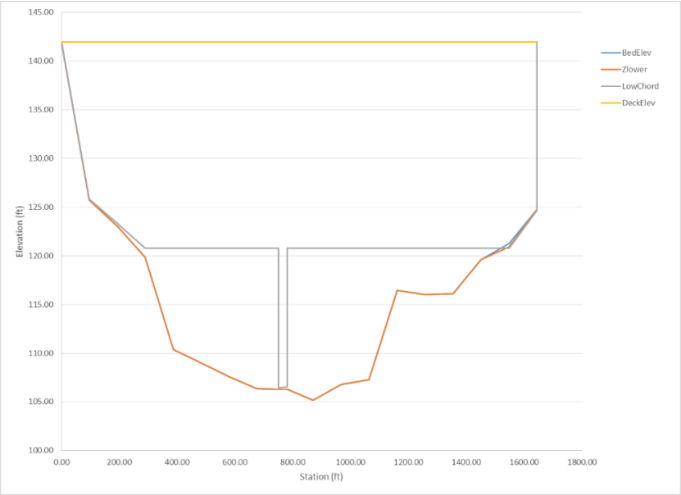


Figure 9.2 – Front view of a bridge cross section.

Table 9.9 – Variable Descriptions for the bridge cross section geometry file.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BEDELEV	R	-	m or ft	Bed elevation. Must be the lowest elevation for all polylines at a given point.
DECKELEV	R	-	m or ft	Elevation of the bridge deck. Must be the highest elevation for all polylines at a given point.
NP	I	-	> 1	Number of points defining cross section polylines.
STATION	R	-	m or ft	Distance from leftmost point defining cross section polyline. All polylines points must have a common station.
ZLOWER	R	-	m or ft	Elevation of lower polyline. ZLOWER must be larger or equal to BEDELEV and smaller or equal to LOWCHORD for a given point. The space between BEDELEV and ZLOWER is a blocked area to the flow. The space between ZLOWER and LOWCHORD is open space. If the bridge has no holes, ZLOWER must be identical to BEDELEV.
LOWCHORD	R	-	m or ft	Elevation of the lower bridge deck. LOWCHORD must be larger or equal to ZLOWER and smaller or equal to DECKELEV for a particular point. The space between LOWCHORD and DECELEV is a blocked area to the flow.

Note: R = Real variable. I = Integer variable.

9.3.4 Bridge Piers Drag Forces File: .PIERS

This option requires the `.PIERS` data file that is automatically generated by the model based on the geometrical representation entered in the SMS GUI. The `.PIERS` data file has the following format:

Line 1: Number of piers.
NUMBEROFPIERS

NUMBEROFPIERS lines containing the data for each pier.
X Y ANGLEX LENGTH WIDTH CD PIERID

9.3.4.1 Example of a .PIERS file

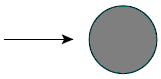
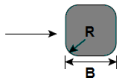
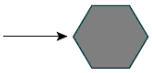
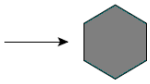
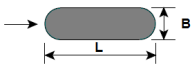
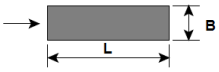
```
124
2042658.82    14214769.48    47.33    19.00    4.00    0.64    P1
2042690.52    14214739.87    46.66    19.00    4.00    0.64    P2
2042786.89    14214649.58    44.88    19.00    4.00    0.64    P3
2042818.95    14214619.53    45.17    19.00    4.00    0.64    P4
2042851.05    14214589.46    45.72    19.00    4.00    0.64    P6
2042883.13    14214559.39    45.10    19.00    4.00    0.64    P7
2042915.40    14214529.15    43.01    19.00    4.00    0.64    P8
2040307.66    14214876.12    0.00     70.00    1.00    0.90    P9
2040328.07    14214790.33    0.00     70.00    1.00    0.90    P10
2040351.38    14214705.48    0.00     70.00    1.00    0.90    P11
2040375.99    14214622.12    0.00     70.00    1.00    0.90    P12
```

Table 9.10 – Variable Descriptions for the .PIERS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ANGLEX	R	$0 - 180$	Deg.	Pier angle with respect to X axis. See comment 1.
C_D	R	$0.5 - 2.5$	-	Non-dimensional drag coefficient of the pier. See comment 2 and **Error! Reference source not found**.
LENGTH	R	-	m or ft	Pier length.
PIERID	S	$< 26'$	-	Name of pier. Should not contain spaces and must have less than 26 characters.
WIDTH	R	-	m or ft	Pier width.
X	R	-	m or ft	X coordinate of pier centroid.
Y	R	-	m or ft	Y coordinate of pier centroid.

Note: R = Real variable. S = Text variable.

Table 9.11 – Drag Coefficients for Bridge Piers. Adapted from Froehlich (2003).

PIER PLAN SHAPE AND APPROACH VELOCITY	DRAG COEFFICIENT C_D
Round cylinder 	1.0
Square cylinder 	2.2
Square cylinder 	1.6
Square cylinder with rounded corners 	R/B: 0 → 2.2; 0.02 → 2.0; 0.17 → 1.2; 0.33 → 1.0
Hexagonal cylinder 	1.0
Hexagonal cylinder 	0.7
Round-nosed section 	L/B: 1 → 1.0; 2 → 0.7; 4 → 0.68; 6 → 0.64
Square-nosed section 	L/B: 1 → 2.2; 2 → 1.8; 4 → 1.3; 6 → 0.9

9.3.4.2 Comments for the .PIERS File

1. Angle ANGLEX applies only to piers that are rectangular in plan. For example ANGLEX = 90 corresponds to a pier whose longest axis is perpendicular to the X-axis.

2. The drag coefficient C_D is related to the drag force though the following formula:

$$F_D = \frac{1}{2} C_D \rho U^2 A_P \tag{9.2}$$

where C_D is the pier drag coefficient, is the water density, U is the water velocity, and A_P is the pier wetted area projected normal to the flow direction.

To account for the drag force that the pier exerts on the flow, RiverFlow2D converts it to the distributed shear stress on the element where the pier centroid coordinate is located. The resulting pier shear stress expressions in x and y directions are as follows:

$$\tau_{px} = \frac{1}{2} C_D \rho U \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right) \tag{9.3}$$

$$\tau_{py} = \frac{1}{2} C_D \rho V \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right) \tag{9.4}$$

where A_e is the element area.

9.3.5 Culverts Data File: .CULVERTS

The culvert component allows accounting for hydraulic structures that convey flow between two locations. The discharge between the structure inflow and outflow ends will be computed based on a user provided hydraulic structure rating table. The model will determine the flow direction based on the hydraulic conditions on the structure ends.

Line 1: Number of culverts.
NCULVERTS

NCULVERTS groups containing
CulvertID
CulvertType
CulvertFile
X1 Y1 X2 Y2
...

9.3.5.1 Example of a .CULVERTS file

2
CulvertA
2
CulvertA.txt
799550.846 309455.307 799363.544 309031.842
CulvertB
1
CulvertB.txt
798858.644 309313.609 799153.441 309004.154

Table 9.12 – Variable Descriptions for the .CULVERTS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CulvertFile	S	< 26	-	Culvert rating table or culvert characteristic file name. See next section for details about the culvert characteristic file. Should not contain spaces and must have less than 26 characters.
CulvertID	S	< 26	-	Culvert name. Should not contain spaces and must have less than 26 characters.
CulvertType	I	$0, 1, 2$	-	Type of culvert. See comments 1 and 2.
NCULVERTS	I	> 0	-	Number of culverts.
X1 Y1 X2 Y2	R	-	m or ft	Coordinates of vertices defining each culvert line.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.5.2 Culvert Depth-Discharge Rating table Data Files for CulvertType=0

This format applies to the culvert depth vs. discharge rating table.

Line 1: Number points in data series
NDATA

NDATA lines containing depth and discharge.
DEPTH(I) Q(I)

Where DEPTH(I) is depth corresponding to discharge Q(I).

INVERT_Z1
INVERT_Z2

Where INVERT_Z1 and INVERT_Z2 are the invert elevations for the inlet and outlet respectively.

9.3.5.3 Example of the Culvert Depth-Discharge Rating Table File

The following example shows a depth-discharge rating table for a culvert. NDATA is 7 and there are 7 lines with pairs of depth and corresponding discharge:

```
7
0     0.20
0.1     1.00
1.00     36.09
2.00     60.00
3.00     84.78
4.00     110.01
100.00     110.02
5.0
1.0
```

Table 9.13 – Variable Descriptions of Culvert Depth-Discharge Data files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
INVERT_Z1	R	> 0	m or ft	Inlet invert elevation. If INVERT_Z1 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet element.
INVERT_Z2	R	> 0	m or ft	Outlet invert elevation. If INVERT_Z2 = -9999, the model makes INVERT_Z2 equal to the average bed elevation of the inlet element.
DEPTH	R	> 0	m or ft	Water depth.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

Note: I = Integer variable. R = Real variable.

9.3.5.4 Culvert Characteristic Data Files for CulvertType = 1, 2

The culvert characteristic data has the following structure:

Nb
Ke
nc
Kp
M
Cp
Y
m
If CulvertType=1
Hb
Base
Else if CulvertType=2
Dc
INVERT_Z1
INVERT_Z2

9.3.5.5 Example of the culvert characteristic data file

1
0.5
0.012
1
1
1.1
0.6
-0.5
0.10
5.0
1.0

This example culvert characteristics data file indicates that the culvert one barrel (Nb =1), Ke=0.4, nc=0.012, Kp=1, cp =1, M =1.1, Y=0.6, m=-0.5, and Dc=0.10, INVERT_Z1=5.0 and INVERT_Z2 = 1.0.

Table 9.14 – Variable Descriptions for the Culvert Characteristic file.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
Nb	I	-	-	Number of identical barrels. The computed discharge for a culvert is multiplied by Nb to obtain the total culvert discharge.
Ke	R	0-1	-	Entrance Loss Coefficient given by Table 9.16.
nc	R	0.01-0.1	-	Culvert Manning's n Coefficient given by Table 9.15.
K'	R	0.1-2.0	-	Inlet Control Coefficient given by Table 9.17.
M	R	0.6-2.0	-	Inlet Control Coefficient given by Table 9.17.

Continued on next page

Table 9.14 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
c'	R	0.6-2.0	-	Inlet Control Coefficient given by Table 9.17.
Y	R	0.5-1.0	-	Inlet Control Coefficient given by Table 9.17.
m	R	0.7,-0.5	-	Inlet form coefficient. m=0.7 for mitered inlets, m=-0.5 for all other inlets.
Hb	R	> 0	m or ft	Barrel Height for box culverts. Only for CulvertType = 1.
Base	R	> 0	m or ft	Barrel Width for box culverts. Only for CulvertType = 1.
Dc	R	> 0	m or ft	Diameter for circular culverts. Only for CulvertType = 2.
INVERT_Z1	R	> 0	m or ft	Inlet invert elevation. If INVERT_Z1 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet.
INVERT_Z2	R	> 0	m or ft	Outlet invert elevation. If INVERT_Z2 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet element.

Note: I = Integer variable. R = Real variable.

Table 9.15 – Manning's n roughness coefficients for various culvert materials. Adapted from Froehlich (2003).

Culvert barrel material	Entrance description	Manning's n n_c
Concrete	Good joints, smooth walls	0.012
	Projecting from fill, square-cut end	0.015
	Poor joints, rough walls	0.017
Corrugated metal	2-2/3 inch × 1/2 inch corrugations	0.025
	6 inch × 1 inch corrugations	0.024
	5 inch × 1 inch corrugations	0.026
	3 inch × 1 inch corrugations	0.028
	6 inch × 2 inch corrugations	0.034
	9 inch × 2 1/2 inch corrugations	0.035

Table 9.16 – Entrance loss coefficients K_e. Adapted from Froehlich (2003).

Type of culvert	Entrance description*	Entrance loss coefficient K_e
Concrete pipe	Projecting from fill, grooved end	0.2
	Projecting from fill, square-cut end	0.5
	Headwall or headwall with wingwalls (concrete or cement sandbags)	
	Grooved pipe end	0.2
	Square-cut pipe end	0.1
	Rounded pipe end	0.7
	Mitered end that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7

Continued on next page

Table 9.16 – continued from previous page

Type of culvert	**Entrance description***	**Entrance loss coefficient K_e**
Corrugated metal pipe	Projecting from embankment (no headwall)	0.9
	Headwall with or without wingwalls (concrete or cement sandbags)	0.5
or pipe-arch	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7
Reinforced concrete box	Headwall parallel to embankment (no wingwalls)	
	Square-edged on three sides	0.5
	Rounded on three sides to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 30° to 75° to barrel	
	Square-edged at crown	0.4
	Crown edge rounded to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 10° to 30° to barrel	
	Square-edged at crown	0.5
	Wingwalls parallel to embankment	
	Square-edged at crown	0.7

See Table 9.18 for inlet configurations.

Table 9.17 – Culvert inlet control formula coefficients. Adapted from Froehlich (2003).

Barrel material	**Barrel shape**	**Inlet description***	**K'**	**M**	**c'**	**Y**
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45 bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7 bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30 to 75 flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90 and 15 flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0 flares ;square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45 flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18 to 33.7 flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; 3/4 inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45 bevels	1.5752	0.6670	1.0101	0.8200
Concrete	Rectangular	Headwall; 33.7 bevels	1.5466	0.6670	0.8107	0.8650
Concrete	Rectangular	Headwall; 45 skew; 3/4 in chamfers	1.6611	0.6670	1.2932	0.7300

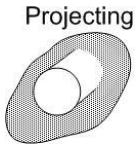
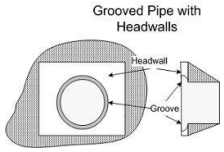
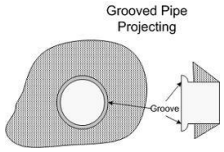
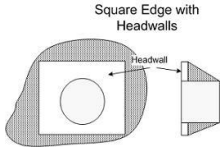
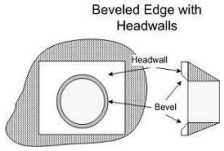
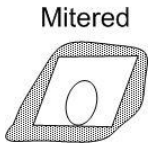
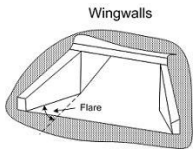
Continued on next page

Table 9.17 – continued from previous page

Barrel material	Barrel shape	Inlet description*	K'	M	c'	Y
Concrete	Rectangular	Headwall; 30 skew; 3/4 in chamfers	1.6961	0.6670	1.3672	0.7050
Concrete	Rectangular	Headwall; 15 skew; 3/4 in chamfers	1.7343	0.6670	1.4493	0.6800
Concrete	Rectangular	Headwall;10-45 skew; 45 bevels	1.5848	0.6670	1.0520	0.7500
Concrete	Rectangular	Wingwalls; non-offset 45/flares	1.5816	0.6670	1.0906	0.8030
Concrete	Rectangular	Wingwalls; non-offset 18.4/flares; 3/4 in chamfers	1.5689	0.6670	1.1613	0.8060
Concrete	Rectangular	Wingwalls; non-offset 18.4/flares; 30/skewed barrel	1.5752	1.2418	0.7100	0.6670
Concrete	Rectangular	Wingwalls; offset 45/flares; beveled top edge	1.5816	0.6670	0.9715	0.8350
Concrete	Rectangular	Wingwalls; offset 33.7/flares; beveled top edge	1.5752	0.6670	0.8107	0.8810
Concrete	Rectangular	Wingwalls; offset 18.4/flares; top edge bevel	1.5689	0.6670	0.7303	0.8870
Cor. metal	Rectangular	Headwall	0.2670	2.0000	1.2192	0.6900
Cor. metal	Rectangular	Projecting; thick wall	0.3023	1.7500	1.3479	0.6400
Cor. metal	Rectangular	Projecting; thin wall	0.4593	1.5000	1.5956	0.5700
Concrete	Circular	Tapered throat	1.3991	0.5550	0.6305	0.8900
Cor. metal	Circular	Tapered throat	1.5760	0.6400	0.9297	0.9000
Concrete	Rectangular	Tapered throat	1.5116	0.6670	0.5758	0.9700
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45 bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7 bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30 to75 flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90 and 15 flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0 flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45 flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18 to 33.7 flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45 bevels	1.5752	0.6670	1.0101	0.8200

See Table 9.18 for inlet configurations.

Table 9.18 – Culvert inlet configurations. (Adapted from `www.xmswiki.com/xms/`).

Inlet configuration	Description
	End of the culvert barrel projects out of the embankment.
	Grooved pipe for concrete culverts decreases energy losses through the culvert entrance.
	This option is for concrete pipe culverts.
	Square edge with headwall is an entrance condition where the culvert entrance is flush with the headwall.
	'Beveled edges' is a tapered inlet edge that decreases head loss as flow enters the culvert barrel.
	Mitered entrance is when the culvert barrel is cut so it is flush with the embankment slope.
	Wingwalls are used when the culvert is shorter than the embankment and prevents embankment material from falling into the culvert.

9.3.5.6 Comments for the .CULVERTS and culvert characteristics files

1. For CulvertType 0, culvert discharge is computed using a given rating table on the CulvertFile file.

2. For CulvertTypes 1 and 2, the model will calculate culvert discharge for inlet and outlet

control using the FHWA procedures (Norman et al.,1985) that were later restated in dimensionless form by Froehlich (2003). See User's Guide for more details.

9.3.6 GATES Data Files: .GATES

This component requires the `.GATES` data file that is automatically generated by the model based on the geometrical representation entered in the SMS GUI. The `.GATES` file has the following format:

Line 1: Number of gates.
NUMBEROFGATES

NUMBEROFGATES lines containing the data for each gate.

Gate Id
GATES_ID

Crest elevation height Cd
CRESTELEV GATEHEIGHT CD

Time series of gate aperture
GATE_APERTURES_FILE

Number of cells pairs along gates alignment
NC

NUMBEROFCELLS lines containing pairs of cells numbers along gate alignment

CELL_A[1] CELL B[1]
...
CELL_A[NC] CELL B[NC]

9.3.6.1 Example of the .GATES File

```
2
Gate2
102.00      2.00       1.720
Gate2.dat
5
3105       29
3103       79
3101       87
3099       137
3097       141
GATE1
111.00      11.00      1.710
GATE1.dat
8
4099       285
4097       283
4033       281
```

4031 279
4029 277
4027 156
4026 82
4024 16

Table 9.19 – Variable Descriptions for the .GATES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CD	R	> 0	-	Gate discharge coefficient.
CRESTELEV	R	> 0	-	Gate crest elevation.
GATE_APERTURES_FILE	S	< 26	-	Gate aperture time series.
GATEHEIGHT	R	> 0	-	Gate height.
GATE_ID	S	< 26	-	Gate ID.
CELL_A[i] CELL B[i]	I	-	-	Cell numbers of cell pairs along gate alignment.
NC	I	> 0	-	Name of pier. Should not contain spaces and must have less than 26 characters.
NUMBEROFGATES	I	> 0	-	Number of cells along the gate alignment.

Note: I = Integer variable. R = Real variable, S = Text variable.

9.3.7 Gate Aperture Time Series File

Line 1: Number of points in time series of gate aperture data.
NPOINTS

NPOINTS lines containing:

Time Aperture.
TIME APERTURE

9.3.7.1 Example of a Gates Aperture Data File

3
0 0.0
2 0.5
4 1.0

Table 9.20 – Variable Descriptions for the .GATES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPOINTS	I	> 1	-	Number of data points in the gate aperture time series.
TIME	R	> 0	$Hrs.$	Time.
H(I)	R	-	$m\text{-}ft$	Gate aperture for the corresponding time.

Note: I = Integer variable. R = Real variable.

9.3.8 Internal Rating Table Data File: .IRT

This data file allows modeling complex hydraulic structures inside the modeling domain. The user would enter polylines coincident with mesh nodes and assign a rating table of discharge vs. water surface elevation to the polyline. In other words, the IRT polylines must connect nodes of the triangular element mesh. For each time step, the model will compute the discharge crossing the polyline and find by interpolation the corresponding water surface elevation from the provided rating table. The model will then impose that water surface elevation to all nodes along the polyline. Velocities will be calculated using the standard 2D equations. Therefore, in internal rating table polylines, computed velocities may not necessarily be perpendicular to the IRT polyline.

The file structure is as follows:

Line 1: Number of internal rating table (IRT) polylines.
IRT_NPL

IRT_NPL line groups containing the IRT polyline ID, the number of vertices defining each polyline, the IRT boundary condition type (always equal to 19 in this version), the Rating Table file name, followed by the list of polyline coordinate vertices as shown:

IRT_ID
IRT_NV IRT_BCTYPE IRT_FILENAME
X_IRT(1) Y_IRT(1)
X_IRT(2) Y_IRT(2)
...
X_IRT(IRT_NV) Y_IRT(IRT_NV)

9.3.8.1 Example of the .IRT file

```
2
IRT_A
4    19    IRT_A.DAT
799429.362    308905.287
799833.895    308354.857
799986.424    307738.111
799847.158    307141.259
IRT_B
4    19    IRT_A.DAT
799482.440    309453.678
799135.525    309118.164
798914.020    309269.634
798787.701    309467.583
```

This file has 2 internal rating table polylines, the ID of the first one is IRT_A, which has 4 vertices, BCTYPE 19 and file name IRT_A.DAT.

Table 9.21 – Variable Descriptions for the .IRT File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IRT_NPL	I	> 0	-	Number of IRT polylines.
IRT_NV	I	≥ 2	-	Number of points defining each IRT polyline.

Continued on next page

Table 9.21 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IRT_ID	S	< 26	-	Name of IRT. Should not contain spaces and must have less than 26 characters.
IRT_BCTYPE	I	19	-	Boundary condition always equals to 19 in this version corresponding to discharge vs. water surface elevation tables. Future versions will include further options.
X_IRT Y_IRT	R	-	m or ft	Coordinates of vertices defining each IRT polyline. See comment 1.
IRT_FILENAME	S	< 26	-	File name containing internal rating table in the format described as a stage-discharge data file. Should not contain spaces and must have less than 26 characters.

Note: I = Integer variable. R = Real variable, S = Text variable.

9.3.8.2 Comments for the .IRT file

1. IRT polylines should be defined avoiding abrupt direction changes (e.g. 90 degree turns). Polyline alignments as such may create errors in the model algorithm that identifies the nodes that lie over the polyline. Therefore, it is recommended that the IRT follow a more or less smooth path.

9.3.9 Mud and Debris Flow Data File: .MUD

This file only applies to the RiverFlow2D Plus model. This file provides the parameters necessary to model mud and debris flow using the RiverFlow2D Plus model.

Line 1: Flow resistance relation.
MF_FRR

Line 2: Yield stress.
MF_YS

Line 3: Bingham viscosity.
MF_BVIS

Line 4: Internal friction angle.
MF_THETA

Line 5: Material density.
MF_DENS

9.3.9.1 Example of the .MUD file

1
300.

0.00899
3.5
2200.

Table 9.22 – Variable Descriptions for the .MUD File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
MF_BVIS	I	> 0	Poise or lb/in^2	Bingham viscosity.
MF_DENS	R	> 0	kg/m^3 or lb/ft^3	Material density.
MF_FRR	R	1-7	-	Flow resistance relation (see Table 9.23).
MF_THETA	R	> 0	Degrees	Internal friction angle.
MF_YS	R	> 0	N/m^2 or lb/in^2	Yield stress.

Note: I = Integer variable. R = Real variable.

9.3.9.2 Comments for the .MUD file

1. The equations that describe the tangential forces generated by the stresses can be lumped into the same mathematical formula despite of having a different nature. In this way several shear stresses formulations can be considered: turbulent stress τ_t, dispersive stress τ_d, Coulomb-type frictional stress τ_f, yield stress τ_y and viscous stress τ_μ. Table 9.23 includes turbulent-collisional, yield stress and laminar flow resistance terms in a quadratic rheological law, where a standard value of $\kappa = 24$ is assumed. In this table is also included the input parameters which are required by its computation.

Table 9.23 – Flow resistance relation.

MF_FFR	Flow Resistance Relation	Flow resistance term	Input parameters
1	Turbulent (default)	$\tau_b = \tau_t$	Manning's n, density
2	Full Bingham	$\tau_b = \tau_0$ with $f_1(\tau_b, \tau_0) = 0$	Bingham viscosity, yield stress, density
3	Simplified Bingham	$\tau_b = 1.5\tau_y + 3\tau_\mu$	Bingham viscosity, yield stress, density
4	Turbulent and Coulomb	$\tau_b = \tau_t + \tau_f$	Manning's n, density, Friction angle
5	Turbulent and Yield	$\tau_b = \tau_t + \tau_y$	Manning's n, density, yield stress
6	Turbulent, Coulomb and Yield	$\tau_b = \tau_t + \min(\tau_y, \tau_f)$	Manning's n, density, yield stress, friction angle
7	Quadratic	$\tau_b = \tau_t + \tau_y + \frac{\kappa}{8}\tau_\mu$	Manning's n, density, yield stress, viscosity
8	Granular flow	$\tau_b = \tau_f$	Friction angle, density. For this formula, the model automatically sets Manning's $n = 0$ for all cells.

9.3.10 Oil Spill on Land Data File: .OILP

This file only applies to the OilFlow2D Plus model. This file provides the parameters necessary to model flow of viscous fluids including oil over complex terrain using the OilFlow2D model.

Line 1: Flow resistance relation.
OL_FRR

Line 2: Yield stress. Not used in this release.
OL_YS

Line 3: Fluid viscosity.
OL_BVIS

Line 4: Internal friction angle. Not used in this release.
OL_THETA

Line 5: Oil density.
OL_DENS

Line 6: Temperature time series file.
OL_TEMPTSERIES

Line 7: Temperature - Viscosity - Density table file.
OL_TEMPVISCDENS

9.3.10.1 Example of the .OILP file

3
1.
0.00899
1
2200.
Temptseries.txt
Tempviscdensetable.txt

Table 9.24 – Variable Descriptions for the .OILP File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
OL_BVIS	I	> 0	$Pa \cdot s$ or $lb - s/in^2$	Fluid viscosity that will be used if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
OL_DENS	R	> 0	kg/m^3 or lb/ft^3	Fluid density that will be used if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
OL_FRR	R	3	-	Flow resistance relation. Always should be equal to 3 in this release.
OL_TEMPTSERIES	S	≤ 26	-	Temperature time series file.

Continued on next page

Table 9.24 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
OL_TEMPVISCDENS	S	≤ 26	-	Temperature-viscosity-density table file. The model will use the temperature for a given time to interpolate in the table the viscosity and density.
OL_THETA	R	> 0	Degrees	Not used in this release.
OL_YS	R	> 0	N/m^2 or lb/in^2	Yield stress. Not used in this release.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.10.2 Example of the Temperature Time Series file

```
7
0.0    60
2.0    65
4.0    70
6.0    75
8.0    77
10.0    79
12.0    65
```

In this example there are 7 entries in the Temperature Time series. For the initial time 0.0 hours the temperature is $60°F$, and so on.

9.3.10.3 Example of the Temperature-Viscosity-Density Table file

```
2
50    1.38E-06    58.00
120    6.94E-07    56.00
```

In this example there are 2 lines in the Temperature-Viscosity-Density table file. For $50°F$ the viscosity is 1.38E-06 $lb - s/in^2$ and the density 50 lb/ft^3.

9.3.11 Pollutant Transport Module Data File: .SOLUTES

The pollutant transport module calculates concentration of passive or reactive pollutants (solutes) based on advection and dispersion. The required date is included in the `.SOLUTES` data file that has the following format:

Line 1: Number of pollutants (solutes)
NPOLLUTANTS_MAX

Line 2: Number of pollutants used in this run
NPOLLUTANTS_USED

Line 3: List of solutes used (all in one line)
I_1 I_2 ... I_NPOLLUTANTS_USED

Line 4: Longitudinal and transversal dispersion coefficients
DISPL DISPT

Line 5...: List of names of pollutants (one in each line)
NAME_I1
NAME_I1
...
NAME_NPOLLUTANT_MAX

Line 6 to NPOLLUTANT_MAX+6: First order reaction rate coefficient matrix K(I,J)
K(1,1) K(1,2) ... K(1,NPOLLUTANT_MAX)
K(2,1) K(2,2) ... K(2,NPOLLUTANT_MAX)
...
K(NPOLLUTANT_MAX,1) K(NPOLLUTANT_MAX,2) ... K(NPOLLUTANT_MAX,NPOLLUTANT_MAX)

9.3.11.1 Example of the .SOLUTES file

```
5
5
1  2  3  4  5
0.1  0.02
NITRATE
AMMONIUM
TEMPERATURE
CHLORINE
PHOSPHATE
0.2  0.0  0.0  0.0  0.0
0.0  0.1  0.0  0.0  0.0
0.0  0.0  1.1  0.0  0.0
0.0  0.0  0.0  0.2  0.0
0.0  0.0  0.0  0.0  0.5
```

Table 9.25 – Explanation of the example .SOLUTES file.

DATA	DESCRIPTION
5	There are five pollutants
5	Five pollutants will be used in this run
1 2 3 4 5	The active pollutants will be 1 through 5
0.1 0.02	Longitudinal dispersion coefficient is 0.1 and transversal = 0.02.
D	Pollutant No. 1 name is: NITRATE
AMMONIUM	Pollutant No. 2 name is: AMMONIUM
TEMPERATURE	Pollutant No. 3 name is: TEMPERATURE
CHLORINE	Pollutant No. 4 name is: CHLORINE
PHOSPHATE	Pollutant No. 5 name is: PHOSPHATE
0.2 0.0 0.0 0.0 0.0	First order reaction constant for pollutant No. 1 is 0.2
0.0 0.1 0.0 0.0 0.0	First order reaction constant for pollutant No. 2 is 0.1
0.0 0.0 1.1 0.0 0.0	First order reaction constant for pollutant No. 3 is 1.1
0.0 0.0 0.0 0.2 0.0	First order reaction constant for pollutant No. 4 is 0.2
0.0 0.0 0.0 0.0 0.5	First order reaction constant for pollutant No. 5 is 0.2

Table 9.26 – Variable Descriptions for the .SOLUTES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
DISPL	R	> 0	m^2/s or ft^2/s	Longitudinal dispersion coefficient.
DISPT	R	> 0	m^2/s or ft^2/s	Transversal dispersion coefficient.
K(I,J)	R	-	s^{-1}	First-order reaction constants, where K(I,I) is the reaction constant for pollutant I and K(I,J) the reaction constant of pollutant I with pollutant J.
NPOLLUTANTS_MAX	I	$0, 1, 2$	-	Maximum number of pollutants.
NPOLLUTANTS_USED	I	> 0	-	Number of pollutants used in the present run. Should be equal to NPOLLUTANTS_MAX

Note: I = Integer variable. R = Real variable.

9.3.12 Pollutant Transport Module Initial Concentration Data File: .CINITIAL

The initial concentration data file is created by when the dataset with the initial concentrations is exported from SMS based on the data entered as Initial Concentrations.

Each column in this file corresponds to each of the pollutants indicated in the corresponding `.SOLUTES` file. Each line corresponds to the initial concentrations for each cell. Therefore the `.CINITIAL` file must have NPOLLUTANTS_MAX columns and NELEM lines.

9.3.13 Manning's n Variable with Depth Data File: .MANNN

This file is created by SMS based on the data you enter in the ManningsNz coverage. It is used account for spatially distributed Manning's n variable with depth data.

Line 1: Number of zones defined by polygons where Manning's n variable with depths are defined.
NNZONES

NRZONES group of lines containing Manning's n variable with depth data file for each zone
MANNNFILE

Number of vertices of polygon i
NPZONE[i]

List of NPZONE[i] vertex coordinates
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.13.1 Example of the .MANNN file

2
Manning1.txt

```
4
25.0    25.0
25.0    75.0
75.0    75.0
75.0    25.0
Manning2.txt
4
25.0    125.0
25.0    175.0
75.0    175.0
75.0    125.0
```

In this example, there are two polygons. The Manning's n data file for the first polygon is `Manning1.txt` and the polygon is defined by four vertices.

Table 9.27 – Variable Descriptions for the .MANNN File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NNZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NNZONES	I	-	-	Number of zones.
MANNNFILE	S	≤ 26	-	Manning's n file. See comment 1.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.13.2 Comments for the .MANNN file

1. The spatial distribution of Manning's n variable with depth is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygon (complementary area) would be assigned the default Manning's n file.

9.3.13.3 Manning's n variable with depth data file

Line 1: Number of points in Manning's n file.
NPRE

ND lines containing:

DEPTH MANNINGS N

9.3.13.4 Example of a Manning's variable with depth data file

```
3
0.    0.1
0.3   0.1
1.0   0.03
```

Table 9.28 – Variable Descriptions for the Manning's n Cariable with Depth Data File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
DEPTH	R	≥ 0	m or ft	Flow depth. See comment 1.
MANNINGS N	R	≥ 0	-	Manning's n corresponding to DEPTH. See comment 1.
ND	I	-	-	Number values in file.

Note: I = Integer variable. R = Real variable.

9.3.13.5 Comments for the Mannign's n variable with depth data file

1. To calculate the Manning's n over the mesh, the model will first identify the polygon over each cell and then will use the interpolated n value for cell depth from the table corresponding to the polygon. In the example above, for all depth between 0.3 and 1, Manning's n will be obtained by linear interpolation between 0.1 and 0.03.

2. The user should provide a "DefaultManningsn.dat" file in the project folder and the program will apply the data contained in that file to the complementary area to the polygons provided. If the "DefaultManningsn.dat" does not exist, the model will apply a default value of 0.035 to the areas not covered by Manning's n polygons.

9.3.14 Rainfall And Evaporation Data File: .LRAIN

Use this file to enter spatially distributed and time varying rainfall and evaporation data. The model assumes that the rainfall and evaporation can vary over the modeling area.

Line 1: Number of zones defined by polygons where rainfall time series are defined.
NRZONES

NRZONES group of lines containing hyetograph and evaporation data file for each zone
RAINEVFILE[i]

Number of vertices of polygon i
NPZONE[i]

List of NPZONE[i] vertex coordinates
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.14.1 Example of the .LRAIN file

```
2
hyeto1.txt
4
25.0     25.0
25.0     75.0
75.0     75.0
75.0     25.0
hyeto2.txt
4
```

25.0 125.0
25.0 175.0
75.0 175.0
75.0 125.0

In this example, there are two polygons. The rainfall and evaporation data file for the first polygon is `hyeto1.txt` and the polygon is defined by four vertices.

Table 9.29 – Variable Descriptions for the .LRAIN File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NRZONES	I	-	-	Number of zones.
RAINEVFILE	S	≤ 26	-	Rainfall intensity. See comment 1.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.14.2 Comments for the .LRAIN file

1. The spatial distribution of rainfall and evaporation is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons would have no rainfall or evaporation imposed onto the mesh.

9.3.14.3 Hyetograph and Evaporation data file

Line 1: Number of points in time series of rainfall and evaporation.
NPRE

NPRE lines containing:

Time Daily Rainfall, Daily Evaporation.
TIME RAINFALL EVAPORATION

9.3.14.4 Example of a Hyetograph and Evaporation data file

3
0. 0.0 1.0
24 4.0 2.0
48 12.0 3.0

Table 9.30 – Variable Descriptions for the Hyetograph and Evaporation Data File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
EVAPORATION	R	≥ 0	mm/hr or in/hr	Evaporation intensity. See comment 1.
NPRE	I	-	-	Number of times in rainfall and evaporation time series.
RAINFALL	R	≥ 0	mm/hr or in/hr	Rainfall intensity. See comment 1.
TIME	R	> 0	hours	Time interval

Note: I = Integer variable. R = Real variable.

9.3.14.5 Comments for the Hyetograph and Evaporation data file

1. To calculate the rainfall/evaporation over the mesh, the model will use the interpolated rainfall and evaporation intensities between each time interval. For instance in the example above, for all times between 0 and 24 hours the rainfall will be determined by linear interpolation between 0 mm/hr and 4 mm/hr, and so on for evaporation.
2. If the user has a "DefaultRainEvap.dat" file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.15 Infiltration Data File: .LINF

Use this file to enter spatially distributed infiltration parameters.

Line 1: Number of zones defined by polygons where infiltration parameters are defined.
NIZONES

NIZONES group of lines containing:

Infiltration data file for each zone
INFILFILE

Number of vertices of polygon i
NPZONE[i]

List of NPZONE[i] vertex coordinates
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.15.1 Example of the .LINF file

```
2
inf1.inf
4
0.0     0.0
0.0     200.0
200.0     200.0
200.0     0.0
Inf2.inf
4
200.0     200.0
400.0     200.0
400.0     0.0
200.0     0.0
```

In this example, there are two polygons. The infiltration data file for the first polygon is `inf1.inf` and the polygon is defined by four vertices.

Table 9.31 – Variable Descriptions for the .LINF File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NIZONES	I	-	-	Number of zones. See Comments 1 and 2.
INFILFILE	S	≤ 26	-	Infiltration parameter file.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.15.2 Comments for the .LINF file

1. The spatial distribution of infiltration parameters is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons would have no infiltration loss calculated.

2. Each polygon can have a different infiltration method assigned.

3. If the user has a "DefaultInfiltration.dat" file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.15.3 Infiltration parameters data file

Line 1: Model to calculate infiltration.
INFILMODEL

Line 2: Number of infiltration parameters.
NIPARAM

If INFILMODEL = 1: Horton method then:
Line 3: **K $\mathbf{f_c}$ $\mathbf{f_0}$**

If INFILMODEL = 2: Green and Ampt method then:
Line 3: **KH PSI DELTATHETA**

If INFILMODEL = 3: SCS-CN method then:
Line 3: **CN POTRETCONST FUTUREUSE FUTUREUSE**

9.3.15.4 Example of a Infiltration parameter data file

1
3
8.3E-04 3.47E-06 2.22E-5

In this example the infiltration loss method is set to 1 corresponding to the Horton model. There are 3 parameters as follows: K = 8.3E-04, f_c = 3.47E-06 and f_0 = 2.22E-5.

Table 9.32 – Variable Descriptions for the Infiltration Parameter File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CN	R	> 0	-	Curve Number. See USDA (1986) to determine adequate values depending on land cover. Typical values range from 10 for highly permeable soils to 99 for paved impermeable covers.
DeltaTheta	R	> 0	-	Difference between saturated and initial volumetric moisture content. Default value = 3E-5.
f_c	R	[0,5E-4]	m/s or ft/s	Final infiltration rate. Default = 2E-5.
f_0	R	[0,5E-4]	m/s or ft/s	Initial infiltration rate. Default = 7E-5.
INFILMODEL	I	1,2,3	-	Infiltration method. 1: Horton, 2: Green and Ampt, 3: SCS-CN.
K	I	[0,30]	$1/s$	Decay coefficient used in Horton method. Default = 1.
Kh	I	≥ 0	m/s or ft/s	Hydraulic conductivity used in Green and Ampt method. Default = 0.00001.
NIPARAM	I	[3-4]	-	Number of data parameters depending on the infiltration model selected. Should be set as follows: 3 for Horton of Green and Ampt methods, 4 for SCS-CN method.
POTRETCONST	R	[0-1]	-	Potential maximum retention constant. Typically = 0.2.
PSI	R	[0-1]	m or in	Wetting front soil suction head. Default = 0.05.

Note: I = Integer variable. R = Real variable.

9.3.16 RiverFlow2D Plus Sediment Transport Data Files: .SEDS and .SEDB

I This file provides the parameters necessary to model sediment transport using the RiverFlow2D Plus model.

9.3.16.1 .SEDS file for suspended sediment data

The `.SEDS` file provides the data for the suspended sediment transport model.

Line 1: Suspended sediment activation switch.
ISSACT

Line 2: Number of suspended sediment fractions.
NSSNFRAC

Line 3: Equilibrium concentration formula.
ISSTF

Line 4: Sediment density for each fraction.
SSDEN(1) ... SSDEN(NSSNFRAC)

Line 5: Initial suspended sediment concentration for each fraction.
INICON(1) ... INICON(NSSNFRAC)

Line 6: For future use.
DUMMY1(1) ... DUMMY1(NSSNFRAC)

Line 7: Suspended sediment D50 size for each fraction.
D50(1) ... D50(NSSNFRAC)

Line 8: For future use.
DUMMY2(1) ... DUMMY2(NSSNFRAC)

Line 9: Porosity for each fraction.
SSPOR(1) ... SSPOR(NSSNFRAC)

Line 10: Critical Shield Stress for each fraction.
THETAC(1) ... THETAC(NSSNFRAC)

Line 11: Friction angle for each fraction.
FRICANG(1) ... FRICANG(NSSNFRAC)

Line 12: Equilibrium concentration formula factor for each fraction.
SSTFACT(1) ... SSTFACT(NSSNFRAC)

Line 13: Settling velocity formula.
ISETFOR

Line 14: Settling velocity formula factor.
SETFORFACT

9.3.16.2 Example of the .SEDS file

```
1
2
1
165.00 165.00
0.01 0.01
0.003937008 0.003937008
0.0039370 0.0039370
0.003937008 0.003937008
0.40 0.40
0.047 0.047
35.00 35.00
1.00 1.00
1
1
```

Table 9.33 – Variable Descriptions for the .SEDS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
INICON	R	0,0.3	-	Initial volumetric sediment concentration. See comment 1.
ISSACT	I	0,1	-	Suspended sediment activation switch. If ISSACT = 1, the model will compute suspended sediment transport. If ISSACT = 0 suspended sediment transport will not be calculated.
NSSNFRAC	I	1-10	-	Number of suspended sediment fractions.
ISSTF	I	1,2	-	Equilibrium concentration formula. This code indicates the formula according to this list: 1: Bagnold (1966) 2: Van Rijn (1984a)
DUMMY1	R	> 0	-	Not used in this release, but must be present.
D50	R	> 0	mm or in	Sediment median size. 50% of the sediment is finer than D50. See comment 1.
DUMMY2	R	> 0	-	Not used in this release, but must be present.
SSPOR	R	0.3-0.6	-	Porosity. See comment 1.
THETAC	R	0.03-0.06	-	Critical Shield Stress. See comment 1.
FRICANG	R	5-45	-	Friction angle. See comment 1.
ISETFOR	I	0-9	-	Settling velocity formula. It is a unique formula for all fractions. This code indicates the formula according to this list: 1: Rubey (1983) 2: Zhang (1961) 3: Zanke (1977) 4: Van Rijn (1984a) 5: Raudkivi (1990) 6: Julien (1998) 7: Cheng (1997) 8: Jimenez-Madsen (2003) 9: Wu-Wong (2006)
SETFORFACT	R	-	-	Settling velocity formula factor. This factor multiplies the settling velocity calculated by the formula selected in ISETFOR. It's a factor that may be used for calibrating the model.
SSDEN	R	-	kg/m^3 or lb/ft^3	Suspended sediment density. See comment 1.
SSTFACT	R	-	-	Equilibrium concentration formula factor for each fraction. This factor multiplies the equilibrium concentration formula ISSTF. It's a factor that may be used for calibrating the model. See comment 1.

Note: I = Integer variable. R = Real variable.

9.3.16.3 Comments for the .SEDS file

1. There should be one value for each sediment fraction up to NSSNFRAC.

9.3.16.4 .SEDB file for bed load transport data

The `.SEDB` file provides the data for the bed load transport model.

Line 1: Bed load sediment transport activation switch.
IBLACT

Line 2: Number of bed load sediment fractions.
NBLNFRAC

Line 3: Sediment transport formula.
IBLTF

Line 4: Sediment density for each fraction.
BLDEN(1) ... BLDEN (NBLNFRAC)

Line 5: Sediment D30 size for each fraction.
D30(1) ... D30(NBLNFRAC)

Line 6: Sediment D50 size for each fraction.
D50(1) ... D50(NBLNFRAC)

Line 7: Sediment D90 size for each fraction.
D90(1) ... D90(NBLNFRAC)

Line 8: Porosity for each fraction.
BLPOR(1) ... BLPOR(NSSNFRAC)

Line 9: Critical Shield Stress for each fraction.
THETAC (1) ... THETAC (NBLNFRAC)

Line 10: Friction angle for each fraction.
FRICANG (1) ... FRICANG (NBLNFRAC)

Line 11: Fractions on the bed.
BEDFRACT (1) ... BEDFRACT (NBLNFRAC)

Line 12: Transport formula factor for each fraction.
BLFORFACT(1) ... BLFORFACT (NBLNFRAC)

Line 13: Coupled or uncoupled computation switch.
ICOUPLED

9.3.16.5 Example of the .SEDB file

```
1
2
2
165.0 165.0
0.0039370 0.0039370
0.0039000 0.0039370
0.0039370 0.0039370
0.40 0.40
0.047 0.047
35.00 35.00
0.50 0.50
1.00 1.00
0
```

Table 9.34 – Variable Descriptions for the .SEDS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IBLACT	I	0,1	-	Bed load transport activation switch. If IBLACT = 1, the model will compute bed load sediment transport. If IBLACT = 0 bed load transport will not be calculated.
ICOUPLED	I	0,1	-	Bed load transport activation switch. If ICOUPLED = 0 the model will run in coupled mode, where the bed load transport will be computed together with the hydrodynamic model for each time step. If ICOUPLED = 1 the model will run the bed load transport uncoupled from the hydrodynamic model.
NBLNFRAC	I	1–10	-	Number of sediment fractions.
IBLTF	I	1–10	-	Coupled or uncoupled computation switch. This code indicates the formula according to this list: 1: Meyer-Peter & Muller (1948) 2: Ashida (1972) 3: Engelund (1976) 4: Fernandez (1976) 5: Parker fit to Einstein (1979) 6: Smart (1984) 7: Nielsen (1992) 8: Wong 1 (2003) 9: Wong 2 (2003) 10: Camenen-Larson (2005)
D30	R	> 0	m or in	Sediment D30 size. 30% of the sediment is finer than D30. Only used for Smart Formula. See comment 1.
D50	R	> 0	m or in	Sediment median size. 50% of the sediment is finer than D50. See comment 2.
D90	R	> 0	m or in	Sediment D90 size. 90% of the sediment is finer than D90. Only used for Smart Formula. See comment 1.
BLPOR	R	0.3–0.6	-	Porosity. See comment 1.

Continued on next page

Table 9.34 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
THETAC	R	0.03–0.06	-	Critical Shields Stress. See comment 3.
FRICANG	R	5–45	-	Friction angle. See comment 1.
BEDFRACT	R	-	-	Sediment fraction. The sum of all fractions should add to 1. See comment 1.
BLDEN	R	-	kg/m^3 or lb/ft^3	Sediment density. See comment 1.
BLFORFACT	R	-	-	Transport formula factor for each fraction. This factor multiplies the result of the transport formula selected (IBLTF). It's a factor that may be used for calibrating the model. See comment 1.

Note: I = Integer variable. R = Real variable.

9.3.16.6 Comments for the .SEDB file

1. There should be one value for each sediment fraction up to NBLNFRAC.
2. Characteristic diameter for all sediment transport formulas.
3. The default critical Shields stress is 0.047.

9.3.17 Sources and Sinks Data File: .SOURCES

Use this file to enter data to simulate point inflows or outflows at any location. This feature is typically used when modeling intakes (outflow) or point inflows. The user may provide time varying hydrographs that will be applied to each point.

Line 1: Number of source and sink points.
NSOURCES

NSOURCES groups of lines containing source/sink point identification text, name of the file containing the discharge time series, and the coordinates of the point as follows:

SOURCEID
ISFILENAME
X_S(I) Y_S(I)
...

9.3.17.1 Example of the .SOURCES file

2
IntakeA
Intake.txt
799019.633 309402.572
DischargeIn
Discharge.txt
799222.740 309048.493

This file has 2 sources/sinks. The first one is named IntakeA located at coordinate: X = 799019.633 and Y = 309402.572 and discharge data is on file: `Intake.txt`.

Table 9.35 – Variable Descriptions for the .SOURCES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NSOURCES	I	> 0	-	Number of source or sink points.
ISFILENAME	S	-	-	Name of file containing the time series of each point source or sink. Must not contain blank spaces. See comments 1 and 2.
SOURCEID	S	< 26	-	Name of point source or sink. Should have less than 26 characters and must not contain blank spaces.
X_S Y_S	R	-	m or ft	Coordinates of source/sink.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.17.2 Comments for the .SOURCES File

1. The file name is arbitrary but must not contain blank spaces. The file format is the same as the *one variable boundary condition* file described in Section 9.6.1.

2. To model inflows use positive discharge values and to model outflows use negative values.

9.3.18 Weirs Data File: .WEIRS

This data file allows using weir calculations along user defined polylines representing road or weir overtopping. The user selects the weir coefficient associated with each weir and the model will determine the nodes on each polyline and the discharge across each pair of nodes based on the weir formulae and methods described on Hydraulics of Bridge Waterways FHWA, 1978 (see comment 1). The present version allows defining a variable crest elevation along the weir polyline.

Line 1: Number of weir polylines.
NWEIRS

NWEIRS group of lines including weir ID, number of vertices defining each weir polyline, the weir coefficient followed by the coordinates each vertex as shown:

WEIR_ID
NV CF WRCRESTELEV
X_W(1) Y_W(1) WRCREST(1)
X_W(2) Y_W(2) WRCREST(2)
...
X_W(NV) Y_W(NV) WRCREST(NV)

9.3.18.1 Example of the .WEIRS file

```
2
WEIR_A
4    3.0    -999
799429.362    308905.287    200.
799833.895    308354.857    201.
799986.424    307738.111    202.
799847.158    307141.259    203.
WEIR_B
4    3.04    -999
```

799482.440 309453.678 203.5
799135.525 309118.164 204.0
798914.020 309269.634 204.9
798787.701 309467.583 205.0

This file has 2 weirs. The first one is named WEIR_A and is defined by a polyline with 4 vertices. Weir discharge coefficient is equal to 3.0.

Table 9.36 – Variable Descriptions for the .WEIRS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CF	R	> 0	-	Weir coefficient. See comment 1.
NWEIRS	I	> 0	-	Number of weir polylines.
NV	I	≥ 2	-	Number of points defining each weir polyline.
WEIR_ID	S	< 26	-	Name of weir. Should have less than 26 characters and must not contain blank spaces.
WRCRESTELEVR		-	m of ft	Weir crest elevation for all the weir. If WRCRESTELEV = -999 a weir elevation is provided for each weir polyline vertex.
WRCREST(I)	R	-	m of ft	Weir crest elevation for vertex I.
X_W(I) Y_W(I)	R	-	m of ft	Coordinates of vertices defining each weir polyline. See comment 2.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.18.2 Comments for the .WEIRS File

1. Weir discharge is computed between pairs of nodes along the polyline based on the following formula:

$$Q = C_f L H^{3/2} \tag{9.5}$$

where L is the distance between nodes, H is the total head upstream of the polyline segment and C_f is the discharge coefficient. The model checks for submergence and it occurs C_f will be corrected according to the correction factor defined by (FHWA, 2001).

2. Weir polylines should be defined avoiding abrupt direction changes (e.g. ≥ 90 degree turns), because such angles may create errors in the algorithm that identifies the nodes that lie over the polyline.

9.3.19 Wind Data File: .WIND

Use this file to enter spatially distributed and time varying wind velocity data. The model assumes that the wind velocity can vary over the modeling area. The user should provide a set of polygons and a time series of velocities for each polygon.

Line 1: Number of zones defined by polygons where wind velocity time series are defined.
NWZONES

Line 2: Wind stress coefficient.
CD

Line 3: Air density.
AIRDENSITY

NWZONES group of lines containing hyetograph and evaporation data file for each zone.
WINDFILE

Number of vertices of polygon i.
NPZONE[i]

List of NPZONE[i] vertex coordinates.
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.19.1 Example of the .WIND file

```
2
0.009
1.225
Wind1.txt
4
25.0 25.0
25.0 75.0
75.0 75.0
75.0 25.0
Wind2.txt
4
25.0 125.0
25.0 175.0
75.0 175.0
75.0 125.0
```

In this example, there are two polygons. The C_d coefficient is set to 0.009 and the wind density to 1.225 kg/m^3. The wind velocity file for the first polygon is `wind1.txt` and the polygon is defined by four vertices.

Table 9.37 – Variable Descriptions for the .WIND File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
AIRDENSITY	R	≥ 0	-	Air density. Always given in metric units. Default = 1.225.
CD	R	≥ 0	-	Wind stress coefficient. Always given in metric units. Default = 0.009.
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NWZONES	I	-	-	Number of zones.
WINDFILE	S	≤ 26	-	Wind velocity vector time series file. See Comment 1.
X(I) Y(I)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.3.19.2 Comments for the .WIND File

1. The spatial distribution of wind is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons will be considered as having no wind stress.

2. If the user has a "DefaultWind.dat" file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.19.3 Wind Velocity Data File

Line 1: Number of points in time series of wind velocity data.
NPOINTS

NPOINTS lines containing:

Time Wind velocity component in X and Y directions.
TIME UX UY

9.3.19.4 Example of a Wind Velocity and Data File

```
3
0.    0.0    0.0
24    4.0    -3.0
48    4.0    -3.0
```

Table 9.38 – Variable Descriptions for the Wind Velocity File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPOINTS	I	> 1	-	Number of data points in the wind velocity time series.
TIME	R	> 0	Hrs	Time.
UX(I) UY(I)	R	-	m/s or ft/s	Wind velocity components in x and y directions.

Note: I = Integer variable. R = Real variable.

9.4 Output control data

9.4.1 Observation Points Data File: .OBS

This file contains data to allow the model reporting time series of results at user specified locations based on point coordinates. The user would indicate the number of observation points and then the list of point coordinates.

Line 1: Number of observation points.
NOBSPOINTS

NOBSPOINTS groups of lines containing the observation point ID, and coordinate of each point:
ObsID
X_OP(I) Y_OP(I)
...

9.4.1.1 Example of the .OBS file

3
PointA
798798.380 309627.950
PointB
799146.926 309430.876
PointC
799721.8608 309041.615

This `.OBS` file has three points. The first point is named PointA and has coordinates: X=798798.380 Y=309627.950.

Table 9.39 – Variable Descriptions for the .OBS File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ObsID	S	< 26	-	Name of observation point. Should have less than 26 characters and must not contain blank spaces.
NOBSPOINTS	I	> 0	-	Number of observation points.
X_OP Y_OP	R	-	m or ft	Coordinates of point.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.4.2 Graphical Output Control Data File: .PLT

This file provides parameters to control graphic output options for plots generated while the model is running.

Line 1: Plot control variables.
IGRAPHCODE COLORSCHEME IAXES IDXF IGRAPHFILES IVSF

Line 2: Velocity vector scale multiplier.
SF_MULT

Line 3: Coordinates for plot window.
XMING XMAXG YMING YMAXG

Line 4: Limits of plotted variable.
MINVARG MAXVARG

Line 5: Maximum velocity to plot.
MAXVELOC

Line 6: Transparency.

USEBACKIMAGE

Line 7: Transparency.
TRANSP

Line 8: Background aerial image.
IMAGEFILE

Line 9: Background aerial image world file.
IMAGEWF

9.4.2.1 Example of the .PLT file

```
100    5    1    0    1    0
5
0    0    0    0
0    0
7
1
0.6
C:\Projects\Example\Aerial.gif
C:\Projects\Example\Aerial.gwf
```

Table 9.40 – Variable Descriptions for the .PLT File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
COLORSCHEME	I	1-6	-	Variable to select plot option. See comment 1.
IAXES	I	0,1	-	Switch to control weather to plot axes. 0: Do not plot X and Y axes. 1: Plot X and Y axes.
IDXF	I	0,1	-	Switch to control velocity field output in DXF CAD format. 0: Do not output DXF mesh and velocity field. 1: Create mesh and velocity field DXF files for each output time.

Continued on next page

Table 9.40 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IGRAPHCODE	I	100, 101, 102, 103, 110, 201, 202, 203, 204, 600-610	-	Parameter to indicate the plot type to display while the program is running. 100: Plot velocity field using vectors in black. 101: Plot velocity field using vectors in black over depths. 102: Plot velocity field over bed elevations. 103: Plot water elevations. 110: Plot velocity field using vectors in color. 201: Plot depths. 202: Plot bed elevations. 203: Plot velocity field over water elevations. 204: Plot bed elevation changes. 600-610: Plot suspended sediment or pollutant concentrations.
IGRAPHFILES	I	0,1	-	Variable to control whether to output graphic files. 0: Do not output graphic files. 1: Output graphic files.
IMAGEFILE	S	-	-	Name of aerial image file including path and extension. Supported formats include `.BMP`, `.GIF`, `.PNG` and `.TIFF`. Other graphic file formats will be included in forthcoming releases.
IMAGEWF	S	-	-	Name of aerial image world file including path and extension. World file format should follow ESRI specifications. See comment 2.
IVSF	I	0,1	-	Switch to control velocity field output in a file that allows creating shapefiles. 0: Do not output SF velocity field. 1: Create velocity field SF files for each output time. See comment 3.
MAXVELOC	R	-	m/s or ft/s	Use this variable to control the maximum velocity displayed in vector plots. If MAXVELOC = 0, the whole velocity range will be plotted. If MAXVELOC > 0, it will define the maximum velocity to be displayed.
MINVARG, MAXVARG	R	-	-	These variables define the minimum and maximum values to be displayed of the selected variable. If equal to 0, the maximum range will be displayed.
SF_MULT	R	> 1	-	Variable to control velocity vector scale. Use this variable to adjust velocity vectors. Velocities will be scaled according to SF_MULT.
TRANSP	R	[0,1]	-	Variable to control plot transparency when using a background image. TRANSP should be in the range from 0.0 to 1.0, where 0.0 means a fully transparent color and 1.0 means a fully opaque color.

Continued on next page

Table 9.40 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
USEBACKIMAGE	I	[0,1]	-	Variable to controls whether to use a background image for dynamic plots during model run. If value is = 1, the plot will include as background the image provided in IMAGEFILE and IMAGEWF.
XMING, XMAXG, YMING, YMAXG	R	-	m or ft	These variables indicate the coordinates of a rectangle that define the plot window. If all values are 0, the full extent of the modeling area will be displayed.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.4.2.2 Comments for the .PLT file

1. COLORSCHEME defines the color palette that will be used for all plots. The available palettes are shown in this figure:

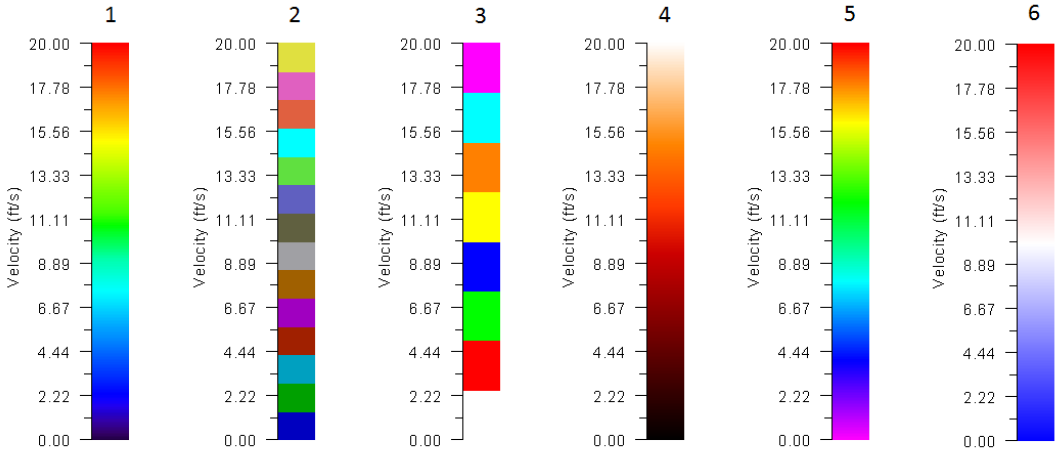


Figure 9.3 – Color palettes.

2. The generic meaning of world file parameters are:

   Line 1: A: pixel size in the x-direction in map units/pixel

   Line 2: D: rotation about y-axis (ignored in this version)

   Line 3: B: rotation about x-axis (ignored in this version)

   Line 4: E: pixel size in the y-direction in map units, almost always negative

   Line 5: C: x-coordinate of the center of the upper left pixel

   Line 6: F: y-coordinate of the center of the upper left pixel.

   Example:

   2.05
   0.00
   0.00

-2.05
795944.99
310049.73

In this example, 2.05 is the pixel size in x-direction, rotation in x and y axes is 0.00, pixel size in y direction is 2.05 (shown in negative), x-coordinate of upper left pixel is 795944.99 and y-coordinate of upper left pixel is 310049.73.

The following table indicates the supported image formats and their corresponding world file extensions.

Table 9.41 – Supported image formats and their corresponding world file extensions.

IMAGE FILE FORMAT	WORLD FILE EXTENSION
.BMP	.BMPW, .BPW
.GIF	.GFW, .GIFW, .WLD
.PNG	.PGW, .PNGW, .WLD
.TIF, .TIFF	.TFW, .WLD

3. Vector field, depth and water surface elevation text files are used for creating vector field shapefiles. Using the IVSF = 1 option, will allow the model to generate ESRI Shapefile support files for each report interval. This option is not used in the RiverFlow2D SMS.

9.4.3 Data for Profile Result Output: .PROFILES

Use this file to provide profiles (polylines) along which results will be generated.

Line 1: Number of profiles.
NPROFILES

NPROFILES group of files including: Profile ID, number of vertices in profile I, the number of intervals to divide each profile, and coordinates for each vertex in polyline.
PROFILEID
NVERTICES_PR(I) ND_PR
X_PRF(I), Y_PRF(I)
...

9.4.3.1 Example of the .PROFILES file

```
2
ProfileA
2 10
800500.45 306895.63
799095.07 307457.34
ProfileB
3 10
800503.45 306896.63
799500.00 306900.00
799095.07 307457.34
```

This file indicates there are 2 profiles. First profile ID is: ProfileA which is defined with a 2-vertex polyline and will be divided in 10 segments.

Table 9.42 – Variable Descriptions for the .PROFILES File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ND_PR	I	> 2	-	Intervals to divide each profile sub-segment between vertices. Results will be reported at each interval.
NPROFILES	I	> 0	-	Number of profiles.
NVERTICES_PR(I)	I	> 1	-	Number of vertices in each profile.
PROFILEID	S	< 26	-	Profile name. Should have less than 26 characters and must not contain blank spaces.
X_PRF(I,J), Y_PRF(I,J)	R	-	m or ft	Coordinates of each vertex J in profile I.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.4.4 Cross Section Data for Result Output File: .XSECS

Cross sections are used to output numeric results at user defined lines on the mesh.

Line 1: Number of cross sections.
NCROSS_SECTIONS

NCROSS_SECTIONS groups of lines containing the cross section ID, the number of vertices defining the cross section (always equal to 2), the number of intervals to divide the cross section and the list of coordinates of initial and final point in cross section:

XSECID
NPXSEC ND_CS
X1_CS(I) Y1_CS(I)
X2_CS(I) Y2_CS(I)

9.4.4.1 Example of the .XSECS file

3
CrossSectionA
2 40
800500.45 306895.63
799095.07 307457.34
CrossSectionB
2 40
800492.17 307163.36
799171.99 307594.56
CrossSectionC
2 40
800449.99 307404.31
799223.97 307690.20

This `.XSECS` file indicates there are 3 cross sections. The first one has ID = CrossSectionA and will be divided in 40 segments.

Table 9.43 – Variable Descriptions for the .XSECS File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NCROSS_SECTIONS	I	> 0	-	Number of cross sections.
ND_CS	I	> 2	-	Cross section will be divided in ND_CS segments. Results will be reported at each segment. See comment 1.
NPXSEC	I	2	-	Number of points defining cross section. In the present version only the two extreme points are allowed to define the cross section, therefore this value should always be 2.
X1_CS, Y1_CS, X2_CS, Y2_CS	R	-	m or ft	Coordinates of initial and ending point of each cross section.
XSECID	S	< 26	-	Cross section name. Should have less than 26 characters and must not contain blank spaces.

Note: I = Integer variable. R = Real variable. S = Text variable.

9.4.4.2 Comments for the .XSECS File

1. The model will cut the mesh using the cross section line and extract results at the division points. If ND_CS is too small, the program may not capture anything in between the divisions, and the computed cross section discharges may have big errors.

9.5 Elevation data

9.5.1 X Y Z data with header

These files contain scattered data in the format suitable to import it in the SMS data layer, for example the BedElevations data layer. It usually has `.EXP` extension, but can have any other file extension provided that the format is as described herein. Each point is identified by its X and Y coordinates and the elevation value for that coordinate.

Line 1: Number of points and number of parameters (header)
NUMBER OF DATA POINTS NUMBER OF PARAMETERS

NUMBER OF DATA POINTS lines with X, Y and parameters data.
X(POINT) Y(POINT) P1(POINT) P2(POINT) ... PN(POINT)

9.5.1.1 Example of an .EXP File

```
11086    1
798439.73    306063.87    160.00
798477.04    309506.95    201.10
798489.45    309522.30    200.93
798498.09    306222.29    162.00
798504.45    305915.63    160.00
798511.71    306075.55    161.00
798516.09    309412.73    201.74
798517.37    309592.42    163.14
...
```

In this example `.EXP` file, there are 11086 elevation data points, one parameter per point (the elevation for each point).

Table 9.44 – Variable Descriptions for the .EXP File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NUMBER OF DATA POINTS	I	> 0	-	Number of data points in the file.
NUMBER OF PARAMETERS	I	> 0	-	Number of parameters for each point. In the case of the elevation data file this value is equal to 1.
X	I	-	m or ft	X Coordinate of each elevation point. See comment 1.
Y	R	-	m or ft	Y Coordinate of each elevation point. See comment 1.
P	R	-	m or ft	Parameter value. See comment 2.

Note: I = Integer variable. R = Real variable.

9.5.1.2 Comments for the .EXP Data File

1. X and Y coordinates may be given in either meters or feet, depending on the units being used in the project. Coordinate system should always correspond to plane projection. RiverFlow2D does not support geographical coordinates in Latitude/Longitude format. If

the available data is in Lat/Lon format, it will need to be converted to UTM or plane Cartesian coordinates before importing coordinates into SMS . Presently RiverFlow2D does not provide tools to make this conversion.

2. Elevation values should be given in the same units as the corresponding coordinates.

9.5.2 General X Y Z data

General XYZ data can be imported using PANEL, space or comma delimited ASCII files or ASCII Grid Files using the PIE/Import.../Import Data command.

9.6 Boundary conditions data

9.6.1 One Variable Boundary Condition Files

This format applies to the following data files:

- Time vs. Water Surface Elevation (BCTYPE = 1)
- Time vs. Discharge (BCTYPE = 6)
- Time vs. U velocity component in x direction (BCTYPE = 2)
- Time vs. V velocity component in y direction (BCTYPE = 3)

Note: BCTYPE parameter is described in Table 7.

Line 1: Number points in data series.
NDATA

NDATA lines containing
TIME(I) VARIABLE(I)

Where VARIABLE(I) is WSE, Q, U or V, depending on the boundary condition **BCTYPE**.

9.6.1.1 Example of the Boundary Condition File for One Variable Time Series

The following example shows an inflow hydrograph where NDATA is 7 and there are 7 lines with pairs of time and discharge:

```
7
0.    20.
1.    30.
1.3    50.
2.    90.
4.    120.
5.    200.
7.    250.
```

Table 9.45 – Variable Descriptions of Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	Hrs	Time in hours. The time interval is arbitrary.
VARIABLE	R	-	-	Represents Water Surface Elevation, Discharge, U or V velocity components depending on the boundary condition.

Note: I = Integer variable. R = Real variable.

9.6.2 Two Variables Boundary Condition Files

This format applies to the following data files:

- Time vs. U and V velocity components (BCTYPE = 4)
- Time vs. Discharge and Water Surface Elevation (BCTYPE = 5)
- Time vs. U velocity component and Water Surface Elevation (BCTYPE = 7)
- Time vs. V velocity component and Water Surface Elevation (BCTYPE = 8)

Line 1: Number points in data series.
NDATA

NDATA lines containing time and two values.
TIME(I) VARIABLE1(I) VARIABLE2(I)

Where VARIABLE1(I) and VARIABLE2(I) depend on the boundary condition type as follows:

BCTYPE	VARIABLE1	VARIABLE2
4	U	V
5	Q	WSE
7	U	WSE
8	V	WSE
26	Q	Qs

9.6.2.1 Example of the Two-Variable Boundary Condition File

The following example shows a file for BCTYPE=5 where discharge and WSE are given, NDATA is 10 and there are 10 lines with pairs of time, discharge and WSE:

```
10
0.    20.    1420.
1.    30.    1421.5
1.3    50.    1423.
...
7.    250.    1420.
8.1    110.    1426.
```

10. 60. 1423.5
20. 20. 1421.

Table 9.47 – Variable Descriptions of Two-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	Hrs	Time in hours. The time interval is arbitrary.
VARIABLE1	R	-	-	Represents Water Surface Elevation, Discharge, U or V velocity components depending on the boundary condition.
VARIABLE2	R	-	-	Represents Water Surface Elevation, Discharge, U or V velocity components depending on the boundary condition.

Note: I = Integer variable. R = Real variable.

9.6.3 Multiple-Variable Boundary Condition Files

This format applies to the following data file:

- Time vs. Q water discharge and Qs sediment discharge (BCTYPE = 26)

Line 1: Number points in data series.
NDATA

NDATA lines containing time and two values.
TIME(I) VARIABLE1(I) VARIABLE2(I) ... VARIABLEN(I)

Where VARIABLE1(I) ... VARIABLEN(I) depend on the boundary condition type as follows:

BCTYPE	VARIABLE1	VARIABLE2
26	Q	Qs

9.6.3.1 Example of the Multiple-Variable Boundary Condition File

The following example shows a file for BCTYPE=26 where water discharge and sediment discharge for two fractions are given, NDATA is 10 and there are 10 lines with pairs of time, discharge and WSE:

10
0. 20. 0.001 0.002
1. 30. 0.002 0.005
1.3. 50. 0.003 0.010
...
7. 250. 0.01 0.015
8.1 110. 0.005 0.009
10. 60. 0.004 0.007

20. 20. 0.003 0.005.

Table 9.49 – Variable Descriptions of Multiple-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	*Hrs*	Time in hours. The time interval is arbitrary.
VARIABLE1	R	-	-	Represents Water Discharge.
VARIABLE2..N	R	-	-	Represents Sediment Discharge for the given fraction.

Note: I = Integer variable. R = Real variable.

9.6.4 Stage-Discharge Data Files

This format applies to the stage (water surface elevation) vs. discharge table used for BCTYPE = 9 and 19.

Line 1: Number points in data series.
NDATA

NDATA lines containing stage and discharge.
STAGE(I) Q(I)

Where STAGE(I) is water surface elevation and Q(I) is the corresponding discharge.

9.6.4.1 Example of the Stage-Discharge Boundary Condition File

The following example shows a stage-discharge rating table where NDATA is 21 and there are 21 lines with pairs of stage and corresponding discharge:

```
21
-1.00    0.00
-0.75    1.79
-0.50    5.20
-0.25    9.45
0.00    14.23
0.25    19.37
0.50    24.76
0.75    30.36
1.00    36.09
1.25    41.95
1.50    47.89
1.75    53.92
2.00    60.00
2.25    66.14
2.50    72.31
2.75    78.53
3.00    84.78
3.25    91.05
3.50    97.35
```

3.75 103.67
4.00 110.01

Table 9.50 – Variable Descriptions of Two-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
STAGE	R	> 0	m or ft	Water surface elevation.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

Note: I = Integer variable. R = Real variable.

9.6.5 Culvert Depth-Discharge Data Files

This format applies to the culvert depth vs. discharge table.

Line 1: Number points in data series.
NDATA

NDATA lines containing depth and discharge.
DEPTH(I) Q(I)

Where DEPTH(I) is depth corresponding to discharge Q(I).

9.6.5.1 Example of the Culvert Depth-Discharge File

The following example shows a depth-discharge rating table for a culvert. NDATA is 7 and there are 7 lines with pairs of depth and corresponding discharge:

7
0 0.20
0.1 1.00
1.00 36.09
2.00 60.00
3.00 84.78
4.00 110.01
100.00 110.02

Table 9.51 – Variable Descriptions of Culvert Depth-Discharge Data Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
DEPTH	R	> 0	m or ft	Water depth.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

Note: I = Integer variable. R = Real variable.

10 Output File Reference

RiverFlow2D generates results in many output ASCII text files. These files can be easily accessed with any text editor and they can be imported into SMS program and other GIS software for visualization and analysis. RiverFlow2D always creates also output ASCII files in both English and metric units depending on the units provided in the data files.

10.1 Output File Overview

The following table summarizes the output files generated by RiverFlow2D:

Table 10.1 – List of Output Data Files.

RESULTS	DATA FILE EXTENSION	DESCRIPTION
GENERAL OUTPUT		
Run control parameters, components used, etc..	`.OUTI`, `.OUTE`	Echoes input data read from files including modeling control parameters, mesh data, boundary conditions, and for each report time interval inflow and outflow discharges and velocities `.OUTI` file is in metric units and `.OUTE` file in English units.
Triangular element mesh information	`.MESHOUTI`, `.MESHOUTE`	These files provide comprehensive information about the triangular element mesh. `.MESHOUTI` is in metric units and `.MESHOUTE` file in English units.
Run progress results	`.ROUT`	This file report for each output interval the computer time, average time step, inflow and outflow water and sediment transport discharge at open boundaries and volume and mass conservation errors.
General results tabular output	`.UVHI`, `.UVHE`	For each output interval nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number are written to file `.UVHI` is in metric units and `.UVHE` in English units.

Continued on next page

Table 10.1 – continued from previous page

RESULTS	DATA FILE EXTENSION	DESCRIPTION
Maximum values tabular output	.MAXI, .MAXE	For each output interval maximum nodal velocity modules, depths, and are written to file .MAXI is in metric units and .MAXE in English units.
Time series at observation points	.OUTI, .OUTE	These files report time series of results at observation points. The model finds the nodes closest to the provided coordinates and the files report time series of velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge. File name format is as follows: RESvsT_NODENUMBER.OUTI for metric units and RESvsT_NODENUMBER.OUTE for English units.
Mass balance	.MBAL	Generated only in the RiverFlow2D FE model to report total inflow and outflow discharges and volume change for each output interval.
Hot start	2binitialized. HOTSTART	Generated in the RiverFlow2D model to restart a simulation from previously computed results. The file contains the time in seconds and the corresponding stateN.out file name from which the model will restart when using the hot start option.
Inundation area	.WETAREAS	These files report the area of inundation (wet area) calculated as the sum of areas of all wet elements. See comment 1.
PROFILE AND CROSS SECTION OUTPUT		
Cross section output	.XSECI, .XSECE	For all output intervals, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number, water and sediment discharge. .XSEC is in metric and .XSECE in English units.
Cross section hydrographs	.XSECH, .XSECSED	Report a hydrograph table for each cross section. .XSECH is the water hydrograph and .XSECSED the sediment flux hydrograph for each cross section. See Comment 1.
Profile output	.PRFI, .PRFE	For each output interval and for a number of points along user defined polylines, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number are written to file .PRFI is in metric and .PRFE in English units.
COMPONENT OUTPUT		
Culverts	.OUT	Output discharge at every culvert for each report interval. File name format is as follows: CULVERT_culvertID.OUT where culvertID is the user provided name.
Internal Rating Tables	.OUT	Output discharge at every IRT for each report interval.File name format is as follows: IRT_irtID.OUT, where irtID is the text provided by the user to identify the Internal Rating Table.
Weirs	.WEIRI, .WEIRE	Report results for weirs. .WEIRI is in metric units and WEIRE in English units.
GIS SUPPORT OUTPUT		

Continued on next page

Table 10.1 – continued from previous page

RESULTS	DATA FILE EXTENSION	DESCRIPTION
Spatial distribution of results in GIS format	.EXP	For each output interval nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number, and sediment transport fluxes, are written to file. These files can be directly imported into GIS programs. The file names are TIME_METRIC_DDDD_HH_MM_SS.EXP for metric units and TIME_ENG_DDDD_HH_MM_SS.EXP for English units.
Maximum values in GIS format	.EXP	These ASCII files report maximum values of velocity module, depth and water surface elevations. The files named as follows: For English units: <ProjectName>_ENG_MAX.EXP. For metric units: <ProjectName>_METRIC_MAX.EXP. See Comment 1.
Shapefile support files	.EXP	Support files that allow generating ESRI Shapefiles for velocity vectors, depths and water surface elevations. See Comment 1.
GRAPHIC OUTPUT		
Mesh	.DXF	When the export to DXF is activated the program will export the mesh in DXF format to the file: <ProjectName>_MESH.DXF. See comment 1.
Velocity vectors	.DXF	Velocity vectors in AutoCAD DXF format. See comment 1.
Various	.GIF	Various outputs in graphic raster GIF format. See comment 1.
AQUAVEO' SMS OUTPUT		
General results	.HDF5	RiverFlow2D model will output .HDF5 binary files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge for each output time interval. .HDF5 files can be used by SMS and other post-processing software to visualize RiverFlow2D results. See comment 1.
Visualization Toolkit Files		
General results	.vtk	When the Create Graphic Output Files check box is selected in the Graphic Output Panel, RiverFlow2D Plus model and RiverFlow2D Plus GPU will output .vtk files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of RiverFlow2D results. ParaView (www.paraview.org) is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities.

10.1.0.1 Comments for Output Files

1. In the RiverFlow2D Plus and Plus GPU models these files are only generated when post processing results using the Plot RiverFlow2D results on the *Data Input Program Graphic Options* panel. Please see the User's Guide for more detail.

10.2 General Output Files

This section describes content of each output file.

10.2.1 Run Options Summary

These files replicate the input data read from files including modeling control parameters, mesh data, boundary conditions, and inflow and outflow discharges and velocities for each output interval. `.OUTI` is in metric units and `.OUTE` in English units. Part of a typical `.OUTI`/`.OUTE` file format is shown below:

10.2.2 Mesh Data and Mesh Metrics

Mesh data is written to files with extensions: `.MESHOUTI` (metric units) and `.MESHOUTE` (English units). These files provide comprehensive information about the triangular element mesh. The following table summarizes the available output.

Table 10.2 – Variables reported on the .MESHOUTI and .MESHOUTE Files.

VARIABLE	DESCRIPTION
Number of elements	Total number of elements in the mesh
Number of nodes	Total number of nodes in the mesh
X	x-coordinate of node
Y	y-coordinate of node
BEDEL	Node initial bed elevation
INITIAL_WSE	Node initial water surface elevation
BC ID	Boundary condition code
BC File	Boundary condition file name
Node1, Node2, Node3	Nodes numbers of each element in counterclockwise order
Manning's n	Manning' n roughness coefficient
Area	Element/cell area
Angle	Minimum angle in element/cell
Total mesh area	Sum of areas of all elements/cells on the mesh
Average element area	Total mesh area divided by number of elements/cells
Average element size	Average size of elements/cells on mesh
Element with minimum area	Smallest element
Minimum element size	Approximate linear size of smallest element/cell
Minimum element area	Area of smallest element/cell
Element with maximum area	Largest element/cell
Maximum element size	Approximate linear size of largest element/cell
Maximum element area	Area of largest element/cell
Minimum element angle	Smallest element/cell internal angle
Element with minimum angle	Element/cell that has the smallest internal angle

This file also reports the list of acute cells that have an internal angle of less than 22.5 degrees. If there are acute cells, the model will give an error message and will not be able to execute.

Part of a typical file format is shown below:

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                   Build    2014.01.02
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                   (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                             ALL RIGHTS RESERVED
                             RUN DATE: 20/JAN/2014
================================================================================

================================================================================
            TIME CONTROL OPTIONS
================================================================================
  Simulation time                :     2.000 hrs.
  Report output interval         :     0.100 hrs.
  Use variable time-steps         : OFF.
  Time-step                      :     1.000 s.
  Start simulation               : Time = 0.
  Manning n multiplier           :      1.00
  Selective Lumping Parameter    :      0.90
  Input units                    : English.
  Depth tolerance for dry bed    :     0.300 ft.
  Wetting-drying method          : B.

================================================================================
            COMPONENT OPTIONS
================================================================================
  Culverts                   : OFF.
  Internal rating tables     : OFF.
  Piers                      : OFF.
  Rainfall/evaporation       : OFF.
  Sediment Transport         : OFF.
  Pollutant Transport        : OFF.
  Sources/Sinks              : OFF.
  Weirs                      : OFF.

================================================================================
            OUTPUT OPTIONS
================================================================================
  Output for cross sections : OFF.
  Output for profile        : OFF.
  Text output               : ON.
  Graphical output          : ON.

================================================================================
            INITIAL CONDITIONS
================================================================================
  Initial dry bed.

================================================================================
            PARALLEL COMPUTATION
================================================================================
 Model used  0 processors/cores with  0 theads.

================================================================================
            MESH DATA
================================================================================
 Number of elements/cells       :     1702
 Number of nodes                :      919

 Total mesh area                :    161.53 Acres
 Average element area           :   4135.77 ft2.
 Average element size (approx.) :     90.95 ft.
 Element with minimum area      :       192
 Minimum element size (approx.) :     73.37 ft.
 Minimum element area           :   2691.63 ft2.
 Minimum element area           :      0.06 Acres
 Element with maximum area      :       667
 Maximum element size (approx.) :    106.85 ft.
 Maximum element area           :   5708.34 ft2.
 Maximum element area           :      0.13 Acres
 Minimum element angle          :     39.69
 Element with minimum angle     :       279
 Maximum bed elevation          :    188.69
 Minimum bed elevation          :    149.75
```

```
================================================================================
              OPEN BOUNDARIES
================================================================================
  Number of Open Boundaries:    2

   Number of nodes on open boundary  1:       6
   node   bc type  bc file name
      2       6     QIN.DAT
      3       6     QIN.DAT
     23       6     QIN.DAT
     21       6     QIN.DAT
     19       6     QIN.DAT
     17       6     QIN.DAT

   Number of nodes on open boundary  2:      20
   node   bc type  bc file name
      5      12     0.0025
      6      12     0.0025
     41      12     0.0025
    124      12     0.0025
    141      12     0.0025
    162      12     0.0025
    164      12     0.0025
    166      12     0.0025
    168      12     0.0025
    170      12     0.0025
    172      12     0.0025
    174      12     0.0025
    176      12     0.0025
    178      12     0.0025
    180      12     0.0025
    182      12     0.0025
    160      12     0.0025
     48      12     0.0025
     11      12     0.0025
      9      12     0.0025

  Boundary    2 has uniform flow condition.

    Cross section for boundary No.    2

 Node  Distance Bed elevation
            ft.        ft.
          5      0.00    160.80
          6     99.14    161.20
         41    198.27    161.85
        124    297.41    162.05
        141    396.54    162.00
        162    495.68    161.40
        164    594.81    157.20
        166    693.95    154.80
        168    793.08    152.99
        170    892.22    156.09
        172    991.35    156.71
        174   1090.49    157.45
        176   1189.62    159.94
        178   1288.76    160.52
        180   1387.89    160.78
        182   1487.03    162.20
        160   1586.17    162.60
         48   1685.30    161.87
         11   1784.44    165.00
          9   1883.57    165.60

    Calculated Rating Table for boundary No.    2 using Slope = .250E-02 is:
         WSE              Q
          ft.            cfs
       153.150          0.409
       154.790        244.593
       156.431       1264.688
       158.071       4064.665
       159.712       9428.131
       161.352      14858.572
       162.993      25204.231
       164.633      47577.938
       166.273      74645.159
       167.914     109762.625
       169.554     150031.525
```

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                   Build    2014.01.02
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                 (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                           ALL RIGHTS RESERVED
                           RUN DATE: 21/JAN/2014
================================================================================

================================================================================
           MESH DATA
================================================================================
 Number of elements/cells       :   470328
 Number of nodes                :   235682

   Node        X           Y       BEDEL    Initial WSE  BCType      BC File Name
               (ft)        (ft)      (ft)       (ft)
      1  1354873.46    70353.15    744.00     744.00      0               0
      2  1354870.94    70358.53    744.00     744.00      0               0
      3  1354870.95    70353.65    744.00     744.00      0               0
...
 235680  1355362.35    69365.86    753.00     753.00      0               0
 235681  1355393.07    69373.33    753.00     753.00      0               0
 235682  1355422.53    69380.48    753.00     753.00      0               0

  ELEMENT CONECTIVITY AND MANNING's n

  Element    Node1    Node2    Node3  Manning's n    Area        Angle
                                                    ft2         deg.
       1        1        2        3     0.027          6.11     24.95
       2        4        5        6     0.021          5.06     37.60
       3        7        8        9     0.021          4.28     30.56
  ...
  470326   233886   233885   234125     0.015         44.87     37.96
  470327   233886   234125   233792     0.015         45.49     37.50
  470328   233886   233792   233884     0.015         42.04     43.72

================================================================================
 Total mesh area                :    935.67 Acres
 Average element area           :     86.70 ft2.
 Average element size (approx.) :     13.17 ft.
 Element with minimum area      :     59246
 Minimum element size (approx.) :      1.21 ft.
 Minimum element area           :      0.73 ft2.
 Minimum element area           :      0.00 Acres
 Element with maximum area      :    452423
 Maximum element size (approx.) :     59.82 ft.
 Maximum element area           :   1789.11 ft2.
 Maximum element area           :      0.04 Acres
 Minimum element angle          :     22.68
 Element with minimum angle     :    456512
 Maximum bed elevation          :    865.00
 Minimum bed elevation          :    636.70

 ================================================================================
  ACUTE ELEMENT REPORT (Elements with an internal angle less than 22.5 degrees.)

================================================================================

  The mesh has no acute angle elements.

================================================================================
              EXTERNAL BOUNDARY NODES (COUNTERCLOCKWISE)
================================================================================
  Number of Nodes on Boundary:          913
    Node        X             Y
                ft.           ft.
      40   1345765.970     66480.270
      41   1345775.791     66472.959
...
     383   1353535.918     70972.028
     251   1353532.384     70953.376
```

10.2.3 Run Summary

Run summary report is written to file with extension: `.ROUT`. These files report for each output interval the computer time, average time step, and for each open boundary inflow (positive) or outflow (negative) discharge (m^2/s or ft^3/s), volume conservation error (%), volumetric sediment discharge (m^2/s or ft^3/s) and sediment mass conservation error (%)

10.2.3.1 Example of the .ROUT file

```
================================================================================
================================================================================
        RiverFlow2D Plus - Release 4.2.1
                           Build Jul 28 2015
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2015 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 02/Sep/2015
================================================================================

================================================================================
                          MODEL RUN REPORT
================================================================================
    TIME          CPU Time       Ave.DT        Open Boundary          Q      Vol. Error      Qs       Mass Error
                                   (s)                               (cfs)      (%)          (cfs)       (%)
0000:00:06:00   0000:00:00:00     4.000            01                0.00   0.000e-000       0.00     0.000e-000
                                                   02                0.00   0.000e-000       0.00     0.000e-000
                                                   03               -0.00   0.000e-000      -0.00     0.000e-000
0000:00:12:00   0000:00:00:00     4.000            01                0.00   0.000e-000       0.00     0.000e-000
                                                   02                0.00   0.000e-000       0.00     0.000e-000
                                                   03               -0.00   0.000e-000      -0.00     0.000e-000
0000:07:24:00   0000:00:08:59     0.104            01              894.84   1.552e-014      -1.80     1.552e-014
                                                   02              894.84   1.552e-014       0.76     1.552e-014
                                                   03            -1876.58   1.552e-014      -2.84     1.552e-014
0000:07:30:00   0000:00:09:11     0.103            01              838.03   0.000e+000      -1.73     0.000e+000
                                                   02              838.03   0.000e+000       0.76     0.000e+000
                                                   03            -1782.05   0.000e+000      -2.40     0.000e+000
0000:07:36:00   0000:00:09:23     0.102            01              777.48   3.390e-014      -1.11     3.390e-014
                                                   02              777.48   3.390e-014       0.74     3.390e-014
                                                   03            -1656.45   3.390e-014      -2.09     3.390e-014
0000:07:42:00   0000:00:09:35     0.102            01              719.20   8.900e-014      -1.06     8.900e-014
                                                   02              719.20   8.900e-014       0.71     8.900e-014
                                                   03            -1533.72   8.900e-014      -1.92     8.900e-014
0000:07:48:00   0000:00:09:45     0.101            01              692.86   7.390e-014      -0.56     7.390e-014
                                                   02              692.86   7.390e-014       0.70     7.390e-014
                                                   03            -1442.47   7.390e-014      -1.71     7.390e-014
0000:07:54:00   0000:00:09:55     0.101            01              649.77   0.000e+000      -0.55     0.000e+000
                                                   02              649.77   0.000e+000       0.65     0.000e+000
                                                   03            -1380.99   0.000e+000      -1.69     0.000e+000
0000:08:00:00   0000:00:10:05     0.100            01              623.81   1.985e-014      -0.63     1.985e-014
                                                   02              623.81   1.985e-014       0.67     1.985e-014
                                                   03            -1294.89   1.985e-014      -1.58     1.985e-014
```

10.2.4 Tabular Output Files

These files report nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number, and sediment transport fluxes for each output interval. `.UVHI` is in metric units and `.UVHE` in English units.

A typical output `.UVHE` file is shown below:

```
================================================================================
================================================================================
        RiverFLO-2D Plus - Release 4.1
                         Build Jan 22 2014
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                   (C) COPYRIGHT 2009-2014 Hydronia, LLC.
                            ALL RIGHTS RESERVED
                            RUN DATE: 27/Jan/2014
================================================================================
```

NODE	Ux (m/s)	Uy (m/s)	VELOCITY (m/s)	WSEL (m)	DEPTH (m)	BEDEL_ORI (m)	BEDEL (m)	DELTA_BED (m)	DRY-WET	FROUDE	QSX (m2/s)	QSY (m2/s)	QS (m2/s)
1	0.000	0.000	0.000	4.302	0.000	4.302	0.000	0.000	DRY	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	4.435	0.000	4.435	0.000	0.000	DRY	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	4.701	0.000	4.701	0.000	0.000	DRY	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	3.941	0.000	3.941	0.000	0.000	DRY	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	4.445	0.000	4.445	0.000	0.000	DRY	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	3.345	0.000	3.345	0.000	0.000	DRY	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	4.530	0.000	4.530	0.000	0.000	DRY	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	3.751	0.000	3.751	0.000	0.000	DRY	0.000	0.000	0.000	0.000
9	0.000	0.000	0.000	4.250	0.000	4.250	0.000	0.000	DRY	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	4.338	0.000	4.338	0.000	0.000	DRY	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	3.153	0.000	3.153	0.000	0.000	DRY	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	3.744	0.000	3.744	0.000	0.000	DRY	0.000	0.000	0.000	0.000
13	0.000	0.000	0.000	4.245	0.000	4.245	0.000	0.000	DRY	0.000	0.000	0.000	0.000
14	0.000	0.000	0.000	3.146	0.000	3.146	0.000	0.000	DRY	0.000	0.000	0.000	0.000
15	0.000	0.000	0.000	4.335	0.000	4.335	0.000	0.000	DRY	0.000	0.000	0.000	0.000
16	0.000	0.000	0.000	3.164	0.000	3.164	0.000	0.000	DRY	0.000	0.000	0.000	0.000
17	0.000	0.000	0.000	3.420	0.000	3.420	0.000	0.000	DRY	0.000	0.000	0.000	0.000
18	0.000	0.000	0.000	2.767	0.000	2.767	0.000	0.000	DRY	0.000	0.000	0.000	0.000
19	0.000	0.000	0.000	3.877	0.000	3.877	0.000	0.000	DRY	0.000	0.000	0.000	0.000
20	0.000	0.000	0.000	3.923	0.000	3.923	0.000	0.000	DRY	0.000	0.000	0.000	0.000
21	0.000	0.000	0.000	3.329	0.000	3.329	0.000	0.000	DRY	0.000	0.000	0.000	0.000
22	0.000	0.000	0.000	2.379	0.000	2.379	0.000	0.000	DRY	0.000	0.000	0.000	0.000
23	0.000	0.000	0.000	2.033	0.000	2.033	0.000	0.000	DRY	0.000	0.000	0.000	0.000
24	0.000	0.000	0.000	3.470	0.000	3.470	0.000	0.000	DRY	0.000	0.000	0.000	0.000
25	0.000	0.000	0.000	4.204	0.000	4.204	0.000	0.000	DRY	0.000	0.000	0.000	0.000
26	0.000	0.000	0.000	2.466	0.000	2.466	0.000	0.000	DRY	0.000	0.000	0.000	0.000
27	0.000	0.000	0.000	1.820	0.000	1.820	0.000	0.000	DRY	0.000	0.000	0.000	0.000
28	0.000	0.000	0.000	3.243	0.000	3.243	0.000	0.000	DRY	0.000	0.000	0.000	0.000
29	0.000	0.000	0.000	3.997	0.000	3.997	0.000	0.000	DRY	0.000	0.000	0.000	0.000
30	0.000	0.000	0.000	2.664	0.000	2.664	0.000	0.000	DRY	0.000	0.000	0.000	0.000

10.2.5 Area of Inundation Files

These files report the area of inundation (wet area) calculated as the sum of areas of all wet elements. The file name has extension `.WETAREAS`. RiverFlow2D Plus model maximum values will only be generated using the post processing Plot RiverFlow2D results button on the Graphic Output Options panel.

A typical output is shown below:

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                  Build    2013.31.10
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                   (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                              ALL RIGHTS RESERVED
                              RUN DATE: 15/DEC/2013
================================================================================

    Time        Discharge       Wet Area
    (hrs)         (cfs)         (Acres)


    0.00            0.00           0.36
    0.10         7401.14          12.74
    0.20        13936.11          29.94
    0.30        20575.41          53.13
    0.40        27186.71          63.14
    0.50        33721.15          76.88
    0.60        40251.22          87.35
    0.70        46784.87          98.52
    0.80        53314.32         108.80
    0.90        59847.18         116.09
    1.00        66377.14         124.38
    1.10        66385.90         128.41
    1.20        66386.14         128.69
    1.30        66386.19         129.24
    1.40        66386.11         129.54
    1.50        66386.24         129.71
    1.60        66386.18         129.71
    1.70        66386.18         130.11
    1.80        66386.21         130.60
    1.90        66386.13         131.04
    2.00        66386.23         131.08
```

10.2.6 Maximum Value Tabular Files

These files report maximum nodal values of velocity module, depth, water surface elevations, and bed changes over the complete simulation. `.MAXI` is in metric units and `.MAXE` in English units. The reported variables are described in the following table:

Table 10.3 – Variables Reported on the Observation Point Files.

COLUMN	VARIABLE	DESCRIPTION	ENGLISH UNITS	METRIC UNITS
1	Node	Time in hours	-	-
2	X	Velocity component in x direction	ft	m
3	Y	Velocity component in y direction	ft	m
4	MAX VEL	Maximum velocity magnitude $\sqrt{U^2+V^2}$	ft/s	m/s
5	MAX DEPTH	Maximum water depth	ft	m
6	MAX WSEL	Maximum water elevation	ft	m
7	MAX BEDEL	Maximum bed elevation*	ft	m
8	MIN BEDEL	Maximum bed elevation*	ft	m
9	MIN EROSION DEPTH	Minimum erosion depth*	ft	m
10	MAX DEPOSITION DEPTH	Maximum deposition depth*	ft	m

**Output when using the sediment transport component.*

A typical output `.MAXE` file follows:

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                   Build    2013.31.10
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                          ALL RIGHTS RESERVED
                          RUN DATE: 15/DEC/2013
================================================================================

 MAXIMUM VELOCITIES AND DEPTHS IN ENGLISH UNITS

         FOR TIME (DDDD:HH:MM:SS): 0000:02:00:00

         NODE  MAX VELOCITY   MAX DEPTH     WSEL
                   (ft/s)        (ft)       (ft)
           1      3.086        4.284       164.686
           2      2.692        4.288       164.690
           3      4.978        4.234       164.636
           4      1.628        4.382       164.784
           5      0.000        2.322       168.450
           6      0.000        0.000      -999.000
           7      0.000        0.000      -999.000
           8      2.285        2.705       166.117
           9     13.216       15.618       177.182
          10     12.827       15.858       177.419
          11     13.316       14.995       176.896
          12     12.970       16.006       177.390
          13      3.224        4.363       164.764
          14      3.158        4.398       164.800
          15      0.000        0.000      -999.000
          16      0.000        0.000      -999.000
          17      4.297        4.096       164.631
          18      4.994        3.968       164.636
          19      1.525        1.819       166.588
          20      3.048        2.076       164.345
          21     13.404       14.807       176.820
          22     12.814       14.486       176.781
          23     13.135       15.867       177.282
          24     12.954       16.014       177.398
          25     13.251       15.594       177.129
          26      3.591        4.330       164.765
          27      3.593        4.388       164.790
          28     13.021       16.196       177.362
          29     13.041       16.356       177.414
          30     13.020       16.126       177.293
```

10.2.7 Observation Point Output Files

These files report time series of results at observation points. The program finds the closest node to the observation point coordinate and writes the result time series of the following variables:

Table 10.4 – Variables Reported on the Observation Point Files.

COLUMN	VARIABLE	DESCRIPTION	ENGLISH UNITS	METRIC UNITS
1	Time	Time in hours	-	-
2	U	Velocity component in x direction	ft	m
3	V	Velocity component in y direction	ft	m
4	VELOCITY	Maximum velocity magnitude $\sqrt{U^2 + V^2}$	ft/s	m/s
5	DEPTH	Maximum water depth	ft	m
6	WSEL	Maximum water elevation	ft	m
7	BEDEL_ORI	Maximum bed elevation*	ft	m
8	BEDEL	Maximum bed elevation*	ft	m
9	DELTA_BED	Minimum erosion depth*	ft	m
10	Froude	Maximum deposition depth*	ft	m
11	QSX	Volumetric sediment discharge per unit width in x direction	ft^2/s	m^2/s
12	QSY	Volumetric sediment discharge per unit width in y direction	ft^2/s	m^2/s
13	QS	Volumetric sediment discharge magnitude $Q_s = \sqrt{Q_{sx}^2 + Q_{sy}^2}$	ft^2/s	m^2/s

The file name for each node is:

`RESvsT_NODENUMBER.OUTI` for metric units and
`RESvsT_NODENUMBER.OUTE` for English units.

For example: `RESvsT_0000010.OUTI` is the file name for time series results of node 10. An example of this file is shown below.

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                Build    2013.31.07
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                            ALL RIGHTS RESERVED
                            RUN DATE: 29/OCT/2013
================================================================================

 RESULTS FOR CELL:     1118 OBSERVATION POINT ID: 1
  LOCATED AT COORDINATE: (  6238711.74),(  1964942.41)
```

TIME (hours)	U (ft/s)	V (ft/s)	VELOCITY (ft/s)	DEPTH (ft)	WSEL (ft)	BEDEL_ORI (ft)	BEDEL (ft)	DELTA_BED (ft)	FROUDE	QSX (ft2/s)	QSY (ft2/s)	QS (ft2/s)
0.00000	0.000	0.000	0.000	0.000	554.513	554.513	554.513	0.000	0.000	0.00E+00	0.00E+00	0.00E+00
0.50000	7.652	-16.205	17.921	7.404	561.917	554.513	554.513	0.000	1.162	0.00E+00	0.00E+00	0.00E+00
1.00000	7.638	-16.021	17.749	7.161	561.674	554.513	554.513	0.000	1.170	0.00E+00	0.00E+00	0.00E+00
1.50000	7.592	-15.784	17.515	6.857	561.369	554.513	554.513	0.000	1.180	0.00E+00	0.00E+00	0.00E+00
2.00000	7.394	-15.384	17.068	6.678	561.190	554.513	554.513	0.000	1.165	0.00E+00	0.00E+00	0.00E+00
2.50000	7.376	-15.237	16.929	6.494	561.007	554.513	554.513	0.000	1.172	0.00E+00	0.00E+00	0.00E+00
3.00000	7.174	-14.790	16.438	6.599	561.112	554.513	554.513	0.000	1.129	0.00E+00	0.00E+00	0.00E+00
3.50000	7.172	-14.785	16.433	6.595	561.108	554.513	554.513	0.000	1.129	0.00E+00	0.00E+00	0.00E+00
4.00000	7.203	-14.878	16.529	6.651	561.164	554.513	554.513	0.000	1.130	0.00E+00	0.00E+00	0.00E+00
4.50000	7.266	-15.066	16.726	6.777	561.290	554.513	554.513	0.000	1.133	0.00E+00	0.00E+00	0.00E+00
5.00000	7.499	-15.472	17.194	6.888	561.401	554.513	554.513	0.000	1.155	0.00E+00	0.00E+00	0.00E+00
5.50000	7.703	-15.876	17.647	7.110	561.623	554.513	554.513	0.000	1.167	0.00E+00	0.00E+00	0.00E+00
6.00000	7.654	-16.040	17.772	7.556	562.069	554.513	554.513	0.000	1.140	0.00E+00	0.00E+00	0.00E+00
6.50000	7.839	-16.614	18.370	7.881	562.394	554.513	554.513	0.000	1.154	0.00E+00	0.00E+00	0.00E+00
7.00000	7.784	-16.788	18.505	8.583	563.096	554.513	554.513	0.000	1.114	0.00E+00	0.00E+00	0.00E+00
7.50000	7.736	-16.986	18.665	9.090	563.603	554.513	554.513	0.000	1.092	0.00E+00	0.00E+00	0.00E+00
8.00000	7.785	-17.432	19.091	9.531	564.044	554.513	554.513	0.000	1.091	0.00E+00	0.00E+00	0.00E+00
8.50000	7.653	-17.293	18.911	10.330	564.842	554.513	554.513	0.000	1.038	0.00E+00	0.00E+00	0.00E+00
9.00000	7.548	-17.342	18.913	11.028	565.541	554.513	554.513	0.000	1.004	0.00E+00	0.00E+00	0.00E+00
9.50000	7.507	-17.283	18.843	11.818	566.331	554.513	554.513	0.000	0.967	0.00E+00	0.00E+00	0.00E+00
10.00000	7.736	-17.857	19.461	12.142	566.655	554.513	554.513	0.000	0.985	0.00E+00	0.00E+00	0.00E+00
10.50000	7.872	-18.155	19.788	12.754	567.266	554.513	554.513	0.000	0.977	0.00E+00	0.00E+00	0.00E+00
11.00000	7.869	-18.237	19.862	13.724	568.236	554.513	554.513	0.000	0.946	0.00E+00	0.00E+00	0.00E+00
11.50000	8.038	-18.733	20.385	14.445	568.958	554.513	554.513	0.000	0.946	0.00E+00	0.00E+00	0.00E+00
12.00000	8.083	-18.898	20.554	15.214	569.726	554.513	554.513	0.000	0.929	0.00E+00	0.00E+00	0.00E+00
12.50000	8.199	-19.286	20.956	15.884	570.397	554.513	554.513	0.000	0.927	0.00E+00	0.00E+00	0.00E+00
13.00000	8.307	-19.552	21.243	16.294	570.807	554.513	554.513	0.000	0.928	0.00E+00	0.00E+00	0.00E+00
13.50000	8.453	-19.908	21.629	16.718	571.231	554.513	554.513	0.000	0.933	0.00E+00	0.00E+00	0.00E+00
14.00000	8.662	-20.378	22.142	17.210	571.722	554.513	554.513	0.000	0.941	0.00E+00	0.00E+00	0.00E+00
14.50000	8.801	-20.715	22.507	17.872	572.384	554.513	554.513	0.000	0.939	0.00E+00	0.00E+00	0.00E+00
15.00000	8.977	-21.152	22.978	18.292	572.805	554.513	554.513	0.000	0.948	0.00E+00	0.00E+00	0.00E+00
15.50000	9.013	-21.414	23.234	18.892	573.405	554.513	554.513	0.000	0.943	0.00E+00	0.00E+00	0.00E+00
16.00000	9.321	-21.982	23.877	19.307	573.820	554.513	554.513	0.000	0.958	0.00E+00	0.00E+00	0.00E+00
16.50000	9.645	-22.399	24.387	19.744	574.257	554.513	554.513	0.000	0.968	0.00E+00	0.00E+00	0.00E+00
17.00000	9.770	-22.614	24.634	20.302	574.815	554.513	554.513	0.000	0.964	0.00E+00	0.00E+00	0.00E+00
17.50000	9.884	-22.837	24.884	20.815	575.328	554.513	554.513	0.000	0.962	0.00E+00	0.00E+00	0.00E+00
18.00000	10.108	-23.088	25.204	21.182	575.695	554.513	554.513	0.000	0.966	0.00E+00	0.00E+00	0.00E+00
18.50000	10.037	-23.142	25.224	21.553	576.066	554.513	554.513	0.000	0.958	0.00E+00	0.00E+00	0.00E+00
19.00000	10.017	-23.184	25.256	21.798	576.311	554.513	554.513	0.000	0.954	0.00E+00	0.00E+00	0.00E+00
19.50000	10.024	-23.235	25.305	21.892	576.405	554.513	554.513	0.000	0.954	0.00E+00	0.00E+00	0.00E+00
20.00000	10.014	-23.218	25.285	21.879	576.392	554.513	554.513	0.000	0.953	0.00E+00	0.00E+00	0.00E+00
20.50000	10.011	-23.142	25.214	21.734	576.247	554.513	554.513	0.000	0.954	0.00E+00	0.00E+00	0.00E+00
21.00000	10.005	-23.067	25.143	21.545	576.058	554.513	554.513	0.000	0.955	0.00E+00	0.00E+00	0.00E+00
21.50000	10.065	-23.085	25.184	21.204	575.717	554.513	554.513	0.000	0.965	0.00E+00	0.00E+00	0.00E+00
22.00000	10.073	-22.982	25.092	20.817	575.330	554.513	554.513	0.000	0.970	0.00E+00	0.00E+00	0.00E+00
22.50000	9.834	-22.630	24.674	20.583	575.096	554.513	554.513	0.000	0.959	0.00E+00	0.00E+00	0.00E+00
23.00000	9.742	-22.462	24.484	20.188	574.700	554.513	554.513	0.000	0.961	0.00E+00	0.00E+00	0.00E+00
23.50000	9.447	-22.149	24.079	19.748	574.261	554.513	554.513	0.000	0.956	0.00E+00	0.00E+00	0.00E+00

10.2.8 Hot Start File

The hot start `2binitialized.HOTSTART` file is used to restart a simulation from previously computed results and when hot start option is selected. By default the file contains the name of the last report time in seconds and the corresponding state file. Those results will be used as initial condition to restart the simulation when the hot start option is activated. For example, if the user stops the simulation at 5 hours to review results or for any other reason the, `2binitialized.HOTSTART` file would have the following text:

```
18000 state5.out
```

Note that the stateN.out files are named sequentially. For instance, stateN.out corresponds to the N^{th} report interval.

RiverFlow2D can be restarted from the any existing report time by reading the initial conditions from the stateN.out file indicated in the `2binitialized.HOTSTART` file. To restart from a time different from the last one calculated, just edit the `2binitialized.HOTSTART` file and enter the desired time in seconds and corresponding stateN.out file name that is to be used as initial conditions. For example, to hot start from hour 3 (10800 seconds) and assuming that the report interval is 0.5 hours, the `2binitialized.HOTSTART` file should contain the following entry:

```
10800 state6.out
```

The hot start option is often useful to establishing initial conditions common to a series of simulations for various return periods. For instance, to generate your initial state, you could run the model with a constant discharge inflow until the model converges to a steady state. Assuming that the final report time corresponds to the `state20.out` file, you can edit the `2binitialized.HOTSTART` file as shown:

```
0 state20.out
```

Then when you run the RiverFlow2D model using the hot start option, the model will start assuming that the data in the state20.out file will define the initial conditions. You may want to keep the `2binitialized.HOTSTART` and `state20.out` files in a separate directory and copy them to the project folder for each desired scenario.

Please, keep in mind that the stateN.out files are tied to the mesh you use, so if you modify the mesh in any way, you will need to use the stateN.out corresponding to that mesh.

10.3 Component Output Files

10.3.1 Culvert Output Files

For each culvert, RiverFlow2D creates an output file named: `CULVERT_culvertID.OUT`, where culvertID is the text provided by the user to identify the culvert. Report includes discharge for each report interval as shown:

```
================================================================================
================================================================================
                RiverFLO-2D PLUS Release  4.0
                             Build    2013.31.08
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                  (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                             ALL RIGHTS RESERVED
                             RUN DATE: 20/SEP/2013
================================================================================
================================================================================
  Results for Culvert no.:     1 Culvert ID: Prepust_AC
================================================================================
      Time         Qc        WSEL1      WSEL2
      hrs.         m3/s        m.         m.
     0.04980      0.000     67.715     66.840
     0.09987      0.000     67.715     66.840
     0.14995      0.000     67.715     66.840
     0.19975      0.000     67.715     66.840
     0.24982      0.000     67.715     66.840
     0.29989      0.000     67.715     66.840
     0.34997      0.000     67.715     66.840
     0.39977      0.000     67.715     66.840
     0.44984      0.000     67.715     66.840
     0.49991      0.000     67.715     66.840
     0.54998      0.000     67.715     66.840
     0.59978      0.000     67.715     66.840
     0.64986      0.000     67.715     66.840
     0.69993      0.000     67.715     66.840
     0.74973      0.000     67.715     66.840
     0.79980      0.000     67.715     66.840
     0.84988      0.000     67.715     66.840
     0.89995      0.000     67.715     66.840
     0.94984      0.000     67.715     66.840
     0.99982      0.000     67.715     66.840
     1.04995      0.000     67.715     66.840
     1.09981      0.000     67.715     66.840
     1.15001      0.000     67.715     66.840
     1.19998      0.000     67.715     66.840
     1.24998      0.000     67.715     66.840
     1.29999      0.000     67.715     66.840
     1.35000      0.000     67.056     66.228
     1.39999      1.828     67.450     67.436
     1.45000      2.899     67.523     67.487
     1.50000      3.280     67.801     67.754
     1.54999      3.591     67.831     67.776
     1.59998      3.799     67.927     67.865
     1.64999      4.144     67.953     67.879
     1.69999      4.257     67.961     67.883
     1.74998      4.096     67.959     67.886
     1.79998      4.184     67.961     67.885
     1.84999      4.035     67.959     67.889
```

10.3.2 Internal Rating Table Output Files

For each Internal Rating Table, RiverFlow2D creates an output file named: `IRT_irtID.OUT`, where irtID is the text provided by the user to identify the Internal Rating Table. Report includes discharge for each report interval as shown:

```
================================================================================
================================================================================
        RiverFLO-2D Plus - Release 4.1
                           Build Jan 30 2014
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                        (C) COPYRIGHT 2009-2014 Hydronia, LLC.
                                      ALL RIGHTS RESERVED
                                      RUN DATE: 04/Feb/2014
================================================================================

================================================================================
Results for Internal Rating Table no.:          1 ID: IIRT1
================================================================================
 Time              WSE          Qc
 hrs.               m.          m3/s
0.10000           0.712        4.110
0.20000           1.080        7.888
0.30000           1.290       10.376
0.40000           1.291       10.381
0.50000           1.290       10.375
```

10.3.3 Weir Output Files

These files report results for each weir and for each output interval. File extension is `.WEIRI` for metric units and `.WEIRE` for English units. Output includes the following information:

Table 10.5 – Variables Reported on the Weir Point Files.

VARIABLE	DESCRIPTION
EDGE	Edge number
N1	Cell at side 1 of the edge
N2	Cell at side 2 of the edge
WSE1	Water surface elevation at cell N1
WSE2	Water surface elevation at cell N2
D1	Depth at cell N1
D2	Depth at cell N2
Distance	Edge length
Q	Edge discharge

A typical weir output file format is shown below:

```
================================================================================
================================================================================
        RiverFLO-2D Plus - Release 4.1
                           Build Mar  3 2014
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                        (C) COPYRIGHT 2009-2014 Hydronia, LLC.
                                      ALL RIGHTS RESERVED
                                      RUN DATE: 03/Mar/2014
================================================================================

 WEIR RESULTS IN SI UNITS

| TIME: 0000 days,07 hours,30 min.,00 secs.

 WEIR NO.:   1 WEIR ID: Weir6

 EDGE      N1        N2       WSE1       WSE2       D1        D2     Distance     Q
                              (m)        (m)        (m)       (m)       (m)      (m3/s)
   1    208257     24110     527.13     527.14      0.63      3.14      6.00      0.20
   2    208259     24105     527.13     527.14      0.63      3.14      6.00      0.24
   3    208261     24101     527.13     527.14      0.63      3.14      6.00      0.25
   4    208263     24085     527.13     527.14      0.63      3.14      6.00      0.25
   5    208265      1800     527.13     527.14      0.63      3.14      6.00      0.25
   6    208267       909     527.13     527.14      0.63      3.14      6.00      0.25
   7    208269       111     527.14     527.14      0.64      3.14      6.00      0.15
Total discharge over weir Q =     1.597 m3/s
```

10.4 Profile and Cross Section Output Files

10.4.1 General Cross Section Output Files

When using the *Output results* for *cross sections* option, the model will generate files with extensions `.XSECI` and `.XSECE`, that report results along user provided cross sections. For each output interval and for each user defined cross sections the bed elevation, depth, water surface elevation, depth average velocity, Froude number and volumetric sediment discharge per unit width is written to file .XSECI is in metric and `.XSECE` in English units. A typical `.XSECI` file is as follows:

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                   Build    2013.31.10
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                 (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                           ALL RIGHTS RESERVED
                           RUN DATE: 15/DEC/2013
================================================================================

 CROSS SECTION RESULTS IN SI UNITS

 TIME: 0000 days, 00 hours, 06 min.,00 secs.


 CROSS SECTION NO.:   1 CROSS SECTION ID: CROSS1

 (     243389.02,      94311.52), (     243540.97,      94454.32)
  ELEM    STATION    BEDEL      DEPTH      WSEL      VELOCITY  FROUDE      QS
             (m)       (m)        (m)       (m)        (m/s)              (m2/s)
  1278      25.02     49.54       1.15      50.68       1.39      0.41  0.000000
  1279      33.36     49.54       1.15      50.68       1.50      0.45  0.000000
  1280      35.45     49.43       1.26      50.70       1.44      0.41  0.000000
  1281      41.70     49.42       1.27      50.69       1.55      0.44  0.000000
  1282      47.96     49.42       1.28      50.70       1.56      0.44  0.000000
  1283      50.04     49.27       1.44      50.70       1.58      0.42  0.000000
  1336      54.21     49.34       1.34      50.68       1.65      0.46  0.000000
  1515      58.38     49.16       1.53      50.69       1.74      0.45  0.000000
  1513      66.73     49.12       1.58      50.70       1.70      0.43  0.000000
  1512      68.81     48.92       1.78      50.70       1.75      0.42  0.000000
  1475      77.15     48.79       1.91      50.70       1.80      0.42  0.000000
  1476      79.24     48.65       2.04      50.70       1.73      0.39  0.000000
   381      87.58     48.77       1.91      50.68       1.77      0.41  0.000000
   379      95.92     49.64       1.02      50.65       1.92      0.61  0.000000
   351     104.26     50.24       0.44      50.68       1.83      0.88  0.000000
   352     108.43     50.03       0.49      50.52       1.96      0.90  0.000000
   318     114.68     50.70       0.04      50.74       0.36      0.58  0.000000
   276     123.03     51.08       0.00      51.08       0.00      0.00  0.000000
   235     131.37     51.82       0.00      51.82       0.00      0.00  0.000000
   192     139.71     52.13       0.00      52.13       0.00      0.00  0.000000
   193     148.05     52.13       0.00      52.13       0.00      0.00  0.000000
   129     150.13     52.12       0.00      52.12       0.00      0.00  0.000000
   130     156.39     52.14       0.00      52.14       0.00      0.00  0.000000
    55     158.47     52.27       0.00      52.27       0.00      0.00  0.000000
    56     164.73     52.31       0.00      52.31       0.00      0.00  0.000000
    53     166.81     52.36       0.00      52.36       0.00      0.00  0.000000
    54     173.07     52.36       0.00      52.36       0.00      0.00  0.000000
 Q =    206.975 m3/s.
```

When running only hydrodynamics the `.XSECE` and `.XSECI` files will display the cross section water discharge. When running sediment transport, in addition to the water discharge these files will report the total sediment discharge in ft^3/s or m^3/s.

10.4.2 Cross Section Hydrograph Output Files

These files will only be generated using the post processing Plot RiverFlow2D results button on the *Graphic Output Options* panel. When using the *Output results for cross sections* option, the model will generate files with extension `.XSECH` and `.XSECSED` (if using sediment transport component), that report a hydrograph table for each cross section. A typical path.XSECH file is as follows:

```
================================================================================
================================================================================
                RiverFLO-2D Plus - Release  Plus 4.0
                                   Build    2013.31.10
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                  (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                             ALL RIGHTS RESERVED
                             RUN DATE: 15/DEC/2013
================================================================================

 Hydrograph for cross sections in cfs, time in hours.

     Time         CROSS1         CROSS2         CROSS3

    0.00000          0.000          0.000          0.000
    0.10000       7309.242          0.000          0.000
    0.20000      14110.469       9297.244          0.000
    0.30000      20795.390      15836.941       4226.598
    0.40000      27414.470      22501.921      15747.631
    0.50000      34057.759      29757.209      23021.471
    0.60000      40710.035      36935.477      31311.475
    0.70000      47357.998      44127.966      38937.936
    0.80000      54011.200      50918.418      46109.916
    0.90000      60652.336      57802.940      52271.772
    1.00000      67281.089      64461.808      59843.796
    1.10000      67393.541      66660.524      65363.158
    1.20000      67403.150      66955.816      66720.124
    1.30000      67404.687      67005.883      67011.927
    1.40000      67404.932      67014.168      67066.649
    1.50000      67404.981      67015.506      67076.358
    1.60000      67404.985      67015.795      67077.695
    1.70000      67404.983      67016.257      67077.356
    1.80000      67404.981      67016.418      67076.416
    1.90000      67404.981      67016.747      67075.477
    2.00000      67404.981      67017.041      67074.916
```

10.4.3 Profile Output Files

When using the *Output results for profiles* option, the model will generate files with extensions `.PRFI` and `.PRFE`, that report results along user provided polylines. For each output interval and for the number of points along user defined polylines these files list bed elevation, depth, water surface elevation, depth average velocity, and Froude number. `.PRFI` is in metric and .PRFE in English units. An example output is shown below:

```
================================================================================
================================================================================
        RiverFLO-2D Plus - Release 4.1
                         Build Nov  4 2013
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                    (C) COPYRIGHT 2009-2013 Hydronia, LLC.
                            ALL RIGHTS RESERVED
                            RUN DATE: 17/Dec/2013
================================================================================

 PROFILE RESULTS IN ENGLISH UNITS

 TIME: 0000 days,00 hours,00 min.,00 secs.

 PROFILE NO.:   1 PROFILE ID: PROFILE1

 ELEM   DISTANCE    BEDEL     DEPTH      WSEL    VELOCITY  FROUDE
          (ft)      (ft)       (ft)      (ft)     (ft/s)
    6      0.00    161.38      1.43    162.81      0.00      0.00
   19     50.90    160.63      2.17    162.81      0.00      0.00
   21    101.79    161.30      0.00    161.30      0.00      0.00
   32    152.69    161.45      0.00    161.45      0.00      0.00
   32    203.58    161.45      0.00    161.45      0.00      0.00
   73    254.48    161.79      0.00    161.79      0.00      0.00
   74    305.37    160.85      0.00    160.85      0.00      0.00
   81    356.27    161.03      0.00    161.03      0.00      0.00
   94    407.16    160.90      0.00    160.90      0.00      0.00
   93    458.06    161.29      0.00    161.29      0.00      0.00
   98    508.95    161.25      0.00    161.25      0.00      0.00
  176    559.85    160.68      0.00    160.68      0.00      0.00
  176    610.74    160.68      0.00    160.68      0.00      0.00
  179    661.64    160.41      0.00    160.41      0.00      0.00
  297    712.53    160.39      0.00    160.39      0.00      0.00
  298    763.43    159.86      0.00    159.86      0.00      0.00
  439    814.32    160.61      0.00    160.61      0.00      0.00
  440    863.58    160.43      0.00    160.43      0.00      0.00
  442    912.84    160.86      0.00    160.86      0.00      0.00
  443    962.10    160.40      0.00    160.40      0.00      0.00
  444   1011.36    160.73      0.00    160.73      0.00      0.00
  445   1060.61    159.37      0.00    159.37      0.00      0.00
  446   1109.87    160.68      0.00    160.68      0.00      0.00
  448   1159.13    160.71      0.00    160.71      0.00      0.00
  448   1208.39    160.71      0.00    160.71      0.00      0.00
  450   1257.64    159.37      0.00    159.37      0.00      0.00
  451   1306.90    158.47      0.00    158.47      0.00      0.00
 1177   1356.16    160.29      0.00    160.29      0.00      0.00
 1233   1407.84    159.28      0.00    159.28      0.00      0.00
 1233   1459.53    159.28      0.00    159.28      0.00      0.00
 1259   1511.22    162.75      0.00    162.75      0.00      0.00
```

10.5 Output Files for GIS Post-processing

10.5.1 General Spatial Distribution Files for SMS (.HDF5)

These binary files allow seamless transfer to Aquaveo SMS software. These files can only be read with SMS or other appropriate software. They use the `.HDF5` extension and in RiverFlow2D are named as follows:

Results at nodes for all report times: `<ProjectName>_RESULTS.HDF5`
Maximum values at nodes: `<ProjectName>_MAXIMUM_RESULTS.HDF5`

Where `<ProjectName>` is the mesh name or project name. For example

`Mesh1_RESULTS.HDF5`

`Mesh1_MAXIMUM_RESULTS.HDF5`

Where `Mesh1` is the project name. The content of the *Result at Nodes* files is described on table 10.6.

Table 10.6 – Variables Reported on _RESULTS.HDF5 Output Files.

	VALUE	ENGLISH UNITS	METRIC UNITS
1	Node number	-	-
2	Node x-coordinate	ft	m
3	Node y-coordinate	ft	m
4	Velocity component in x direction U	ft/s	m/s
5	Velocity component in y direction V	ft/s	m/s
6	Velocity magnitude $\left\|\vec{U}\right\| = \sqrt{U^2 + V^2}$	ft/s	m/s
7	Water surface elevation	ft	m
8	Depth H	ft	m
9	Initial bed elevation	ft	m
10	Bed elevation	ft	m
11	Bed elevation change since time = 0	ft	m
12	Dry or wet	Dry=1 / Wet=0	Dry=1 / Wet=0
13	Froude number	-	-
14	Volumetric sediment discharge per unit width in x direction: Q_{sx}	ft^2/s	m^2/s
15	Volumetric sediment discharge per unit width in y direction: Q_{sy}	ft^2/s	m^2/s
16	Volumetric sediment discharge magnitude: $Q_s\sqrt{Q_{sx}^2 + Q_{sy}^2}$	ft^2/s	m^2/s
17	Bed shear stress*: $\tau = \gamma H S_f = \gamma \left(Un/k\right)^2 / H^{1/3}$	lb/ft^2	Pa
18	Accumulated rainfall volume	ft^3	m^3
19	Accumulated infiltration volume	ft^3	m^3
20	Concentration for each solute**	Same as in BC's	Same as in BC's
21	Concentration for each suspended sediment fraction***	Cv	Cv

**English units:* $= 62.4 lb/ft^3$, $k = 1.49$.
**Metric units:* $= 9810 N/m^3$; $k = 1$, n = *Manning's coefficient.*
***Only written when running the PL or WQ modules.*
****Only written when running the ST module with Suspended Sediment.*

The maximum values at nodes files, `<ProjectName>_MAXIMUM_RESULTS.HDF5`, report maximum values of velocity module, depth and water surface elevations, and can also be open and displayed in SMS. The content of these files is described on tables 10.7 and 10.8.

Table 10.7 – Dynamic Variables Reported on the <ProjectName>_MAXIMUM_RESULTS.HDF5 files.

	VALUE	ENGLISH UNITS	METRIC UNITS
1	Node x-coordinate	ft	m
2	Node y-coordinate	ft	m
3	Maximum velocity magnitude $\sqrt{U_x^2 + U_y^2}$	ft/s	m/s
4	Maximum depth	ft	m
5	Maximum water surface elevation	ft	m
6	Maximum bed elevation*	ft	m
7	Minimum bed elevation*	ft	m
8	Maximum erosion depth*	ft	m
9	Maximum deposition depth*	ft	m
10	Inundation time	$hrs.$	$hrs.$
11	Time to 0.30 m (Metric) or 1 ft. (English)*	$hrs.$	$hrs.$
12	Time to 0.60 m (Metric) or 2 ft. (English)*	$hrs.$	$hrs.$
13	Time to 1 m (Metric) or 3 ft. (English)*	$hrs.$	$hrs.$
14	Time to maximum depth*	$hrs.$	$hrs.$

**Output when using the sediment transport component.*

Table 10.8 – Dynamic Variables Reported on the <ProjectName>_MAXIMUM_RESULTS.HDF5 files.

COLUMN	VALUE	HAZARD ZONES
1	USBR Homes	$0, 1, 2$, and 3
2	USBR Passenger Vehicles	$0, 1, 2$, and 3
3	USBR Mobile Homes	$0, 1, 2$, and 3
4	USBR Adults	$0, 1, 2$, and 3
5	USBR Children	$0, 1, 2$, and 3
6	Swiss Method for Water Flooding	$0, 1, 2$, and 3
7	Swiss Method for Debris Flow	$0, 1, 2$, and 3
8	Austrian Method for River Flooding	$0, 1$, and 2
9	Austrian Method for Torrents Tr=100 yrs.	$0, 1$, and 2
10	Austrian Method for Torrents Tr=10 yrs.	$0, 1$, and 2
11	UK Method	$0, 1, 2$, and 3

.

10.5.2 General Spatial Distribution Files (.EXP)

These ASCII files allow seamless transfer to Geographic Information System software. These files use the `.EXP` extension and are named as follows:

For English units: `TIME_ENG_DDDD_HH_MM_SS.EXP`
For Metric units: `TIME_METRIC_DDDD_HH_MM_SS.EXP`

Where DDDD is days, HH is hours, MM is minutes and SS seconds. For example

`TIME_ENG_0001_12_01_34.EXP`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of node parameters (19 by default). Then follows NNODES lines with results for each node in the triangular element mesh as shown:

Table 10.9 – Variables Reported on TIME_.EXP Output Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Node number	-	-
2	Node x-coordinate	ft	m
3	Node y-coordinate	ft	m
4	Velocity component in x direction U	ft/s	m/s
5	Velocity component in y direction V	ft/s	m/s
6	Velocity magnitude $\left\vert\vec{U}\right\vert = \sqrt{U^2+V^2}$	ft/s	m/s
7	Water surface elevation	ft	m
8	Depth H	ft	m
9	Initial bed elevation	ft	m
10	Bed elevation	ft	m
11	Bed elevation change since time = 0	ft	m
12	Dry or wet	Dry=1 / Wet=0	Dry=1 / Wet=0
13	Froude number	-	-
14	Volumetric sediment discharge per unit width in x direction: Q_{sx}	ft^2/s	m^2/s
15	Volumetric sediment discharge per unit width in y direction: Q_{sy}	ft^2/s	m^2/s
16	Volumetric sediment discharge magnitude: $Q_s\sqrt{Q_{sx}^2+Q_{sy}^2}$	ft^2/s	m^2/s
17	Bed shear stress*: $\tau = \gamma HS_f = \gamma\left(Un/k\right)^2/H^{1/3}$	lb/ft^2	Pa
18	Accumulated rainfall volume	ft^3	m^3
19	Accumulated infiltration volume	ft^3	m^3
20	Manning's n		-

**English units:* $= 62.4 lb/ft^3$, $k = 1.49$.
**Metric units:* $= 9810 N/m^3$; $k = 1$, n = *Manning's coefficient.*

The following file is an example of a typical ASCII `.EXP` file:

19																			
1	522934.5	5258759	-0.48109	-0.14149	0.503769	0.999967	1.211967	-0.212	-0.212	0	0	0.146125	0	0	0	2.101571	1.066884	8.635435	0.03
2	522976	5258730	-0.24235	0.145426	0.556546	0.992053	0.103853	0.8882	0.8882	0	0	0.55148	0	0	0	5.817989	1.087993	6.233529	0.03
3	522998.2	5258685	0	0	0	-999	0	1.764567	1.764567	0	1	0	0	0	0	0	1.115509	1.115509	0.03
4	523020.3	5258640	0	0	0	-999	0	2.512567	2.512567	0	1	0	0	0	0	0	1.084248	1.084248	0.03
5	522890.2	5258782	-0.38871	-0.16848	0.423669	1.000264	2.16513	-1.16487	-1.16487	0	0	0.091944	0	0	0	1.225	1.211373	9.804941	0.03
6	522841.9	5258797	-0.332	-0.14048	0.360579	1	2.72415	-1.72415	-1.72415	0	0	0.069763	0	0	0	0.821931	1.187298	9.610075	0.03
7	522793.4	5258785	-0.30863	-0.10377	0.32607	1.000215	2.952448	-1.95223	-1.95223	0	0	0.060598	0	0	0	0.654345	1.079267	8.735666	0.03
8	523449.6	5258171	0	0	0	-999	0	21.08005	21.08005	0	1	0	0	0	0	0	0.923123	0.923123	0.03
9	522744.9	5258770	-0.28633	-0.06636	0.294408	1.000211	3.066077	-2.06587	-2.06587	0	0	0.053691	0	0	0	0.526766	1.031323	8.347601	0.03
10	523042.5	5258596	0	0	0	-999	0	3.5227	3.5227	0	1	0	0	0	0	0	1.080833	1.080833	0.03
11	523064.6	5258551	0	0	0	-999	0	4.4337	4.4337	0	1	0	0	0	0	0	1.074839	1.074839	0.03
12	523086.8	5258506	0	0	0	-999	0	5.3432	5.3432	0	1	0	0	0	0	0	1.045626	1.045626	0.03
13	523109	5258461	0	0	0	-999	0	6.714333	6.714333	0	1	0	0	0	0	0	1.047731	1.047731	0.03
14	523139.5	5258423	0	0	0	-999	0	8.776267	8.776267	0	1	0	0	0	0	0	1.07115	1.07115	0.03
15	522888.8	5258731	-0.30594	-0.14531	0.340768	0.999756	1.697789	-0.69803	-0.69803	0	0	0.083514	0	0	0	0.859412	1.126776	9.113662	0.03
16	522930.2	5258712	-0.20119	0.03197	0.377322	0.995429	0.791409	0.20402	0.20402	0	0	0.135441	0	0	0	1.358944	1.066487	8.628285	0.03
17	522953.1	5258674	0	0	0	-999	0	1.001733	1.001733	0	1	0	0	0	0	0	1.124586	5.284986	0.03
18	522977	5258630	0	0	0	-999	0	1.86185	1.86185	0	1	0	0	0	0	0	1.096724	1.457021	0.03
19	522999.3	5258585	0	0	0	-999	0	2.79365	2.79365	0	1	0	0	0	0	0	1.081976	1.081976	0.03
20	522838.5	5258748	-0.31319	-0.13932	0.343449	1.000582	2.482968	-1.48239	-1.48239	0	0	0.069601	0	0	0	0.769095	1.172116	9.481359	0.03
21	522787.5	5258742	-0.28445	-0.09156	0.299406	1.00073	2.82743	-1.8267	-1.8267	0	0	0.05686	0	0	0	0.559716	1.062548	8.596422	0.03

10.5.3 Pollutant Concentration Files

These ASCII files contains the pollutant. These files use the `.EXP` extension and are named as follows:

`CONC_DDDD_HH_MM_SS.EXP`

Where DDDD is days, HH is hours, MM is minutes and SS seconds. For example

`CONC_0001_12_01_34.EXP`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of solutes used in the PL run (NPOLLUTANTS_MAX). Then follows NNODES lines with results for each node in the triangular element mesh as shown:

Table 10.10 – Variables Reported on CONC*.EXP Output Files.

COLUMN	VALUE	UNITS
1	Node number	-
2	Concentration for solute 1	Same as in BC's
3	Concentration for solute 2	Same as in BC's
...	...	...
5	Concentration for solute NPOLLUTANTS_MAX	Same as in BC's

The following file is an example of a typical `CONC*.EXP` file:

```
3
1   0.000000   0.000000   0.000000
2   0.000000   0.000000   0.000000
3   0.000000   0.000000   0.000000
4   0.202378   0.000000   0.000000
5   0.326602   0.000000   0.000000
6   0.291721   0.000000   0.000000
7   0.000000   0.000000   0.000000
...
```

In this example, the `CONC*.EXP` has 3 pollutants.

10.5.4 Suspended Sediment Concentration Files

These ASCII files contain suspended sediment concentrations. These files use the .EXP extension and are named as follows:

`ST_DDDD_HH_MM_SS.EXP`

Where DDDD is days, HH is hours, MM is minutes and SS seconds. For example

`ST_0001_12_01_34.EXP`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of suspended sediment fractions used in the ST run (NSSNFRAC). Then follows NNODES lines with results for each node in the triangular element mesh as shown:

Table 10.11 – Variables Reported on ST*.EXP Output Files.

COLUMN	VALUE	UNITS
1	Node number	-
2	Concentration by volume for fraction 1	Fraction of 1
3	Concentration by volume for fraction 2	Fraction of 1
...	...	...
5	Concentration by volume for fraction NSSNFRAC	Fraction of 1

The following file is an example of a typical ST*.EXP file:

```
2
1       0.000000       0.000000
2       0.000000       0.000000
3       0.000000       0.000000
4       0.000000       0.000000
5       0.000000       0.000000
6       0.000000       0.000000
7       0.000000       0.000000
```

```
8       0.000000      0.000000
9       0.000287      0.000287
10       0.000456      0.000456
11       0.000356      0.000356
12       0.000334      0.000334
13       0.000000      0.000000
...
```

In this example, the `ST*.EXP` has 2 suspended sediment fractions.

10.5.5 Maximum Value GIS Files

These ASCII files report maximum values of velocity module, depth and water surface elevations and allow seamless transfer to Geographic Information System software. These files use the `.EXP` extension and are named as follows:

For English units: `<ProjectName>_ENG_MAX.EXP`
For Metric units: `<ProjectName>_METRIC_MAX.EXP`

The format for these files is as follows. The first line indicates the number of node parameters (5 by default). There follows NNODES lines with velocity module, depth and water surface elevation for each node as shown:

Table 10.12 – Variables Reported on the <ProjectName>_ENG_MAX.EXP or <ProjectName>_METRIC_MAX.EXP Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Node number	-	-
2	Node x-coordinate	ft	m
3	Node y-coordinate	ft	m
4	Maximum velocity magnitude $\sqrt{U_x^2 + U_y^2}$	ft/s	m/s
5	Maximum depth	ft	m
6	Maximum water surface elevation	ft	m
7	Maximum bed elevation*	ft	m
8	Minimum bed elevation*	ft	m
9	Maximum erosion depth*	ft	m
10	Maximum deposition depth*	ft	m

**Output when using the sediment transport component.*

10.5.6 Time-to-Depth GIS File

This ASCII file are generated when using the Plot RiverFlow2D Results button on the *Graphic Output Option* panel of the DIP. The file reports the time at which certain depths are reached during the simulation and allow seamless transfer to Geographic Information System software. In metric units the file includes time to 0.30 m of depth, time to 1 m and time to maximum depth for each node. In English units the file includes time to 1ft of depth, time to 3 ft and time to maximum depth for each node. Time is always given in hours. The time-to-depth files uses the `.EXP` extension and have this name:

`<ProjectName>_time2depths.exp`

The format for these files is as follows. The first line indicates the number of node parameters (3 by default). There follows NNODES lines with time to 0.30 m (1 ft), time to 1 m (3 ft), and time to maximum depth for each node as shown in Table 10.13. When the node remains dry or depth is below 0.30 m (1 ft) the reported value is -1.

Table 10.13 – Variables Reported on the <ProjectName>_time2depths.exp File.

COLUMN	VALUE	UNITS
1	Node number	-
2	Time to 0.30 m (Metric) or 1 ft. (English)*	*hrs.*
3	Time to 0.60 m (Metric) or 2 ft. (English)*	*hrs.*
4	Time to 1 m (Metric) or 3 ft. (English)*	*hrs.*
5	Time to maximum depth*	*hrs.*

**Time = -1 when node is dry or depth does not reach 0.3 m (1ft).*

10.5.7 Support Files to Generate ESRI Shapefiles

These ASCII files can be imported into Geographic Information System (GIS) programs.

Velocity field files use the .EXP extension and are named as follows:

`VelField_SF_TIME_DDDD_HH_MM_SS.EXP`

Water surface elevation files use the .EXP extension and are named as follows:

`WSE_SF_TIME_TIME_DDDD_HH_MM_SS.EXP`

Where DDDD is days, HH is hours, MM is minutes and SS seconds.

10.5.8 Support Files to Generate USBR Hazard Maps in ESRI Shapefiles format (only for Argus ONE Post Processing)

When you use the *Output* for creating shapefiles in the *Graphic Output Options* panel of the DIP, the program creates the `USBR_Hazard.EXP` that can be imported into Geographic Information systems (GIS) to create Hazard Map shapefiles based on the United States Bureau of Reclamation classification of flood hazards (USBR, 1988). The `USBR_Hazard.EXP` file has as many polygons as flooded elements containing five attributes corresponding of hazard level for houses, mobile homes, vehicles, adults and children. The attributes can get the values of 1, 2 or 3 depending on the hazard level:

Table 10.14 – USBR Hazard Classification in USBR_Hazard.EXP file.

ATTRIBUTE VALUE	HAZARD CLASSIFICATION
1	Low-danger zone
2	Judgment zone
3	High-danger zone

For further details about the USBR Hazard classification, consult the RiverFlow2D User's Guide and USBR (1988).

10.6 Output Files for QGIS Post-processing

10.6.1 General Results at Cells for QGIS

These ASCII files allow seamless transfer to QGIS Geographic Information System software. These files use the `.TEXTOUT` extension and are named as follows:

For English units: `CELL_TIME_ENG_DDDD_HH_MM_SS.TEXTOUT`
For Metric units: `CELL_TIME_METRIC_DDDD_HH_MM_SS.TEXTOUT`

Where DDDD is days, HH is hours, MM is minutes and SS seconds. For example

`CELL_TIME_ENG_0001_12_01_34.TEXTOUT`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line contains the number if cells (NELEM) and the number of cell parameters which is 16. Then it follows NELEM lines with results for each cell in the triangular element mesh as shown:

Table 10.15 – Variables Reported on CELL_TIME_*.TEXTOUT Output Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Velocity component in x direction U	ft/s	m/s
2	Velocity component in y direction V	ft/s	m/s
3	Velocity magnitude $\left\|\vec{U}\right\| = \sqrt{U^2+V^2}$	ft/s	m/s
4	Water surface elevation	ft	m
5	Depth H	ft	m
6	Initial bed elevation	ft	m
7	Bed elevation	ft	m
8	Bed elevation change since time = 0	ft	m
9	Froude number	-	-
10	Volumetric sediment discharge per unit width in x direction: Q_{sx}	ft^2/s	m^2/s
11	Volumetric sediment discharge per unit width in y direction: Q_{sy}	ft^2/s	m^2/s
12	Volumetric sediment discharge magnitude: $Q_s\sqrt{Q_{sx}^2+Q_{sy}^2}$	ft^2/s	m^2/s
13	Bed shear stress*: $\tau = \gamma HS_f = \gamma\left(Un/k\right)^2/H^{1/3}$	lb/ft^2	Pa
14	Accumulated rainfall volume	ft^3	m^3
15	Accumulated infiltration volume	ft^3	m^3
16	Manning's n	-	-

**English units:* $= 62.4lb/ft^3$, $k = 1.49$.
**Metric units:* $= 9810N/m^3$; $k = 1$, n = *Manning's coefficient.*

10.6.2 Maximum Values at Cells QGIS Files

These ASCII files report maximum values of velocity module, depth and water surface elevations and allow seamless transfer to QGIS Geographic Information System software. These files use the `.TEXTOUT` extension and are named as follows:

For English units: `<ProjectName>_CELLS_ENG_MAX.TEXTOUT`
For Metric units: `<ProjectName>_CELLS_METRIC_MAX.TEXTOUT`

The format for these files is as follows. The first line contains the number of cells (NELEM), and the number of cell parameters which is 4 by default, but 8 if the run was made with the Sediment Transport Module. There follows NELEM lines with velocity module, depth and water surface elevation for each cell as shown:

Table 10.16 – Variables Reported on the <ProjectName>_CELLS_ENG_MAX.TEXTOUT or <ProjectName>_CELLS_METRIC_MAX.TEXTOUT Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Maximum velocity magnitude $\sqrt{U_x^2 + U_y^2}$	ft/s	m/s
2	Maximum depth	ft	m
3	Maximum water surface elevation	ft	m
4	Maximum depth × velocity	ft^2/s	m^2/s
5	Maximum bed elevation*	ft	m
6	Minimum bed elevation*	ft	m
7	Maximum erosion depth*	ft	m
8	Maximum deposition depth*	ft	m

**Written only when using the sediment transport component.*

10.6.3 Time-to-Depth at Cells QGIS File

This ASCII file are generated when using the Plot RiverFlow2D Results button on the *Graphic Output Option* panel of the DIP. The file reports the time at which certain depths are reached during the simulation and allow seamless transfer to QGIS Geographic Information System software. In metric units the file includes time to 0.30 m of depth, time to 1 m and time to maximum depth for each cell. In English units the file includes time to 1 ft of depth, time to 3 ft and time to maximum depth for each cell. Time is always given in hours. The time-to-depth files have the following name:

`<ProjectName>_TIME2DEPTHS_CELLS.TEXTOUT`

The format for these files is as follows. The first line indicates the number of cells (NELEM) and the number of cell parameters (5 by default). For the file in Metric Units there follows NELEM lines with time to 0.30 m, time to 0.5 m, time to 1 m, time to maximum depth, and total inundated time for each cell as shown in Table 10.17. When the cell remains dry or depth is below 0.30 m the reported value is -1. For the file in English Units there follows NELEM lines with time to 1 ft, time to 2 ft, time to 3 ft, time to maximum depth, and total inundated time for each cell as shown in Table 10.17. When the cell remains dry or depth is below 1ft the reported value is -1.

Table 10.17 – Variables Reported on the <ProjectName>_TIME2DEPTHS_CELLS.TEXTOUT File.

COLUMN	VALUE	UNITS
1	Time to 0.30 m (Metric) or 1 ft. (English)*	*hrs.*
2	Time to 0.50 m (Metric) or 2 ft. (English)*	*hrs.*
3	Time to 1 m (Metric) or 3 ft. (English)*	*hrs.*
4	Time to maximum depth	*hrs.*
5	Total inundated time with depths $\geq$ 0.1 m	*hrs.*

**Time = -1 when cell is dry or depth does not reach 0.3 m (1ft).*

10.6.4 Hazard Intensity Values at Cells QGIS Files

These ASCII files report the Hazard Intensity values for various hazard classification used in different countries. These include the United State Bureau of Reclamation, Swiss methods, Criteria used in Austria and in the UK. The file can be used to create hazard maps in the QGIS Geographic Information System software. These files use the `.TEXTOUT` extension and are named as follows:

`<ProjectName>_CELLS_HAZARD.TEXTOUT`

The format for these files is as follows. The first line indicates the number of cells (NELEM), and the number of cell parameters (11). There follows NELEM lines with the hazard intensities for each cell as shown:

Table 10.18 – Variables Reported on the <ProjectName>_CELLS_HAZARD.TEXTOUT File.

COLUMN	VALUE	HAZARD ZONES
1	USBR Homes	0, 1, 2, and 3
2	USBR Passenger Vehicles	0, 1, 2, and 3
3	USBR Mobile Homes	0, 1, 2, and 3
4	USBR Adults	0, 1, 2, and 3
5	USBR Children	0, 1, 2, and 3
6	Swiss Method for Water Flooding	0, 1, 2, and 3
7	Swiss Method for Debris Flow	0, 1, 2, and 3
8	Austrian Method for River Flooding	0, 1, and 2
9	Austrian Method for Torrents Tr=100 yrs.	0, 1, and 2
10	Austrian Method for Torrents Tr=10 yrs.	0, 1, and 2
11	UK Method	0, 1, 2, and 3

.

10.7 Graphic Output Files

10.7.1 DXF Files

These files contain the velocity field for each output interval using colored vectors in AutoCAD DXF format. The file name is of this form: `VelField_TIME_DDDD_HH_MM_SS.DXF`.

10.7.2 GIF Output Files

These are raster output files in `.GIF` format showing the plot for each output interval. They can be imported in any word processor, spreadsheet or presentation programs.

10.7.3 VTK Output Files

RiverFlow2D Plus model and RiverFlow2D Plus GPU will output `.vtk` files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of RiverFlow2D results. ParaView (`www.paraview.org`) is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities.

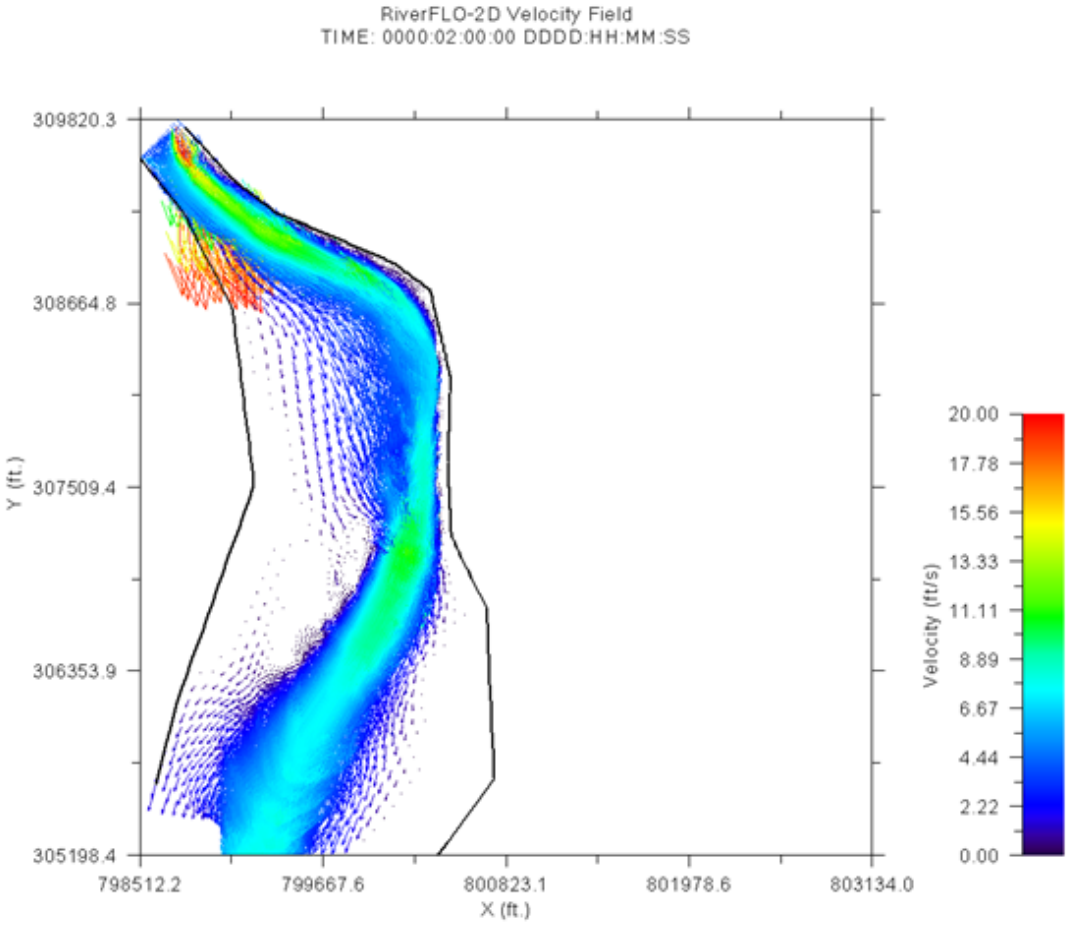


Figure 10.1 – Output color coded by depth and with velocity vectors

11 Data Input Program Tools

This section describes various utilities that are available through the RiverFlow2D Data Input Program.

11.1 HEC-RAS Data Extraction Tool

The purpose of this tool is to facilitate migrating existing HEC-RAS projects to RiverFlow2D. The program allows extraction of point elevation data from geo-referenced cross-section from the HEC-RAS one-dimensional model developed by the USACE. This elevation data can then be used by RiverFlow2D pre-processors as the base bathymetric and topographic data or to complement existing topography. The tool reads HEC-RAS geometry file names with extension .g01, .g02, etc., and creates X Y Z files that can be readily imported in SMS. The utility discriminates the elevations in the channel between the left and right bank on each cross section and exports the files as detailed in the following table.

Table 11.1 – HEC-RAS files with elevation data.

FILE NAME	CONTENT
<HEC-RAS file name>_ALL_STATIONS.EXP	Contains all elevation points in all cross sections in the for all reaches and cross sections in the *<HEC-RAS file name>.g0* file.
<HEC-RAS file name>_ALL_POLY.EXP	Contains polygons that include all elevation points in each reach.
<HEC-RAS file name>_CHANNEL_STATIONS.EXP	Contains only the elevation points between the left and right banks in all cross sections in the for all reaches in the *<HEC-RAS file name>.g0* file.
<HEC-RAS file name>_CHANNEL_POLY.EXP	Contains polygons that include only the elevation in the main channel for each reach.

The HEC-RAS Data Extraction Tool is accessible from the Tools panel of the Data Input Program.

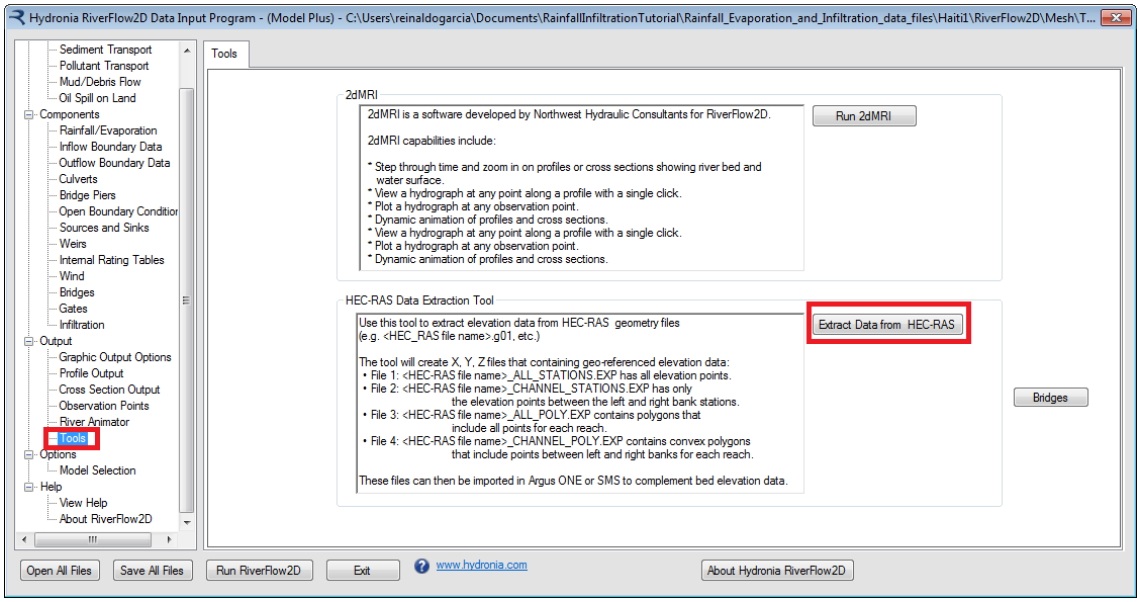


Figure 11.1 – HEC-RAS command on tools panel.

12 References

Abderrezzak, K. & Paquier, A. (2011), 'Applicability of sediment transport capacity formulas to dam-break flows over movable beds', *Journal of Hydraulic Engineering, ASCE* **137**, 209–221.

Ackers, P. & White, W. (1973), 'Sediment transport, new approach and analysis', *Journal of Hydraulic Div. ASCE* **99**(11), 2041–2060.

Akan, A. O. (1993), *Urban Stormwater Hydrology: A Guide to Engineering Calculations*, CRC Press.

ASCE (1996), *Hydrology Handbook*, 2nd edn, American Society of Civil Engineers (ASCE).

Ashida, K. & Michiue, M. (1972), 'Study on hydraulic resistance and bed-load transport rate in alluvial streams', *Transactions Japan Society of Civil Engineering* **206**, 59–69.

Bagnold, R. (1973), 'The nature of saltation of bed load transport in water', *Proc. of Royal Society. Ser. A* **332**, 473–504.

Bai, L. & Jin, S. (2009), 'A conservative coupled flow/transport model with zero mass error', *J. of Hydrodynamics* **21**, 166 – 175.

Baldock, T., Tomkins, M., Nielsen, P. & Hughes, M. (2004), 'Settling velocity of sediments at high concentrations', *Coastal Engineering* **51**, 91–100.

Belleundy, P. (2000), 'Restoring flow capacity in the loire river bed', *Hydrological Processes* **14**, 2331–2344.

Brufau, P., García-Navarro, P. & Vázquez-Cendón, M. (2004), 'Zero mass error using unsteady wetting-drying conditions in shallow flows over dry irregular topography', *Int. Journal for Numerical Methods in Fluids* **45**, 1047–1082.

Camenen, B. & Larson, M. (2005), 'A bedload sediment transport formula for the nearshore', *Estuarine, Coastal and Shelf Science* **63**, 249–260.

Caviedes-Voullième, D., García-Navarro, P. & Murillo, J. (2012), 'Influence of mesh structure on 2D full shallow water equations and SCS Curve Number simulation of rainfall/runoff events', *Journal of Hydrology* **448-449**(0), 39 – 59.

Cheng, N. (1997), 'Simplified settling velocity formula for sediment particle', *J. Hydraulic Eng., ASCE.* **123**(2), 149–152.

Cunge, J., Holly, F. & Vervey, A. (1980), *Practical Aspects of Computational River Hydraulics*, Pitman: London.

De Vriend, H., Zyserman, J., Nicholson, J., Roelvink, J., Péchon, P. & Southgate, H. (1993), 'Medium-term 2DH coastal area modelling', *Coastal Engineering* **21**(1-3), 193–224.

Engelund, F. & Fredsoe, J. (1976), 'A sediment transport model for straight alluvial channels.', *Nordic Hydrology* **7**, 293–306.

Engelund, F. & Hensen, E. (1967), A monograph on sediment transport to alluvial streams, Report, Copenhagen: Teknique Vorlag.

Fernandez-Luque, R. & van Beek, R. (1976), 'Erosion and transport of bed sediment', *Journal of Hydraulic Research, IAHR* **14**(2), 127–144.

FHWA (1978), Hydraulics of bridge waterways, Report EPD-86-101, FHWA (Federal Highway Administration).

Froehlich, D. (2003), *Users Manual for FESWMS FST2DH Two-dimensional Depth-averaged Flow and Sediment Transport Model*, Report No. FHWA-RD-03-053, Federal Highway Administration, Washington, DC.

García, R., Espinoza, R., Valera, E. & González, M. (2006), 'An explicit two-dimensional finite element model to simulate short- and long-term bed evolution in alluvial rivers', *Journal of Hydraulic Research* **44**(6), 755–766.

García, R., González, N. & O'Brien, J. (2009), *Dam-break flood routing. Chapter 4 in: Dam-Break Problems, Solutions and Case Studies*, WIT Press, Southampton-Boston.

Garde, R. & Ranga Raju, K. (1985), *Mechanics of Sediment Transportation and Alluvial Stream Problems*, John Wiley & Sons, New York.

Godunov, S. (1959), 'A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics', *Mat. Sb.* **47**, 271–306.

Gupta, R. S. (1995), *Hydrology and Hydraulic Systems*, Waveland Press.

Henderson, F. (1966), *Open Channel Flow*, MacMillan series in civil Engineering.

Horton, R. (1933), 'The role of infiltration in the hydrologic cycle', *Trans. Am. Geophys. Union* **14**, 446 – 460.

Jimenez, J. & O.S., M. (2003), 'A simple formula to estimate settling velocity of natural sediments', *J. Waterway, Port, Coastal, Ocean Eng* **129**(2), 70–78.

Juez, C., Murillo, J. & P., G.-N. (2014), 'Numerical assessment of bed-load discharge formulations for transient flow in 1d and 2d situations', *Journal of Hydroinformatics* **15**(4), 1234–1257.

Julien, P. (1998), *Erosion and Sedimentation*, Cambridge University Press.

Jury, W. & Horton, R. (2004), *Soil Physics*, John Wiley and Sons.

Karim, F. (1988), 'Bed material discharge prediction for non-uniform bed sediments', *Journal of Hydraulic Engineering, ASCE* **124**(6), 597–604.

LeVeque, R. (2002), *Finite-Volume Methods for Hyperbolic problems*, Cambridge University Press, New York.

Mein, R. G. & Larson, C. L. (1973), 'Modeling infiltration during a steady rain', *Water Resources Research* **9**(2), 384–394.

Meyer-Peter, E. & Muller, R. (1948), 'Formulas for bed-load transport', *Proc. of the Second Meeting. IAHR, Stockholm, Sweden* pp. 39–64.

Mishra, S. & Singh, V. (2003), *Soil Conservation Service Curve Number (SCS-CN) Methodology*, Kluwer Academic Publishers.

Morvan, H., Knight, D., Wright, N., Tang, X. & Crossley, A. (2008), 'The concept of roughness in fluvial hydraulics and its formulation in 1-d, 2-d and 3-d numerical simulation models', *Journal of Hydraulic Research* **46**(2), 191–208.

Murillo, J., Burguete, J., Brufau, P. & García-Navarro, P. (2005), 'Coupling between shallow water and solute flow equations: analysis and management of source terms in 2D', *International Journal for Numerical Methods in Fluids* **49**(3), 267–299.

Murillo, J., Burguete, J., Brufau, P. & García-Navarro, P. (2007), 'The influence of source terms on stability, accuracy and conservation in two-dimensional shallow flow simulation using triangular finite volumes', *International Journal of Numerical Methods in Fluids* **54**, 543–590.

Murillo, J. & García-Navarro, P. (2010*a*), 'An exner-based coupled model for two-dimensional transient flow over erodible bed', *Journal of Computational Physics* **229**(23), 8704–8732.

Murillo, J. & García-Navarro, P. (2010*b*), 'Weak solutions for partial differential equations with source terms: Application to the shallow water equations', *Journal of Computational Physics* **229**(11), 4327–4368.

Murillo, J., García-Navarro, P. & Burguete, J. (2008), 'Analysis of a second-order upwind method for the simulation of solute transport in 2d shallow water flow', *International Journal for Numerical Methods in Fluids* **56**(6), 661–686.

Murillo, J., García-Navarro, P. & Burguete, J. (2012), 'Wave riemann description of friction terms in unsteady shallow flows: Application to water and mud/debris floods', *Journal of Computational Physics* **231**, 1963–2001.

Naef, D., Rickenmann, D., Rutschmann, P., & McArdell, B. (2006), 'Comparison of flow resistance relations for debris flows using a one-dimensional finite element simulation model', *Nat. Hazards Earth Syst. Sci.* **6**, 155–156.

Nielsen, P. (1992), *Coastal Bottom Boundary Layers and Sediment Transport*, Advanced Series on Ocean Engineering. World Scientific Publishing.

Norman, J. M., Houghtalen, R. J. & Johnston, W. J. (1985), Hydraulic design of highway culverts, Hydraulic Design Series No. 5 FHWA-IP-85-15, Federal Highway Administration, Washington, D.C.

Parker, G. (1979), 'Hydraulic geometry of active gravel rivers', *Journal of Hydraulic Engineering, ASCE* **105**(9), 1185–1201.

Parker, G., Klingeman, P. & McLean, D. (1982), 'Bed load and size distribution in paved gravel bed streams', *Journal of Hydraulic Engineering* **108**(4), 544–571.

Ratia, H., Murillo, J. & García-Navarro, P. (2014), 'Numerical modelling of bridges in 2D shallow water flow simulations', *International Journal for Numerical Methods in Fluids* **75**(4), 250–272.

Raudkivi, A. (1990), *Loose boundary hydraulics*, Pergamon Press, Inc., Tarrytown, N.Y.

Rawls, W. & Brakensiek, D. (1983), A procedure to predict green and ampt infiltration parameters, *in* 'Proceeding of ASAE Conferences on Advances in Infiltration', Chicago, Illinois, pp. 102–112.

Rawls, W., Yates, P. & Asmussen, L. (1976), Calibration of selected infiltration equation for the georgia coastal plain, Report ARS-S-113, Agriculture Research Service.

Roe, P. (1981), 'Approximate riemann solvers, parameter vectors, and difference schemes', *Journal of Computational Physics* **43**(2), 357 – 372.

Rogers, B. & Borthwick, A.G.L. Taylor, P. (2003), 'Mathematical balancing of flux gradient and source terms prior to using roe's approximate riemann solver', *J. Comput. Phys.* **192**(2), 422–451.

Rosatti, G., Murillo, J. & Fraccarollo, L. (2008), 'Generalized roe schemes for 1D two-phase, free-surface flows over a mobile bed', *J. Comput. Phys.* **227**(24), 10058–10077.

Rubey, M. (1933), 'Settling velocities of gravel, sand and silt particles', *Amer. J. Soc.* **225**, 325–338.

Sha, Y. (1965), *Introduction to Sediment Dynamics*, Industry Press, 302-310.

Smart, G. (1984), 'Sediment transport formula for steep channels', *J. Hydr. Eng., ASCE.* **111**(3), 267–276.

Te Chow, V. (1959), *Open Channel Hydraulics*, McGraw-Hill, New York.

Te Chow, V., Maidment, D. & Mays, L. (1988), *Applied Hydrology*, McGraw-Hill Civil Engineering Series, MCGRAW-HILL Higher Education.

Toro, E. (2001), *Shock-Capturing Methods for Free-Surface Shallow Flows*, Wiley, New York, p. 109.

USBR (1988), Downstream hazard classification guidelines, ACER Technical Memorandum 5, United States Bureau of Reclamation (USBR).

USDA (1986), Urban hydrology for small watersheds, Environment agency report, United States Department of Agriculture (USDA).

USDA (2004), National engineering handbook part 630 hydrology, Technical report, United States Department of Agriculture (USDA).

Van Rijn, L. (1984*a*), 'Sediment pick-up functions', *J. Hydr. Eng., ASCE.* **110**(10), 1494–1502.

Van Rijn, L. (1984*b*), 'Sediment transport, part i: bed load transport', *J. Hydr. Eng., ASCE.* **110**(10), 1431–1456.

Van Rijn, L. (1984*c*), 'Sediment transport, part ii: suspended load transport', *J. Hydr. Eng., ASCE.* **110**(11), 1613–1641.

Vázquez-Cendón, M. (1999), 'Improved treatment of source terms in upwind schemes for the shallow water equations in channels with irregular geometry', *Journal of Computational Physics* **148**(2), 497–526.

Vreugdenhil, C. (1994), *Numerical methods for shallow water flow*, Water Science and Technology Library. Kluwer Academic Publishers.

Wilcock, P. & Crowe, J. (2003), 'Surface-based transport model for mixed-sized sediment', *Journal of Hydraulic Engineering* **129**(2), 120–128.

Wong, M. (2003), 'Does the bedload transport relation of meyer-peter and muller fits its own data?', *Proc. 30th IAHR-Congress, Thessaloniki, Greece, 8 pp.* .

Wong, M. & Parker, G. (2006), 'Re-analysis and correction of bed load relation of meyer-peter and mller using their own database', *Journal of Hydraulic Engineering* **132**(11), 1159–1168.

Wu, W. (2008), *Computational River Dynamics*, Taylor and Francis.

Wu, W. & Wang, S. (2006), 'Formulas for sediment porosity and settling velocity', *J. Hydraulic Eng., ASCE.* **132**(8), 858–862.

Yang, C. (1996), *Sediment Transport Theory and Practice*, McGraw Hill, New York.

Yarnell, D. (1934*a*), 'Bridge piers as channel obstructions', *U.S. Department of Agriculture, Tech. Bull.* **442**.

Yarnell, D. (1934*b*), 'Pile trestles as channel obstructions', *U.S. Department of Agriculture, Tech. Bull.* **429**.

Zanke, U. (n.d.), Berechnung der sinkgeschwindigkeiten von sedimenten, Technical report, Technischen Universitt Hannover.

Zhang, R. (1961), *River Dynamics (in Chinese)*, Industry Press.