OilFlow2D
Model for Oil Spills
on Water and Land

Reference Manual

August 2026

Hydronia

OilFlow2D™, RiverFlow2D™ models, 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. 2011-2026.
OilFlow2D™ and RiverFlow2D™ are registered trademarks of Hydronia, 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 2026.
Technical Support: support@hydronia.com
Web site: www.hydronia.com

Content

List of Figures ix

1 Introduction 1
- 1.1 Summary of OilFlow2D Features and Capabilities . . . 2
  - 1.1.1 Main capability . . . 2
  - 1.1.2 Mesh Generator . . . 2
  - 1.1.3 Numerical Engine . . . 2
  - 1.1.4 Hydraulic Components . . . 3
  - 1.1.5 Input Data Formats . . . 3
  - 1.1.6 Initial Conditions . . . 3
  - 1.1.7 Boundary Conditions . . . 3
  - 1.1.8 Output Options . . . 4
  - 1.1.9 Output of Results for Maximum Values . . . 4
  - 1.1.10 Pollutant Transport Module PL . . . 4

2 Installing and Activating OilFlow2D 6
- 2.1 Hardware Requirements . . . 6
- 2.2 Software Installation . . . 6
- 2.3 Software Activation . . . 7
  - 2.3.1 Standalone Activation . . . 7
  - 2.3.2 Network Server Installation . . . 8
  - 2.3.3 Network Client Installation . . . 10
- 2.4 Enabling Hydronia Plugins in QGIS . . . 11
  - 2.4.1 Enabling OilFlow2D Plugin (pre-v10 or standalone) . . . 12
  - 2.4.2 Enabling Macros in QGIS . . . 13
- 2.5 Troubleshooting . . . 13
  - 2.5.1 Finding your License Key . . . 13
  - 2.5.2 Find Who is Using the Software in a Network Installation . . . 14
  - 2.5.3 ERROR 641: “You have reached the limit on the maximum number of simultaneous users of this program.” . . . 14
  - 2.5.4 ERROR 659: “This program is configured for network installation only. It cannot be installed as a standalone system.” . . . 15
  - 2.5.5 ERROR 660: “This program is configured for standalone installation only. It cannot be installed as a network system” . . . 15
  - 2.5.6 ERROR 739: “This program has been installed or copied too many times.” . . . 15
- 2.6 OilFlow2D Documentation . . . 15

2.7 OilFlow2D Technical Support . . . 16
2.8 OilFlow2D Tutorials . . . 16

3 Overview of OilFlow2D 17

4 Mesh Generation in OilFlow2D 21
4.1 Cell-size control using the Domain outline . . . 22
4.2 Cell-size control using Polylines in the MeshDensityLine Layer . . . 23
4.3 Cell-size control using Polygons in the MeshDensityPolygon Layer . . . 23
4.4 Cell-size control using Polylines in the MeshBreakLine Layer . . . 24
4.5 Boundary Conditions . . . 25
4.6 Mesh Spatial Data . . . 26
4.6.1 Mannings' n . . . 26
4.7 OilFlow2D Toolbar Functions . . . 27
4.7.1 New OilFlow2D Project . . . 27
4.7.2 Generate TriMesh . . . 28
4.7.3 Export Files for OilFlow2D . . . 29
4.7.4 Maps . . . 29
4.7.5 Animations . . . 29
4.7.6 Cross Sections . . . 29
4.7.7 Tools . . . 29

5 Oil Spill on Land Model 33
5.1 Assumptions of the Viscous Flow Model . . . 33
5.2 Flow equations considering prescribed temperature variations . . . 34
5.3 Equations considering heat transfer . . . 34
5.3.1 Fluid properties and friction laws . . . 35
5.3.2 Temperature Source Term and Heat Transfer Mechanisms . . . 35
5.4 Oil Retention . . . 37
5.5 Pipeline Break Spill Hydrograph Equations . . . 38
5.5.1 Initial Steady Pipeline Head . . . 38
5.5.2 Upstream Leak Flow . . . 39
5.5.3 Downstream Gravity Drainage . . . 39
5.5.4 Combined Source Hydrograph . . . 41
5.6 Finite-Volume Numerical Solution . . . 41
5.6.1 Numerical Optimizations . . . 45
5.6.2 Stability Region . . . 46
5.7 Open Boundary Conditions . . . 46
5.7.1 Single Variable Boundary Condition Types (BCTYPE 1 and 6) . . . 48
5.7.2 Discharge Rating Table (BCTYPE 9) . . . 49
5.7.3 "Free" Open Boundaries (BCTYPE 10, 11) . . . 49
5.7.4 Uniform Flow Boundary Condition (BCTYPE 12) . . . 50
5.7.5 Numerical Implementation of Open Boundaries . . . 50
5.7.6 Closed Boundaries . . . 56
5.8 Dry/Wet Cell Modeling . . . 57

5.8.1 Cell definitions Based on Dry and Wet Conditions . . . 57
5.9 Volume Conservation . . . 58
5.10 Manning's n roughness Coefficients . . . 59

6 Oil Spill on Water Model **60**
6.1 3D Oil Spill Trajectory Algorithm . . . 60
6.1.1 Three-dimensional 3D Flow Field . . . 62
6.1.2 Dispersion Coefficients . . . 63
6.1.3 Comparison with Analytical Solution . . . 63
6.1.4 Comparison with Experiments . . . 64
6.2 Evaporation . . . 65
6.2.1 Stiver-Mackay formulation . . . 65
6.2.2 Fingas formulation . . . 66
6.3 Emulsification . . . 70
6.4 Density and Viscosity Change . . . 71
6.5 Oil Spreading Formulation . . . 71
6.5.1 A_k and B_k Formulation . . . 73
6.6 Containment Booms . . . 75
6.7 Setting up a Oil Spill on Water Simulation . . . 75

7 Pollutant Transport Model: PL **76**
7.1 Model Equations . . . 76
7.2 Pollutant Transport Finite-Volume Numerical Solution . . . 77
7.3 Entering Data for the Pollutant Transport Model . . . 79
7.4 Assumptions of the Pollutant Transport Model . . . 79

8 Code Parallelization **80**
8.1 OilFlow2D CPU . . . 80
8.2 OilFlow2D GPU . . . 81

9 Components to Model Internal Structures **83**
9.1 Bridges Component . . . 83
9.1.1 Bridge Geometry Data File . . . 84
9.1.2 Bridge Calculations . . . 87
9.2 Bridge Piers . . . 91
9.2.1 Bridge Pier Calculation . . . 92
9.3 Culverts Component . . . 94
9.3.1 Culvert Calculation using a Rating Table (CulvertType = 0) . . . 95
9.3.2 Culvert Calculation using Culvert Characteristics (CulvertType = 1,2) . . . 95
9.3.3 Assumptions of Culvert Calculations . . . 101
9.3.4 Culvert Multiple-Cell Volume Exchange Tool in QGIS . . . 101
9.4 Gates Component . . . 103
9.4.1 Gate Calculations . . . 105
9.5 Infiltration . . . 106
9.5.1 Horton Infiltration Model . . . 106
9.5.2 Green-Ampt Infiltration Model . . . 108

9.5.3 SCS-CN Model . . . 109
9.6 Wind Component . . . 111
9.7 Internal Rating Tables . . . 111
9.7.1 Internal Rating Table Calculations . . . 112
9.7.2 Assumptions of Internal Rating Table Calculations . . . 112
9.8 Weirs . . . 113
9.8.1 Weir Calculations . . . 114
9.8.2 Assumptions of Weir Calculations . . . 114
9.9 Dam Breach Modeling . . . 115
9.9.1 Prescribed dam breach . . . 115
9.9.2 Dam breach failure by piping erosion . . . 116
9.10 Flow discharge through the piping cross section . . . 116
9.11 Pipe erosion . . . 117
9.12 Overtopping erosion . . . 118
9.13 Dambreach flow as internal boundary condition . . . 119

10 Hydronia Data Input Program (DIP) 121
10.1 Control Data Panel (.DAT file) . . . 122
10.2 Oil Spill on Land for the OilFlow2D model (.OILP File) . . . 126
10.3 Oil Spill on Water module (.OILW File) . . . 129
10.3.1 Oil Spill on Water Control Data Tab . . . 130
10.3.2 Oil Spill on Water Trajectory Tab . . . 131
10.3.3 Oil Spill on Water Evaporation Tab . . . 132
10.4 Pollutant Transport panel (.SOLUTES) . . . 132
10.5 Graphic Output Options Tab (.PLT File) . . . 133
10.6 Profile Output Panel (.PROFILES File) . . . 135
10.7 Cross Section Output Panel (.XSECS File) . . . 136
10.8 Culverts Panel (.CULVERTS File) . . . 137
10.9 Internal Rating Tables Panel (.IRT File) . . . 139
10.10 Weirs Panel (.WEIRS File) . . . 139
10.11 Sources/Sinks Panel (.SOURCES File) . . . 140
10.12 Bridge Scour Panel (.SCOUR File) . . . 141
10.13 Bridge Piers Panel (.PIERS File) . . . 143
10.14 Observation Points Panel (.OBS File) . . . 144
10.15 Tools Panel . . . 145
10.15.1 Process Rainfall and Evaporation Data from ASCII Grid Files Tool . . . 145
10.15.2 HEC-RAS Data Extraction Tool . . . 146

11 Input Data File Reference 148
11.0.1 Run Control Data File: .DAT . . . 150
11.1 Mesh Data . . . 157
11.1.1 Mesh Data File: .FED . . . 157
11.1.2 Open Boundary Conditions Data Files: .IFL and .OBCP . . . 158
11.1.3 Mesh Boundary Data File: .TBA . . . 162
11.2 Bridges . . . 164

11.2.1 Bridges Data File: .BRIDGES . . . 164
11.2.2 Bridge Cross Section Geometry Data File . . . 165
11.3 Culverts Data File: .CULVERTS . . . 167
11.3.1 Example of a .CULVERTS file . . . 167
11.3.2 Culvert Depth-Discharge Rating table Data Files for CulvertType=0 . . . 168
11.3.3 Culvert Characteristic Data Files for CulvertType = 1, 2 . . . 169
11.3.4 Example of the culvert characteristic data file . . . 169
11.3.5 Comments for the .CULVERTS and culvert characteristics files . . . 174
11.4 Dam Breach Data File: .DAMBREACH . . . 175
11.4.1 Example of a .DAMBREACH file . . . 175
11.4.2 Breach time evolution data file for prescribed failure mode . . . 177
11.4.3 Comments for the .DAMBREACH file . . . 177
11.5 GATES Data Files: .GATES . . . 178
11.5.1 Example of a .GATES File . . . 178
11.5.2 Gate Aperture Time Series File . . . 179
11.5.3 Example of a Gates Aperture Data File . . . 179
11.6 Internal Rating Table Data File: .IRT . . . 179
11.6.1 Example of a .IRT file . . . 180
11.6.2 Comments for the .IRT file . . . 180
11.7 Rainfall And Evaporation Data File: .LRAIN . . . 181
11.7.1 Comments for the .LRAIN file . . . 181
11.7.2 Example of a Hyetograph and Evaporation data file . . . 182
11.7.3 Comments for the Hyetograph and Evaporation data file . . . 182
11.8 Infiltration Data File: .LINF . . . 182
11.8.1 Example of a .LINF file . . . 183
11.8.2 Comments for the .LINF file . . . 183
11.8.3 Example of a Infiltration parameter data file . . . 184
11.9 Manning's n Variable with Depth Data File: .MANNN . . . 185
11.9.1 Comments for the .MANNN file . . . 186
11.9.2 Example of a Manning's variable with depth data file . . . 186
11.9.3 Comments for the Manning's n variable with depth data file . . . 186
11.9.4 Bridge Piers Drag Forces File: .PIERS . . . 187
11.10 Bridge Pier and Scour Data File: .SCOUR . . . 189
11.10.1 Example of a .SCOUR file . . . 190
11.10.2 Comments for the .SCOUR File . . . 191
11.11 Sources and Sinks Data File: .SOURCES . . . 191
11.11.1 Example of a .SOURCES file . . . 192
11.11.2 Comments for the .SOURCES File . . . 192
11.12 Multiple Sources file . . . 192
11.12.1 Example of a multiple source file . . . 193
11.13 Weirs Data Files: .WEIRS and .WEIRP . . . 194
11.13.1 Example of a .WEIRS file . . . 194
11.13.2 Example of a .WEIRP file . . . 195
11.13.3 Comments for the .WEIRS File . . . 196

11.14 Wind Data File: .WIND . . . 196
11.14.1 Example of a file . . . 197
11.14.2 Comments for the File . . . 197
11.14.3 Wind Velocity Data File . . . 197
11.14.4 Example of a Wind Velocity and Data File . . . 198
11.15 Oil Containment Booms Data File: .BOOMS . . . 198
11.15.1 Example of a .BOOMS file . . . 198
11.16 Oil on Land Model File: .OILP . . . 199
11.16.1 Example of a .OILP file . . . 201
11.16.2 Temperature-Viscosity-Density Table file . . . 205
11.16.3 Temperature vs Density Time Series file . . . 206
11.16.4 Temperature vs Viscosity Time Series file . . . 206
11.16.5 Temperature vs Yield Stress Time Series file . . . 206
11.16.6 Environmental Parameters Time Series file . . . 207
11.16.7 Oil Retention Depth Data File: .DETENINITIAL . . . 207
11.17 Oil on Water File: .OILW . . . 208
11.17.1 Example of a .OILW file . . . 212
11.17.2 Accumulated Volume Input File . . . 215
11.17.3 Example of a Accumulated Volume File . . . 215
11.17.4 Spill Release File . . . 215
11.17.5 Example of Spill Release File . . . 215
11.18 Pollutant Transport Module Data File: .SOLUTES . . . 216
11.18.1 Example of a .SOLUTES file . . . 216
11.18.2 Pollutant Transport Module Initial Concentration Data File: .CINITIAL . . . 217
11.19 Urban Drainage Module Data File: .LSWMM . . . 218
11.19.1 Example of a .LSWMM file . . . 218
11.19.2 Depth-Discharge Data Files . . . 219
11.20 Output control data . . . 220
11.20.1 Observation Points Data File: .OBS . . . 220
11.20.2 Graphical Output Control Data File: .PLT . . . 221
11.20.3 Data for Profile Result Output: .PROFILES . . . 224
11.20.4 Cross Section Data for Result Output File: .XSECS . . . 224
11.21 Elevation data . . . 225
11.21.1 X Y Z data with header . . . 226
11.22 Boundary conditions data files . . . 226
11.22.1 One Variable Boundary Condition Files . . . 226
11.22.2 Two Variables Boundary Condition Files . . . 227
11.22.3 Multiple-Variable Boundary Condition Files . . . 228
11.22.4 Stage-Discharge Data Files . . . 229
11.22.5 Culvert Depth-Discharge Data Files . . . 229

12 Output File Reference **231**
12.1 Output File Overview . . . 231
12.1.1 Essential files required to generate maps, graphics and animations . . . 233
12.1.2 Output times .outfiles file . . . 234

12.1.3 Output times for the Oil Spill on Water model .outfilesoilw file 235
12.1.4 Run Options Summary .outi and .oute files 235
12.1.5 Mesh Data and Mesh Metrics .meshouti and .meshoute files 236
12.1.6 Run Summary .rout file 239
12.1.7 Maximum Value Tabular .maxi and .maxe Files 239
12.1.8 Observation Point Output 241
12.1.9 Hot Start `2binitialized.hotstart` File (Version 10, 2026) 241
12.1.10 Mass Balance Output File 242
12.2 Component Output Files 243
12.2.1 Booms .OUTBOOMS Output File 243
12.2.2 Culvert CULVERT_culvertID.out Output Files 244
12.2.3 Internal Rating Table IRT_irtID.out Files 244
12.2.4 Weir Output .weiri and .weire Files 245
12.3 Cross Section and Profile Output Files 245
12.3.1 General Cross Section .xseci and .xsece Files 245
12.3.2 Cross Section Hydrograph .xsech and .xsecsed Files 246
12.3.3 Profile .prfi and .prfe Files 247
12.4 Output Files for QGIS Post-processing 248
12.4.1 General Results at Cells 248
12.4.2 Oil Spill on Land Considering Heat Transfer Concentration Files (OilFlow2D Overland Spills Module) 249
12.4.3 Pollutant Concentration Files (PL Module) 249
12.4.4 Oil and Plastics Output Files (OilFlow2D Spills On Water and Plastics Modules) 250
12.4.5 Maximum Value Files 254
12.4.6 Time-to-Depth at Cells Output File 254
12.4.7 Hazard Intensity Values at Cells Output File 255
12.5 VTK Output Files for Paraview 256

13 OilFlow2D Tools **257**
13.1 Automated Batch Execution of Model Simulations 257
13.1.1 Using a MS-DOS script to perform multiple batch runs 257
13.1.2 Using a Python script within QGIS to perform multiple batch runs 258
13.1.3 Using a Python script through a batch file to perform multiple runs 258

List of Figures

1.1 OilFlow2D triangular-cell mesh. . . . 2
1.2 Plot showing oil depths computed with the OilFlow2D model. . . . 5

2.1 QGIS Plugins window showing Hydronia2DMS installed for version 10 and later. . . . 12
2.2 Location of OilFlow2D Documentation folder. . . . 16

3.1 Standard OilFlow2D layers created when using the New OilFlow2D Project command. 17

4.1 Typical OilFlow2D flexible mesh. . . . 21
4.2 *CellSize* dialog. . . . 22
4.3 Mesh generated based on an external polygon with *CellSize* = 50ft , and an internal polygon with *CellSize* = 10ft, both entered on the *Domain Outline* layer. . . . 22
4.4 Mesh generated based on the polygons of Figure adding the two polylines on the *MeshDensityLine* layer. . . . 23
4.5 Mesh generated based on the polygon on the *MeshDensityPolygon* layer. . . . 24
4.6 Mesh generated based on the polygons and polylines of Figure adding the one polylines on the *MeshBreakLine* layer. Note how unlike in the MeshDensityLine layer, the polylines entered in the *MeshBreakLine* layer force the mesh to have nodes along the polylines. . . . 24
4.7 QGIS Layer Panel showing the *Boundary Conditions* layer selected. . . . 25
4.8 Select Toggle Editing (pencil) and Add Feature. . . . 25
4.9 Inflow Boundary Condition Polygon. . . . 25
4.10 Boundary Condition Dialog. . . . 26
4.11 Boundary Condition BC Data panel. . . . 26
4.12 QGIS Layer Panel showing the Manning N layer selected. . . . 27
4.13 Select Toggle Editing (pencil) and Add Feature. . . . 27
4.14 Manning's n polygon. . . . 27
4.15 New Project Dialog. . . . 28
4.16 Mesh Generation Options. . . . 29
4.17 Export Files Options. . . . 29
4.18 Schematic view of the landslide. . . . 30
4.19 Add LandSlides layer. . . . 30
4.20 Depth or volume landslide attributes. . . . 31
4.21 Concentration for all classes forming the landslide material. . . . 31

5.1 The Shear Infinite Landslide approach scheme. . . . 37
5.2 Piecewise uniform representation of the flow variables. . . . 42

5.3 Cell parameters. . . . 43
5.4 Cell parameters. . . . 44
5.5 Open and closed boundary conditions. . . . 47
5.6 Required gap between adjacent open boundary conditions. . . . 48
5.7 Inflow water discharge imposed as velocities (BCTYPE 6). . . . 49
5.8 Rectangular inlet cross section. . . . 52
5.9 Irregular inlet cross section. . . . 53
5.10 Evaluation of d_{min}. . . . 54
5.11 New water level for the inlet section. . . . 55
5.12 Solid wall condition. . . . 57

6.1 Comparison of numerical and analytical solution for instantaneous spill for different number of particles Np. . . . 64
6.2 Mesa oil spreading experiments. Mackay and OilFlow2D formulations. . . . 65
6.3 Lago Medio oil spreading experiments. Mackay and OilFlow2D formulation. . . . 65
6.4 A_k vs. oil spreading coefficient σ in sea water . . . 74
6.5 B_k as a function of oil density ρ_o . . . 75

7.1 Physical representation of solute mass exchange between cells with $q^{\downarrow}_{i-1/2}, q^{\downarrow}_{i+1/2} > 0$. 78
7.2 Extraction of mass solute in an outlet boundary cell. . . . 78
7.3 DIP *Pollutant Transport* panel. . . . 79

8.1 Speed up using OilFlow2D parallelized code as a function of number of processor cores. . . . 81
8.2 Speed up using OilFlow2D parallelized code as a function of number of processor cores. . . . 82

9.1 DIP *Control Data* panel with the Bridges component selected. . . . 84
9.2 Front view of a bridge cross section. . . . 87
9.3 Top view of a bridge showing the cross sections of interest. Only two piers are depicted for simplicity. . . . 88
9.4 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in free surface bridges. . . . 88
9.5 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in a partially submerged bridges. . . . 89
9.6 Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in fully submerged bridges. . . . 89
9.7 Application of the scheme in triangular structured meshes. Normal bridge (left) and oblique bridge (right). . . . 90
9.8 Bridge pier proportions used to asses the influence of the structure width. . . . 91
9.9 Influence of the structure width on the total head change (ΔH) across the bridge as a function of the Froud number downstream. . . . 91
9.10 DIP *Control Panel* dialog with the Bridge Piers component selected. . . . 92
9.11 Piers inside cells. . . . 93
9.12 Schematic view of a rectangular pier. . . . 93
9.13 DIP *Global Parameters* dialog with the Culverts Component selected. . . . 94

9.14 Inlet and outlet culvert polygons. . . . 102
9.15 Schematic cut view perpendicular to a gate structure. . . . 103
9.16 Flow modes across gates. . . . 104
9.17 DIP *Control Data* panel with the Gates Component selected. . . . 104
9.18 Water levels for discharge under a gate in submerged conditions formulated as in (G1). . . . 105
9.19 Water levels for discharge under a gate in submerged conditions formulated as in (G2). . . . 105
9.20 Schematic dimensions for prescribed dam breach failure mode. . . . 116
9.21 Internal boundary cells. . . . 119

10.1 DIP Open Project Dialog. . . . 121
10.2 Main DIP window. . . . 122
10.3 *Control Data* Panel. . . . 123
10.4 Oil Spill on Land Basic Model Panel (OilFlow2D model). . . . 126
10.5 Oil Spill on Land Heat Transfer Model Panel. Environmental Parameters Tab (OilFlow2D model). . . . 127
10.6 Oil Spill on Land Heat Transfer Model Panel. Density Tab (OilFlow2D model). . . . 127
10.7 Oil Spill on Land Heat Transfer Model Panel. Viscosity Tab (OilFlow2D model). . . . 128
10.8 Oil Spill on Land Heat Transfer Model Panel. Yield Stress Tab (OilFlow2D model). 128
10.9 Oil Spill on Land Heat Transfer Model Panel. Evaporation Tab (OilFlow2D model). 129
10.10 Oil Spill on Land Heat Transfer Model Panel Boundary Control Tab (OilFlow2D model).129
10.11 Oil Spill on Water module selected in the Control Data Panel (OilFlow2D model). . . . 130
10.12 Oil Spill on Water Panel. Control Data Tab (OilFlow2D model). . . . 130
10.13 Oil Spill on Water Panel. Trajectory Tab (OilFlow2D model). . . . 131
10.14 Oil Spill on Water Panel. Evaporation Tab (OilFlow2D model). . . . 132
10.15 Pollutant Transport panel. . . . 133
10.16 Graphic Output Panel. . . . 134
10.17 Profile Output File. . . . 136
10.18 Cross Section Output Panel. . . . 137
10.19 Culverts Panel showing data in rating curve. . . . 138
10.20 Internal Rating Tables Panel. . . . 139
10.21 Weirs Panel. . . . 140
10.22 Sources/Sinks Panel. . . . 141
10.23 Bridge Scour Panel. . . . 142
10.24 Bridge Piers Panel. . . . 143
10.25 Observation Points Panel. . . . 144
10.26 Tools Panel. . . . 145

11.1 Example of a OilFlow2D Mesh. . . . 148
11.2 Front view of a bridge cross section. . . . 166
11.3 Multi-sources file read. . . . 193
11.4 Multi-sources dialog. . . . 193

1 Introduction

This section presents the system of equations, the formulation of the boundary conditions, and the finite-volume scheme used in OilFlow2D. The information can be expanded in the references.

OilFlow2D is a numerical model that simulates the spreading of oil and viscous fluids on the land surface, and the trajectory and fate of crude oils in water. OilFlow2D is part of the Hydronia suite of models that includes RiverFlow2D, and RiverFlow2D GPU. OilFlow2D overland spill component can simulate fluid depths and velocity over complex terrain at high resolution and with remarkable stability, accuracy and speed, accounting for oil evaporation and infiltration. The use of adaptive triangular-cell meshes enables resolving the flow field around key features in irregular geometry and complex terrain environments.

OilFlow2D oil spill in water component computes oil trajectories in rivers, estuaries and coastal waters, incorporating the effects of oil evaporation, emulsification, dissolution, and interaction of oil with shores.

This version of the OilFlow2D includes an advanced Graphical User Interface based upon a plugin developed by Hydronia for the Open Source Geographical Information System QGIS (www.qgis.org). The integration of the OilFlow2D plugin and the QGIS software system provides interactive functions to generate and refine the flexible mesh used by OilFlow2D, familiar GIS layers and tools to construct a high-level representation of the model, facilitating assigning boundary conditions and Manning's n values, and all the other data layers required by the OilFlow2D components, allowing the user to efficiently manage the entire modeling process. OilFlow2D for QGIS offers a comprehensive set of visualization tools including map rendering, animations, and exporting graphs in shapefile format and Google Earth.

OilFlow2D computation engine uses an accurate, fast, and stable finite-volume solution method that ensures exact mass conservation and stable solutions through dynamic and automatic adjustment of the numerical time step. OilFlow2D can integrate hydraulic structures such as culverts, weirs, bridges, gates, weirs, and integrate the effect of wind on the water surface.

This reference manual provides instructions to install the OilFlow2D for QGIS software 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 a detailed description of the input data files and output files. A separate tutorial document provides detailed guidelines to get started using OilFlow2D capabilities.



Figure 1.1 – OilFlow2D triangular-cell mesh.

1.1 Summary of OilFlow2D Features and Capabilities

1.1.1 Main capability

- Simulation of crude oils and viscous fluids on water and over complex terrain.

1.1.2 Mesh Generator

- Automatic generation of flexible triangular-cell mesh.
- Mesh refinement along density polylines or inside polygons.
- Use of breaklines to adjust mesh along terrain features.
- Use of multiple Digital Elevation Models in the same mesh according to user selected areas.
- Spatially interpolation of DEM elevations to cells.
- Mesh cell and node numbering optimization.

1.1.3 Numerical Engine

- Spatial discretization using triangular cells.
- High performance Finite-Volume engine.

- Automatic and dynamic selection of the computational time step.
- Dry cell integration.
- Exact volume conservation.
- Double-precision computations for higher accuracy.
- Fully parallelized with OpenMP for faster execution in Multiple-Core computers.
- GPU version for up to $>$ 700X faster simulations using NVIDIA GPU Graphic Cards.

1.1.4 Hydraulic Components

- Internal dam and levee breaches.
- 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.
- Dam Breach (prescribed failure, piping and overtopping erosion)
- Internal hydraulic structures.
- Sources and sinks.
- Spatially distributed wind stress.
- Bridge pier and abutment scour.
- Sequential automated batch runs for each input source/spill.

1.1.5 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.
- Any raster or vector data format accepted by QGIS.

1.1.6 Initial Conditions

- Dry-bed.
- User-defined constant water elevations over polygons.
- User-defined variable water elevations given by raster data.

1.1.7 Boundary Conditions

- Water discharge hydrograph.
- Water discharge and water elevation vs time.
- Water elevation vs time.

- Uniform flow.
- Rating tables.
- Free outflow.
- Inflow pollutant concentrations (PL Module).

1.1.8 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.
- Froude Number.
- Time to 0.3 m (1 ft), time to 0.5 m (2 ft) , time to 1 m (3 ft), time to peak depth, and frontal wave arrival time.
- Inundation time during which depth is greater than 0.1 m or 4 in.
- Frontal wave arrival time.
- ESRI shapefiles.
- GIS post processing plots including shapefiles and raster images.
- Paraview VTK.

1.1.9 Output of Results for Maximum Values

- Maximum velocity magnitude.
- Maximum depths.
- Maximum water surface elevations.
- Maximum depth times velocity.
- Maximum Shear Stress.
- Maximum Impact force per unit width.

1.1.10 Pollutant Transport Module PL

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

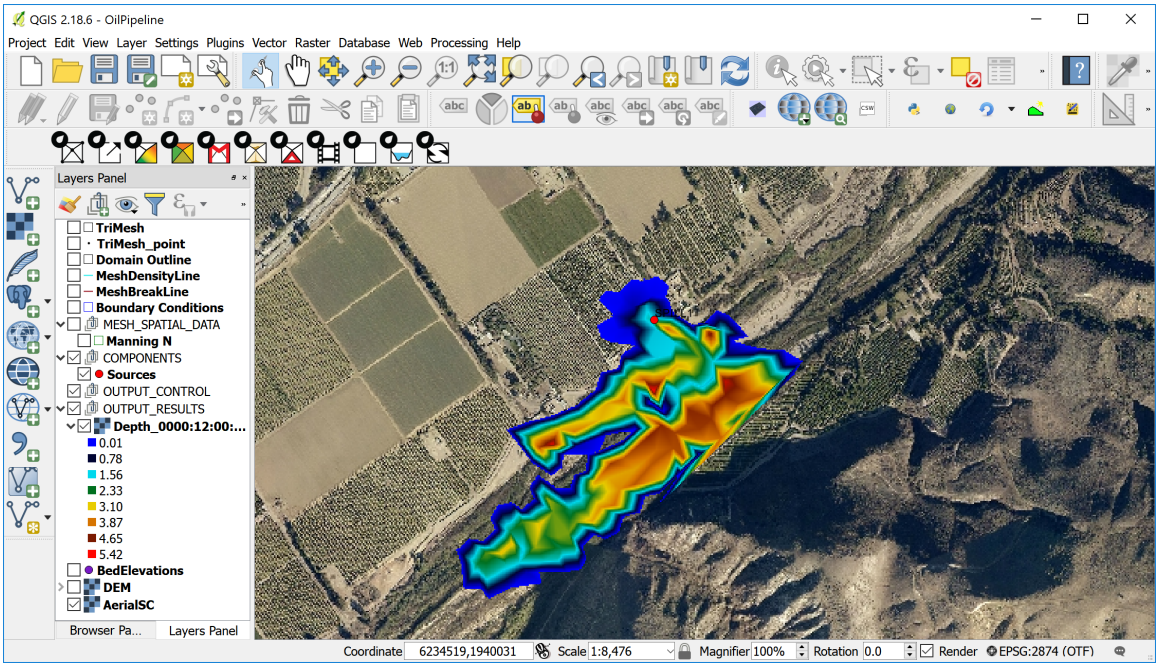


Figure 1.2 – Plot showing oil depths computed with the OilFlow2D model.

2

Installing and Activating OilFlow2D

OilFlow2D installation includes the most current version of QGIS that has been tested to work with the model. This section will assist you in setting up OilFlow2D and enabling it in QGIS.

2.1 Hardware Requirements

OilFlow2D is supported on 64-bit computers running MS-Windows Operating System versions 7 through 11. It is recommended to use a computer with a minimum of 4 GB of RAM and at least 10 GB of free hard disk space. OilFlow2D 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, using the OilFlow2D GPU option, the model can take advantage of NVIDIA Graphic Processing Unit (GPU) cards to run up to 700 times faster than in single-processor computers.

2.2 Software Installation

1. 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. Download your software from the link provided when purchased.

3. Run the installation.

> **Note**
>
> Reboot will be required. Please reboot before proceeding to the next section.

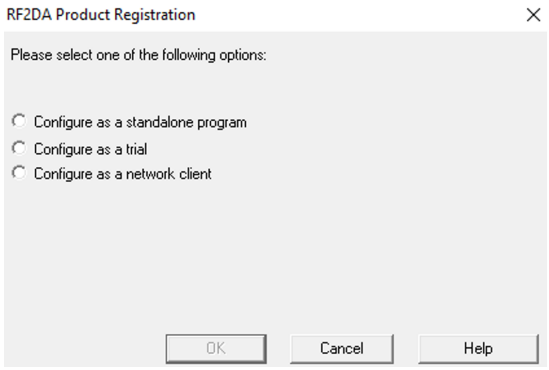
2.3 Software Activation

- Standalone - A single license for one computer.
- Network - A centralized license that allows multiple concurrent users depending on license count purchased. This requires additional license manager software to be installed on an accessible computer on your network.

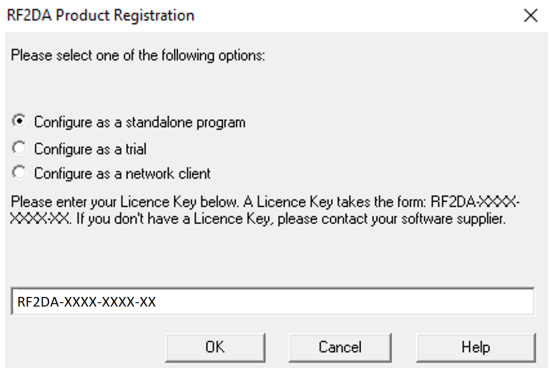
2.3.1 Standalone Activation

Use this activation mode if you have received a single-user stand-alone software license key. If you received a network license key, please proceed to the the next section.

1. To activate your software, you must open DIP (Hydronia DIP) shortcut on your desktop.
2. In the *Control Data* section on the left side, go to *Options* and select *License*.
3. You will be prompted to select one of three options:
    1. Reactivate License
    2. Install Network License Server
    3. Check for Updates
4. Select *Reactivate License*.
5. The following dialog will appear:

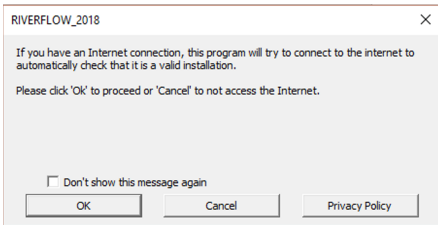


6. Select *Configure as a standalone program*.
7. Enter the license key provided to you (e.g.):

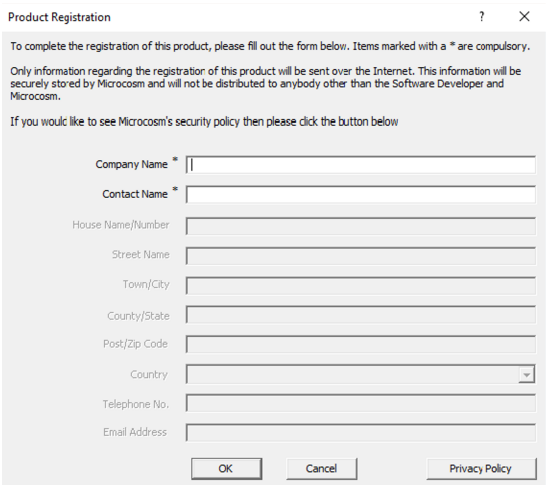


8. Click *OK*.

9. The protection system will connect to the website to check the settings defined for this product code and license. Click *OK* in the dialog.



10. The next dialog asks for the Product Registration data. Please fill out the required fields:



11. Once registration is complete, please proceed to the section *Enabling Hydronia Plugins in QGIS*.

2.3.2 Network Server Installation

Use this activation mode if you have received a network license key. If you received a standalone license key, use the activation procedure described in the previous section.
A network installation allows the use of your software on any number of machines on the same network, but limits the number of simultaneous users of your software.
The client machines will communicate with this server to carry out protection checks.
The Network Administrator will need a license key (configured to allow network installation) to activate the installation on the server. It is recommended that the Network Administrator does not reveal this license key to end-users, to avoid the potential confusion of the user trying to activate their copy of your software as a single user installation using that license key (this will not work).
The Network Administrator runs with its associated files (defined and distributed by Hydronia) in the network server.

1. To activate your software, you must open DIP shortcut on your desktop.

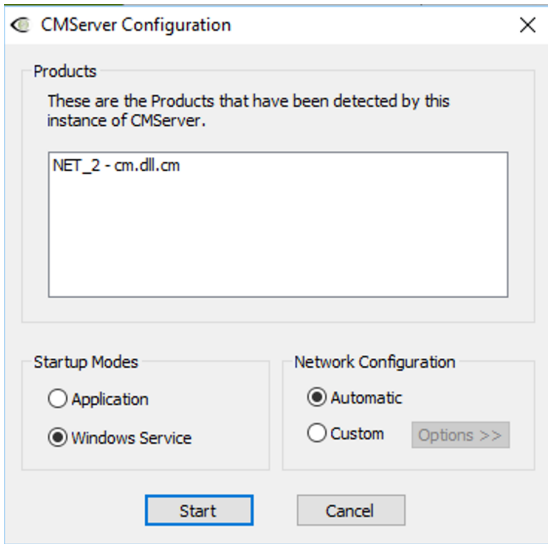
2. In the *Control Data* section on the left side, go to *Options* and select *License*.

3. You will be prompted to select one of three options:

1. Reactivate License
2. Install Network License Server
3. Check for Updates

4. Select *Install Network License Server*.

5. will display a configuration window.

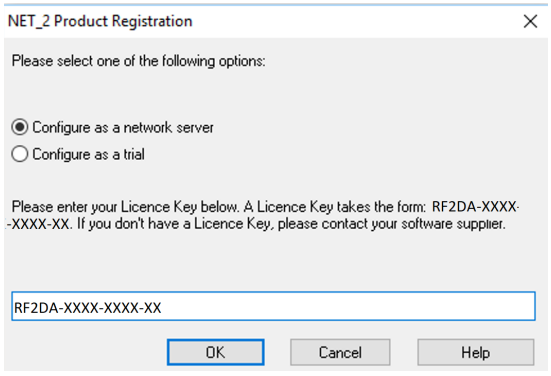


6. Choose *Startup Modes* as *Windows Service*.

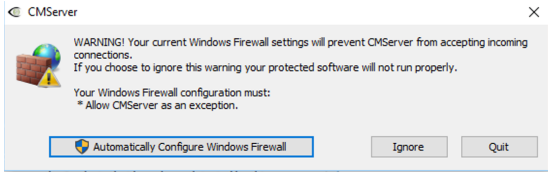
7. In *Network Configuration*, select *Automatic*.

8. Click *Start*.

9. After clicking *Start*, enter the License Key provided by Hydronia.

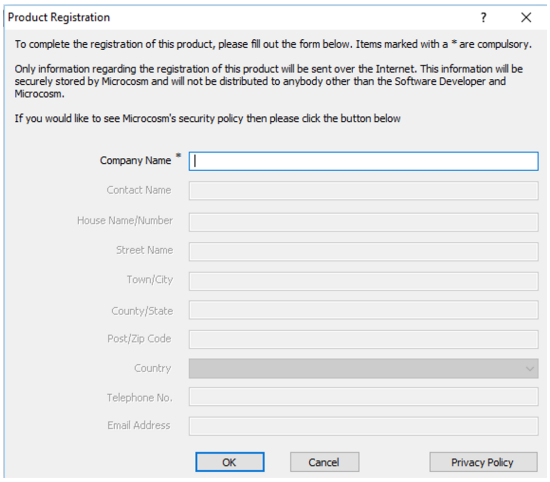


10. You will be prompted to add an exception to the firewall rules.

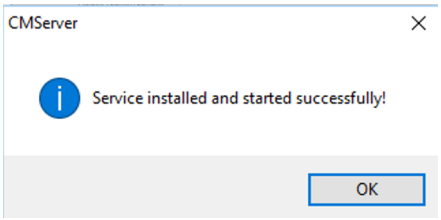


11. Select *Automatically Configure Windows Firewall*.

12. Next, input your details in the Product Registration window:



13. If successful, the following message will appear:



14. You may now install network clients on other machines.

2.3.3 Network Client Installation

Once the network license server is running, the OilFlow2D program and dependencies must be installed on each client workstation computer. Please refer to the *Software Installation* section. The steps below assume you have already run the installation program in the client computer and are ready to activate.

1. To activate your software, open DIP shortcut on your desktop.

2. In the *Control Data* section on the left side, go to *Options* and select *License*.

3. You will be prompted to select one of three options:

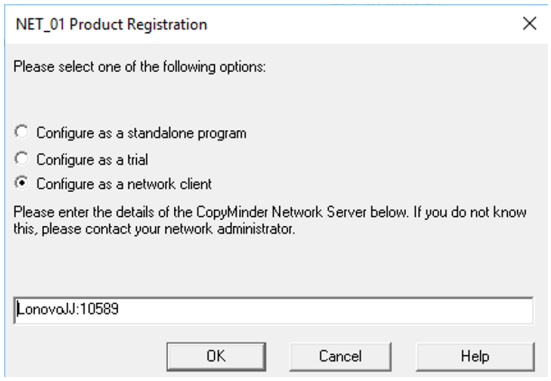
    1. Reactivate License
    2. Install Network License Server
    3. Check for Updates

4. Select *Reactivate License*

5. Select *Configure as a Network Client*.

> **Note**
>
> Note: In most cases the software will automatically detect the presence of the Network Server and the details will be filled in. If it does not fill this information in, input the IP address or computer name followed by.

6. The window shows the name of the network server (in this example):



7. Click *OK*.

8. Repeat this process for each OilFlow2D network client.

> **Note**
>
> Note: The software can be installed and configured as a network client on as many computers as desired in the network. In that case, the model will only run concurrently on as many computers as the number of OilFlow2D licenses that you have purchased.

2.4 Enabling Hydronia Plugins in QGIS

By default, the first time you run QGIS, the OilFlow2D plugin will not be enabled.

For version 10 and later, enable the unified Hydronia2DMS plugin in the *Installed* list. Hydronia2DMS provides the OilFlow2D-flavored tools for your installation. The product-specific plugin entry shown in the following section applies to pre-v10 or standalone installations.

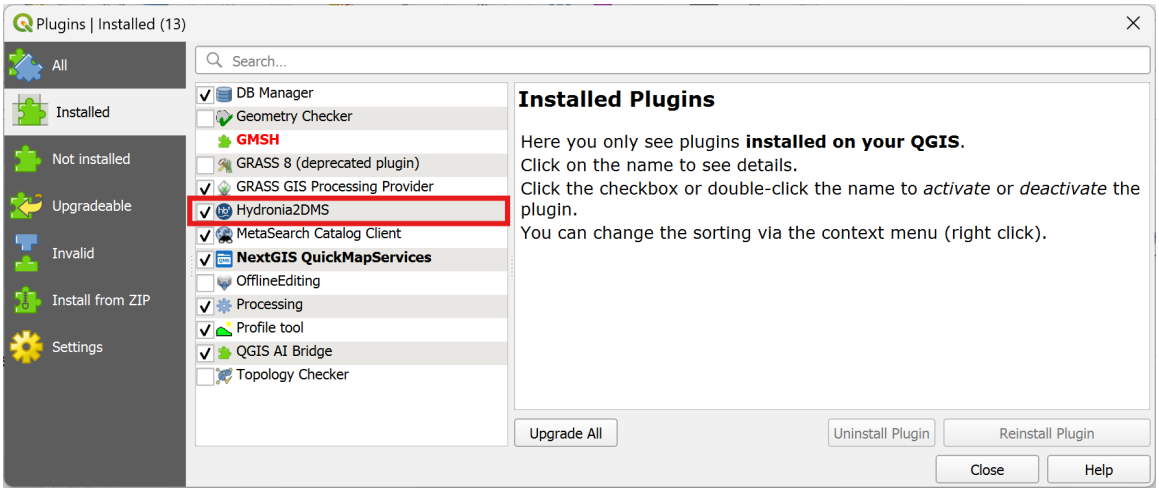
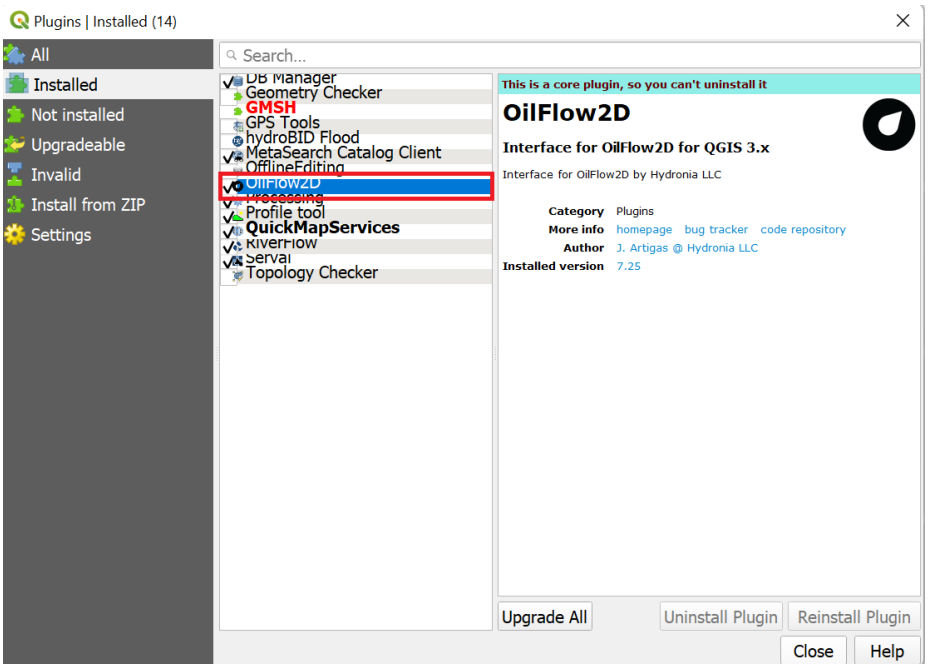


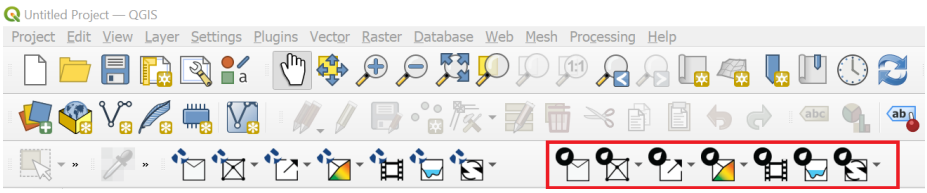
Figure 2.1 – QGIS Plugins window showing Hydronia2DMS installed for version 10 and later.

2.4.1 Enabling OilFlow2D Plugin (pre-v10 or standalone)

For OilFlow2D plugin, please enable it by selecting the *OilFlow2D* check box using the *Plugins | Manage and Install Plugins… menu* as shown.



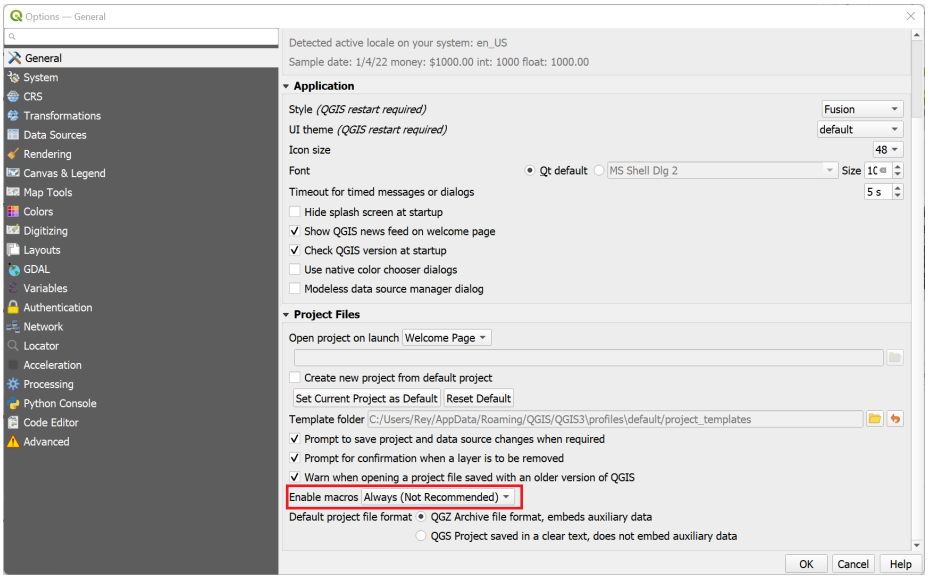
Then verify that the OilFlow2D plugin icons appear in the QGIS toolbar area:



2.4.2 Enabling Macros in QGIS

Successfully using the OilFlow2D plugin requires enabling the use of macros in QGIS. To do that, access the *Options* dialog by using the *Settings* | *Options…* menu, and in the *General* panel scroll down and select the *Always* option on the *Enable Macros* drop down list as indicated in the figure below.

> **Note**
>
> Please, do not pay attention to the (*Not Recommended*) warning on the option, since that is shown to warn about plugins of unknown origin, and that is not applicable to Hydronia Plugins.



2.5 Troubleshooting

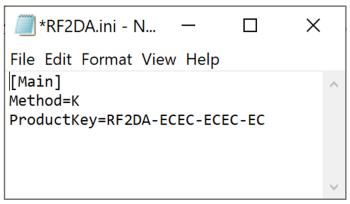
In this section we include solutions to some issues that may occur during the software installation. Bear in mind that you may always contact Hydronia support team at to report any error message or problem that you may encounter during installation.

2.5.1 Finding your License Key

If you ever have an issue related to your installation, you can find the license key in the following files depending on the type of installation that you are using:

- Standalone: C:\ProgramData\AVU\OF2DA.ini
- Network License Server: C:\Program Files\Hydronia\LicenseManager\OF2DA.ini

Open the file with Notepad or any other text editor, and your license key will be indicated following “”. In the following example the license key is:

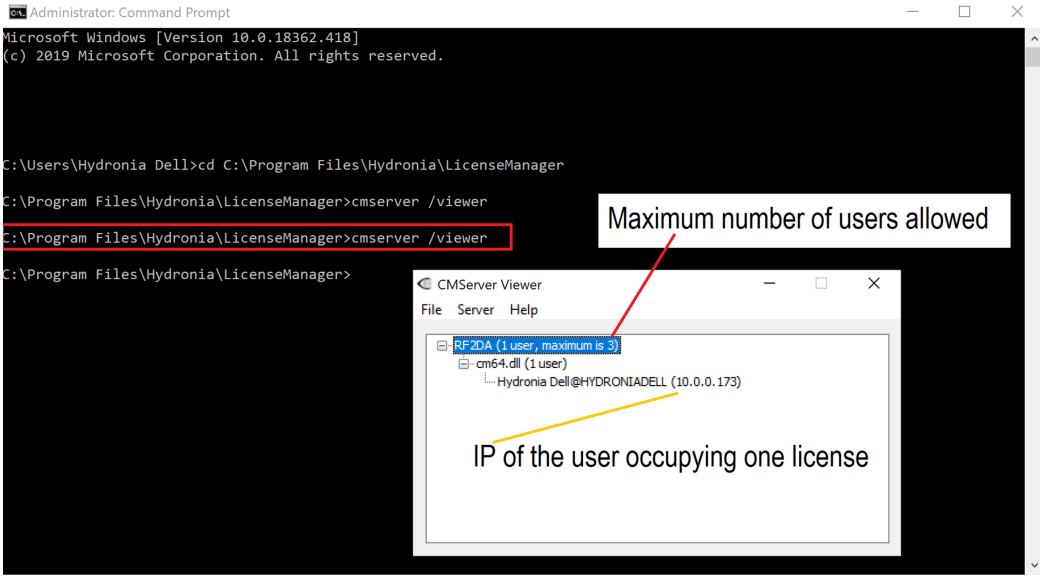


2.5.2 Find Who is Using the Software in a Network Installation

You can check how many users are occupying each license and know their computer ID using the viewer program that is installed on the folder.
In the , all the licenses in use are displayed, with the machine name or IP address and username.
If CMserver is running as a service, you can start the GUI for the viewer from the command line as follows:

1. In the Windows search box write and then press *Enter*
2. Enter and press *Enter*
3. Enter and press *Enter*

The information displayed in the window will be similar as indicated in the following figure



2.5.3 ERROR 641: “You have reached the limit on the maximum number of simultaneous users of this program.”

This error can appear when there are more models trying to run than the number of available licenses. However, sometimes the run was interrupted at a critical stage and the model executable remains in memory. This is interpreted by the protection as if there was more licenses running. To fix this issue, open the Windows task Manager and in the Process tab look for the model executable, select it by clicking on the file and click *End task*. That will remove the model from memory and terminate the idle run.

2.5.4 ERROR 659: “This program is configured for network installation only. It cannot be installed as a standalone system.”

This error is frequently a result of specifying a Product Key on a network client machine instead of the License Manager Host Name. For a network installation, when the model is first run on a client machine, you should select *Configure as a network client* and enter the Host Name of the computer with the Network License manager installed. To solve the issue, follow the activation instructions above.

2.5.5 ERROR 660: “This program is configured for standalone installation only. It cannot be installed as a network system”

This error is the result of having used a license key that is for network installations only with a license that has been configured for standalone use. To solve the issue, please install the Network License Server as instructed in section on page and use the key provided by Hydronia.

2.5.6 ERROR 739: “This program has been installed or copied too many times.”

This error is generated when the OilFlow2D program has been installed or re-activated more times than allowed by the protection program. It does not necessarily indicates improper use of the model. If you get this error please send an email to our support team at indicating the error and your license key. With that information we will reset the license server to release the license.

If you do not know your license key, contact Hydronia support.

2.6 OilFlow2D Documentation

Find OilFlow2D documentation including this manual in the documentation folder installed with the software. The installation also includes example projects, videos, and other useful resources.

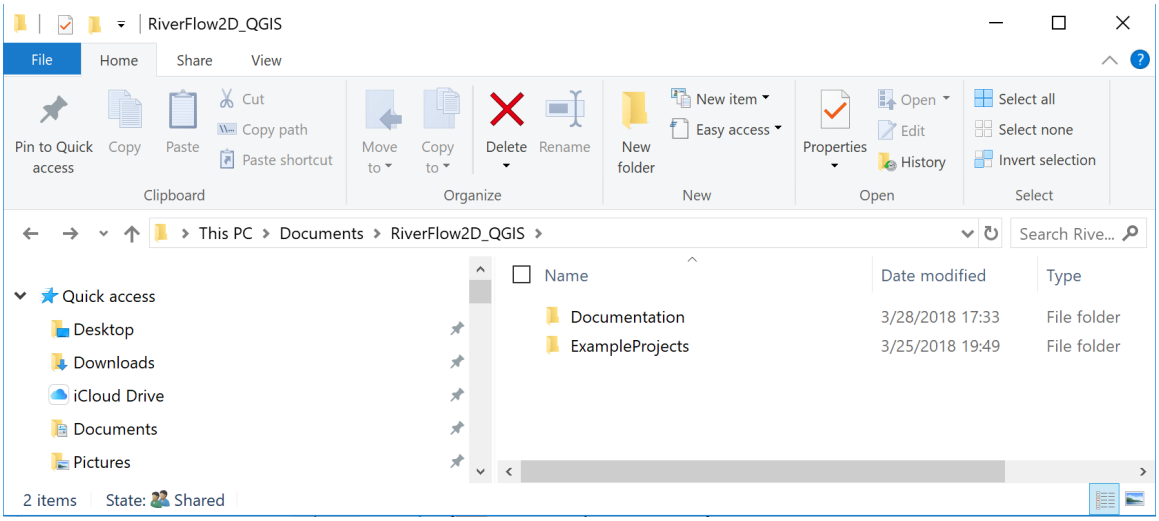


Figure 2.2 – Location of OilFlow2D Documentation folder.

2.7 OilFlow2D Technical Support

If you have any questions or require assistance using OilFlow2D, please send an email to our support team at:
`mailto:support@hydronia.com`. Please make sure you visit our web site `https://www.hydronia.com` regularly to find out about new products and news about the software. Visit `https://www.hydronia.com/software-updates` to download the latest software update and the Release Notes document containing a list of the recent fixes, changes, and updates.

2.8 OilFlow2D Tutorials

The best way to get acquainted with OilFlow2D capabilities is following the tutorials. The accompanying OilFlow2D tutorials help you get started with the model and several of the model components. Each tutorial includes a set of files that you can use to do each exercise.

3

Overview of OilFlow2D

When you create a new OilFlow2D project in QGIS, the plug-in creates a number of empty layers, each one with an specific purpose, and associated with particular components or modules. The standard set includes the layers depicted in Figure. Description of each layer is included in Table.

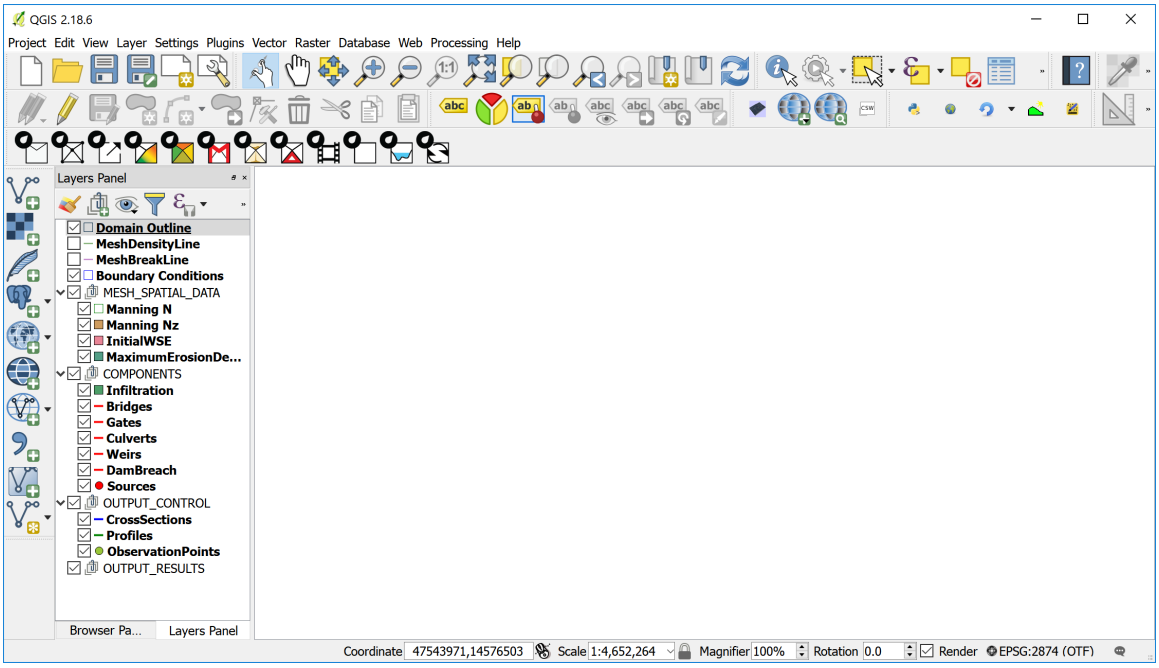


Figure 3.1 – Standard OilFlow2D layers created when using the New OilFlow2D Project command.

Content
Content

		Content
TriMesh	Polygon	Contains the mesh triangular cells. It is automatically created by the mesh generation program.
Domain Outline	Polygon	Container for the required external polygon that defines the extent of the modeling area. It can also include internal polygons that represent impermeable islands or other obstacles that will not contain cells. Each polygon has a *CellSize* attribute that controls the approximate triangle size desired for the generated mesh.
MultipleDemBoundaries	Polygon	It is used to enter polygons that define areas with different terrain elevation data sets. You can associate each polygon to a different raster layer containing a terrain elevation model.
MeshDensityLine	Line	It is used to enter polylines along which the mesh generation program will refine the mesh according to each polyline *CellSize* attribute. The lines do not force the mesh generator to create nodes along the lines. In this sense, they act as soft breaklines.
MeshBreakLine	Line	It is used to enter polylines along which the mesh generation program will refine the mesh according to each polyline *CellSize* attribute. The lines do force the mesh generator to create nodes along the lines. Therefore, they act as hard breaklines.
Boundary Conditions	Polygon	Container for polygons that define the model open boundaries, either inflow or outflow. All the boundary cells laying inside these polygons will be open boundary cells.
MESH_SPATIAL_DATA		
Multiple DEM Boundaries	Polygon	Container for polygons over which different elevation rasters (e.g. DEMs) will be used to interpolate elevations to the cells.
Manning N	Polygon	Defines areas of different Manning's n.
Manning Nz	Polygon	Accepts polygons associated to files that contain tables of Manning's n as a function of depth.
InitialWSE	Polygon	Container for areas of initial Water Surface Elevations (WSE).
COMPONENTS		
Infiltration	Polygon	Defines areas of different Infiltration parameters.
RainEvap	Polygon	Container for areas associated with a rainfall intensity and evaporation.
Wind	Polygon	Defines areas associated with a wind velocity time series that will be used in the model to calculate the wind stress on the water surface.

		Content
Bridges	Line	Includes polylines defining bridges. Each entity will have specific data that characterize the bridge cross section. Also the lines will act as hard breaklines.
Gates	Line	Includes polylines defining gates. Each entity will have specific data that characterizes the gate including gate aperture table. Also each line will act as a hard breakline.
Culverts	Line	Contains lines that connect two points in the modeling area with culverts. The model will calculate the culvert discharge depending on the given data, and transfers discharge from culvert the inlet cell to the culvert outlet cell.
Weirs	Line	Container for polylines defining weirs. Each entity will have specific data that characterizes the weir. The line will act as a hard breakline.
DamBreach	Line	Contains polylines that represent dam or levees in plan. They allow the model to calculate the discharge through levee or dam breaches.
Sources	Point	Container for point sources or sinks. Source data includes a time series of discharge vs. time. When using the Pollutant Transport PL Module, the data must include concentrations for each pollutant in addition to the discharge. Sinks are defined by negative discharges.
OilSpills	Point	Container for spill locations within the mesh. Each spill point needs data to define the spill volume, and other parameters required to simulate the oil trajectory and behavior based on existing results from a hydrodynamic run.
Piers	Point	Container for bridge pier locations. Piers are used to enter data that will allow the model to compute scour around bridge piers.
Abutments	Line	Container for bridge abutment locations. Abutments are used to enter data that will allow the model to compute scour around bridge abutments.
StormDrain	Point	Container to indicate flow exchange points between the surface water with the storm drain network of a EPA-SWMM model.
OUTPUT_CONTROL		
CrossSections	Line	Container for lines that define cross sections where the model will write results including discharge for each report interval.

		Content
Profiles	Line	Defines profiles where the model will write in text files results for each report interval.
Observation Points	Point	Container for locations where the model will write results each report interval.
OUTPUT_RESULTS Group that	include la	yers with model results that will be incorporated by the program when creating specific graphics with model results.
: OilFlow2D layers.		

4

Mesh Generation in OilFlow2D

The basis of OilFlow2D computational engine, OilFlow2D is the flexible mesh, also called unstructured mesh or Triangular Irregular Network (TIN). The mesh is formed by triangles most often of different size, and is called flexible because it can be adapted to irregular topography, boundaries, structures, or any obstacle that may exist on the modeling area (see Figure).

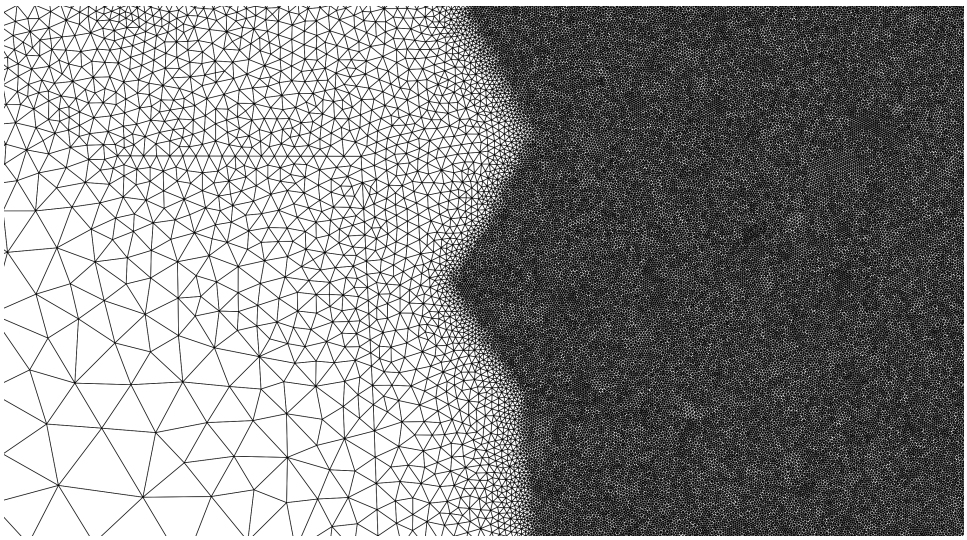


Figure 4.1 – Typical OilFlow2D flexible mesh.

The fundamental computational unit in the OilFlow2D model is the triangular cell, where velocities, depth, and other variables are computed.

There are several tools in OilFlow2D that can be used to control the mesh generation. These tools make use of spatial objects such as polylines and polygons, and parameters that you can enter in the *Domain Outline*, *MeshDensityLine*, *MeshDensityPolygon* and *MeshBreakLine* layers.

4.1 Cell-size control using the Domain outline

The *Domain Outline* is a key layer that defines the mesh limits and the extent of the modeling area. It accepts polygons, and needs to contain at least one polygon. It can also include internal polygons that represent impermeable islands or other obstacles. The internal polygons will not contain any cells.

For each polygon in the *Domain Outline* layer you need to enter a *CellSize* attribute that controls the approximate triangle size desired for the generated mesh (see Figure).

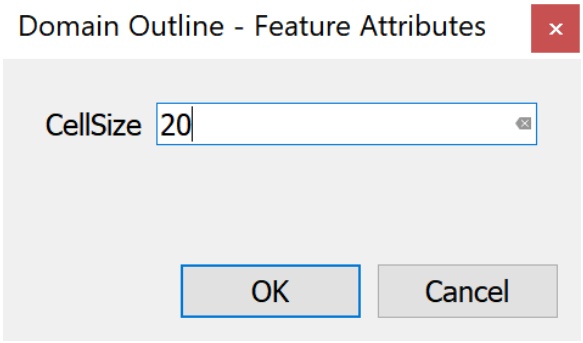


Figure 4.2 – *CellSize* dialog.

Figure shows a mesh with one hole. The mesh is defined by an external polygon with *CellSize* equal to 50 ft, and the internal polygon has a *CellSize* of 10 ft. Note that the resulting mesh has smaller triangles around the internal polygon and larger triangles close to the boundary.

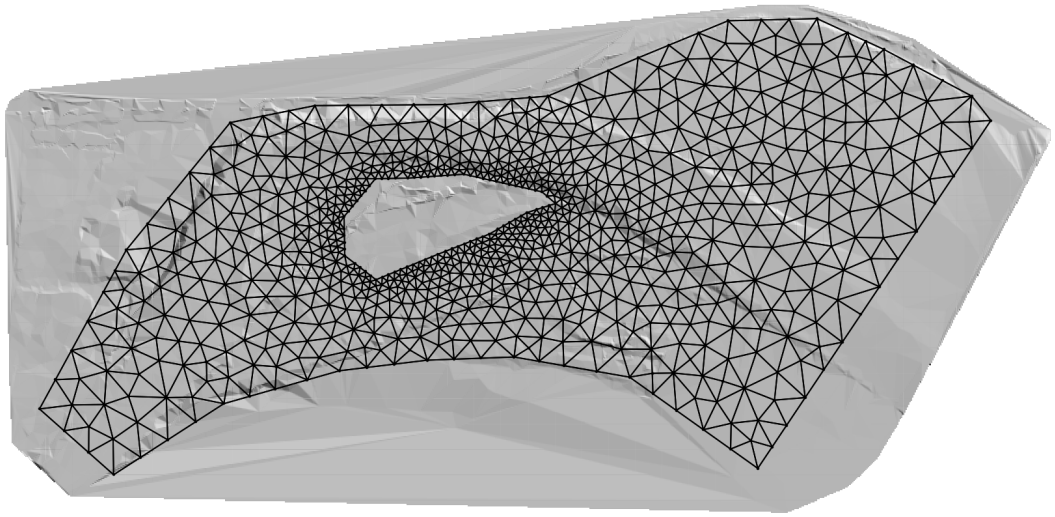


Figure 4.3 – Mesh generated based on an external polygon with *CellSize* = 50ft , and an internal polygon with *CellSize* = 10ft, both entered on the *Domain Outline* layer.

4.2 Cell-size control using Polylines in the MeshDensityLine Layer

The *MeshDensityLine* layer is used to enter polylines over which the mesh generation program will refine the mesh according to each polyline *CellSize* attribute. The polylines will not force the mesh generator to create nodes along the lines. In this sense, they act as soft breaklines (see Figure).

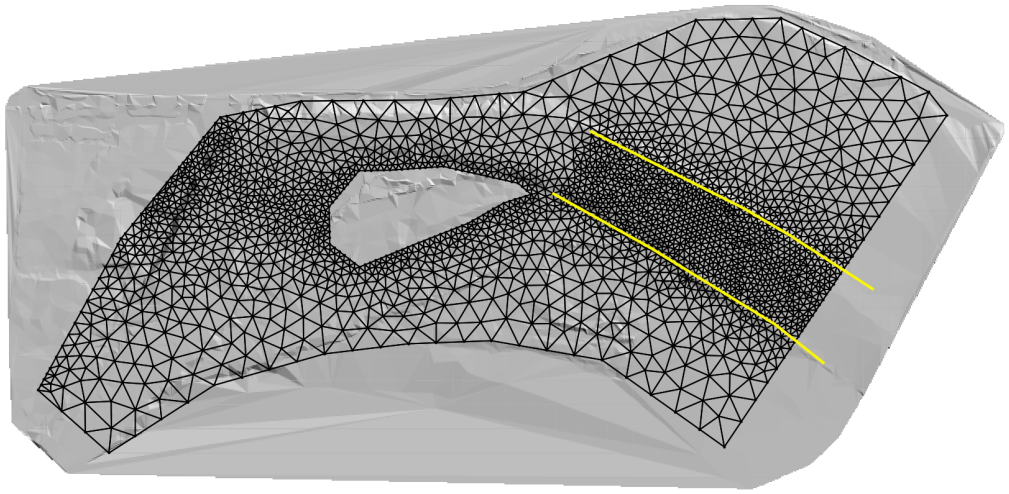


Figure 4.4 – Mesh generated based on the polygons of Figure adding the two polylines on the *MeshDensityLine* layer.

4.3 Cell-size control using Polygons in the MeshDensityPolygon Layer

The *MeshDensityPolygon* layer is used to enter polygons within which the mesh generation program will refine the mesh according to each polygon *CellSize* attribute. The polygon outlines will not force the mesh generator to create nodes along the polygon limits (see Figure).

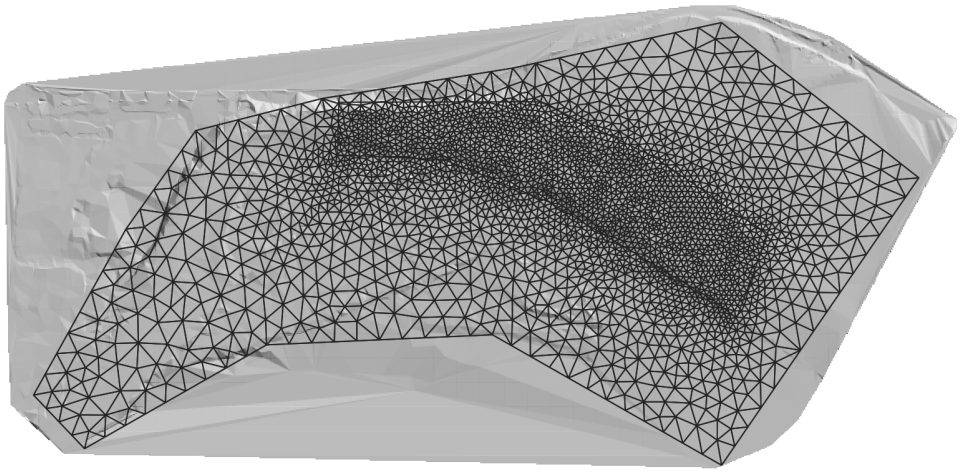


Figure 4.5 – Mesh generated based on the polygon on the *MeshDensityPolygon* layer.

4.4 Cell-size control using Polylines in the MeshBreakLine Layer

The *MeshBreakLine* layer is used to enter polylines along which the mesh generation program will refine the mesh according to each polyline *CellSize* attribute, similarly as in the *MeshDensityLine* layer, but in this case, the lines will force the mesh generator to create nodes along the lines. Therefore, they act as hard breaklines (see Figure).

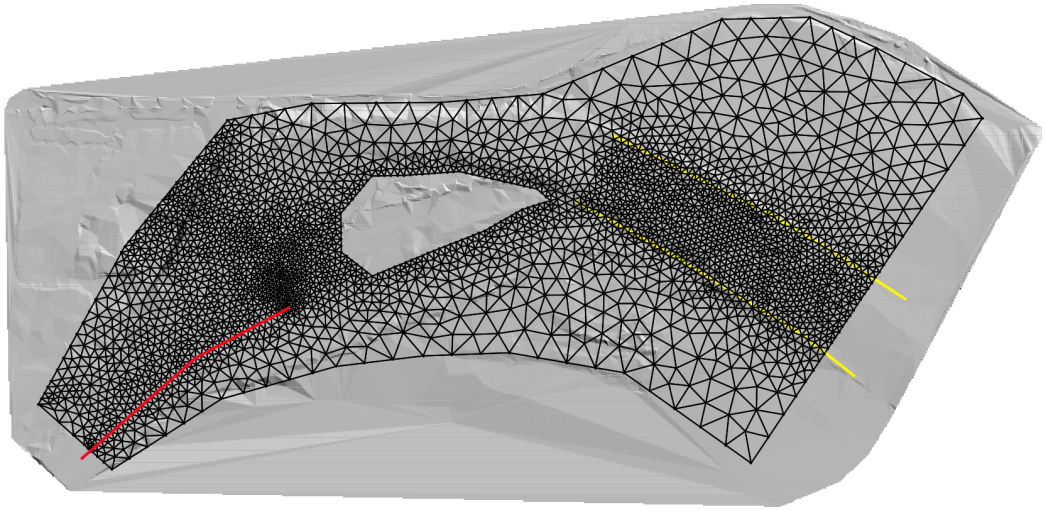


Figure 4.6 – Mesh generated based on the polygons and polylines of Figure adding the one polylines on the *MeshBreakLine* layer. Note how unlike in the MeshDensityLine layer, the polylines entered in the *MeshBreakLine* layer force the mesh to have nodes along the polylines.

In addition to the control offered by the spatial objects entered in the *Domain Outline*, *MeshDensity-*

Line, *MeshDensityPolygon*, and *MeshBreakLine* layers, other layers can be used to adjust the mesh alignment and resolution. For instance, the Bridges, Gates, and Weirs components are entered as polylines on the respective layers and all of them have a *CellSize* attribute that have the same effect as as the mesh breaklines.

4.5 Boundary Conditions

Data to impose open boundary conditions in OilFlow2D should be entered in the *Boundary Conditions* layer. This layer accepts only polygons. Lines or points are not allowed. To enter a polygon, first select the layer by clicking Boundary Conditions on the QGIS layers panel

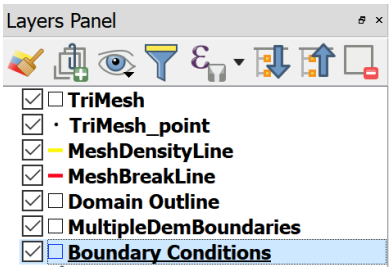


Figure 4.7 – QGIS Layer Panel showing the *Boundary Conditions* layer selected.

Then click on Toggle Editing (pencil), and on the Add Feature (polygon) as shown

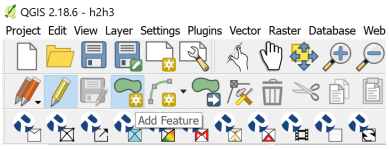


Figure 4.8 – Select Toggle Editing (pencil) and Add Feature.

Using the mouse, click vertices until you create a polygon the covers the area where you want to define as an Open Boundary

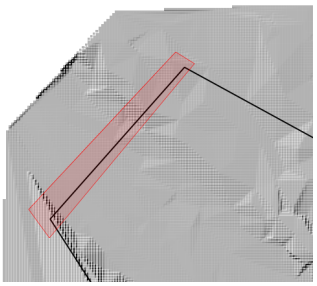


Figure 4.9 – Inflow Boundary Condition Polygon.

To complete entering the polygon, right-click and use the dialog to set the open boundary as Inflow, Discharge vs. time. The data will be written to the associated boundary-condition file.

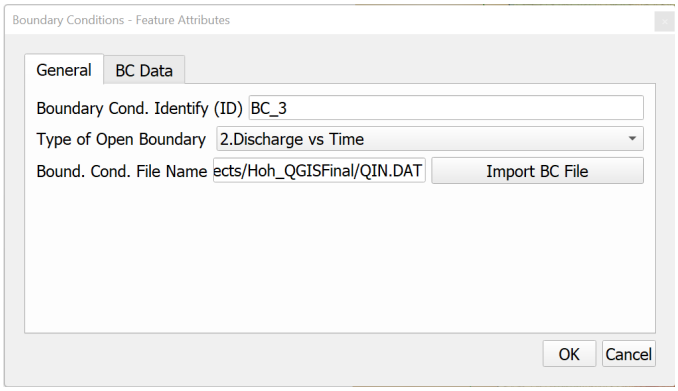


Figure 4.10 – Boundary Condition Dialog.

To complete the data, select the BC Data panel and enter the hydrograph as shown.

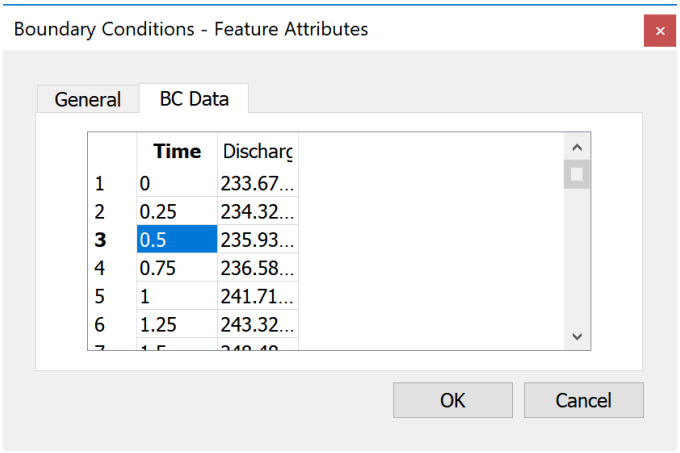


Figure 4.11 – Boundary Condition BC Data panel.

All nodes on the mesh boundary that lie inside the polygon will be considered open boundary nodes.

You can define as many inflow and outflow boundaries as needed. All the boundary not contained within the BC polygons will be considered as closed boundaries and no flow will be allowed to cross it.

4.6 Mesh Spatial Data

4.6.1 Mannings' n

To assign spatially varied Manning's n coefficients in OilFlow2D you enter polygons in the Manning N layer. This layer accepts only polygons. Lines or points are not allowed. To enter a polygon, first select the layer by clicking Manning N on the QGIS layers panel

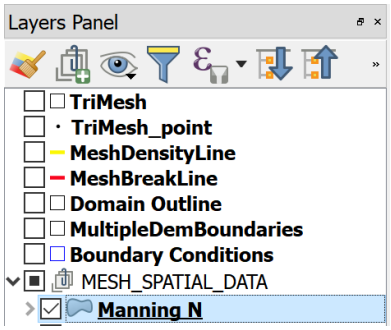


Figure 4.12 – QGIS Layer Panel showing the Manning N layer selected.

Then click on Toggle Editing (pencil), and on the Add Feature (polygon) as shown

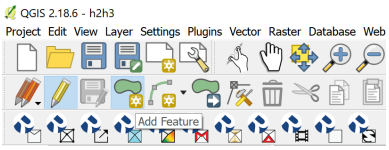


Figure 4.13 – Select Toggle Editing (pencil) and Add Feature.

Using the mouse, click vertices until you create a polygon the covers the area where you want to set an specific Mannings n value
To complete entering the polygon, right-click and use the dialog to set the open boundary as Inflow, Discharge vs. time. The data will be written to the associated boundary-condition file.

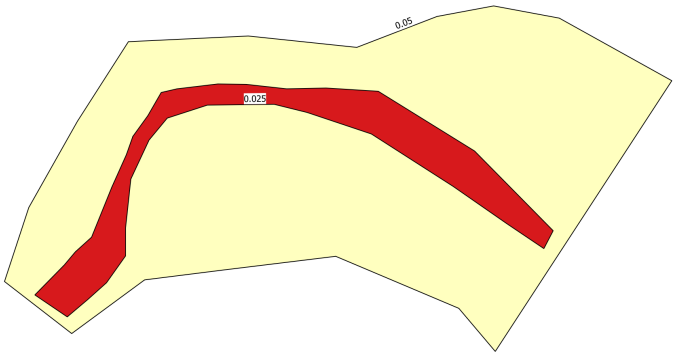
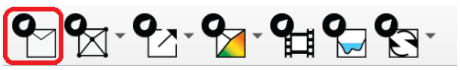


Figure 4.14 – Manning's n polygon.

To complete the data, select the *BC Data* panel and enter the hydrograph as shown. All cells on the Manning's n polygon will be assigned the n value corresponding to that polygon on the mesh boundary that lie inside the polygon will be considered open boundary nodes.

4.7 OilFlow2D Toolbar Functions

4.7.1 New OilFlow2D Project



This icon is used to create a new project template from scratch. There are three things you need to do in the dialog to complete creating a new project:

1. Select the component or mesh layers you want to create.
2. Select the Coordinate Reference System or *Projection*.
3. Enter the working *Project Directory*.

Figure shows the *Create New OilFlow2D Project Dialog*. Note that do not need to select all the available layers, but just the ones that you will be using initially in your project. You can always add more layers later using the *New Template Layer* command in the OilFlow2D *Tools* icon described below.

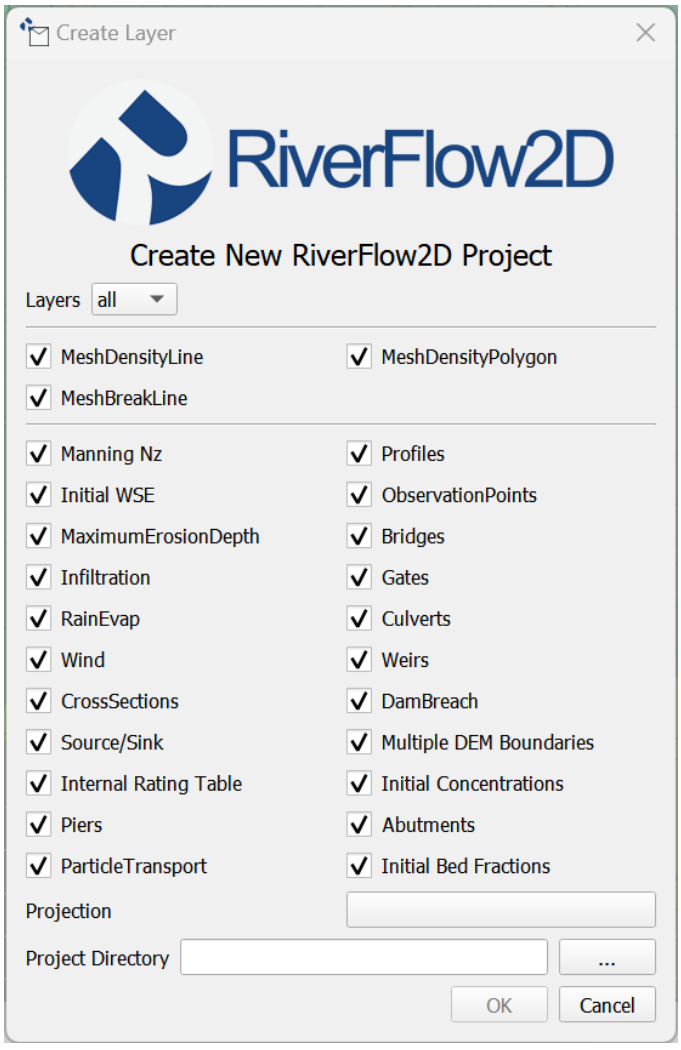
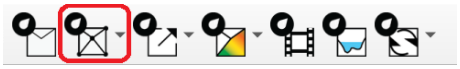


Figure 4.15 – New Project Dialog.

4.7.2 Generate TriMesh



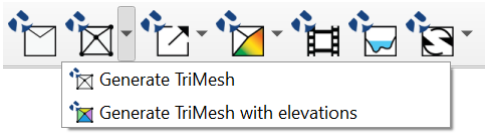


Figure 4.16 – Mesh Generation Options.

4.7.3 Export Files for OilFlow2D



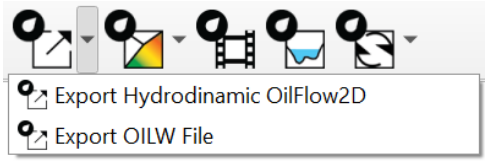


Figure 4.17 – Export Files Options.

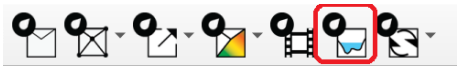
4.7.4 Maps



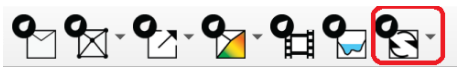
4.7.5 Animations



4.7.6 Cross Sections



4.7.7 Tools



4.7.7.1 Landslide Tool

The Landslide tool allow representing the initial volume of material that could be mobilized during a landslide. The tool requires creating one or more polygons in the Landslide layer and indicating the depth or volume of the material (see Figure) and the concentration of the sediment classes that form this material. With this information, the model will assume that under the given polygon the material down to the given depth, could flow depending on the other model conditions.

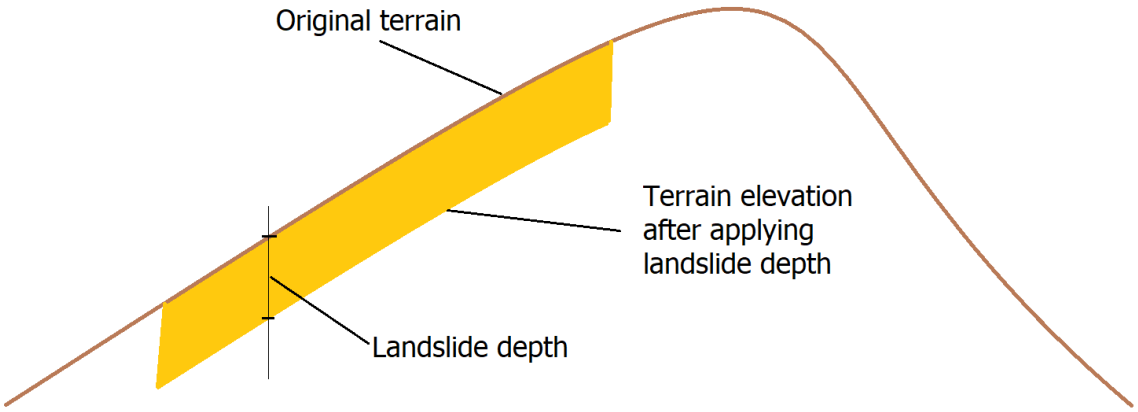


Figure 4.18 – Schematic view of the landslide.

To enter the data, first create the *LandSide* layer.

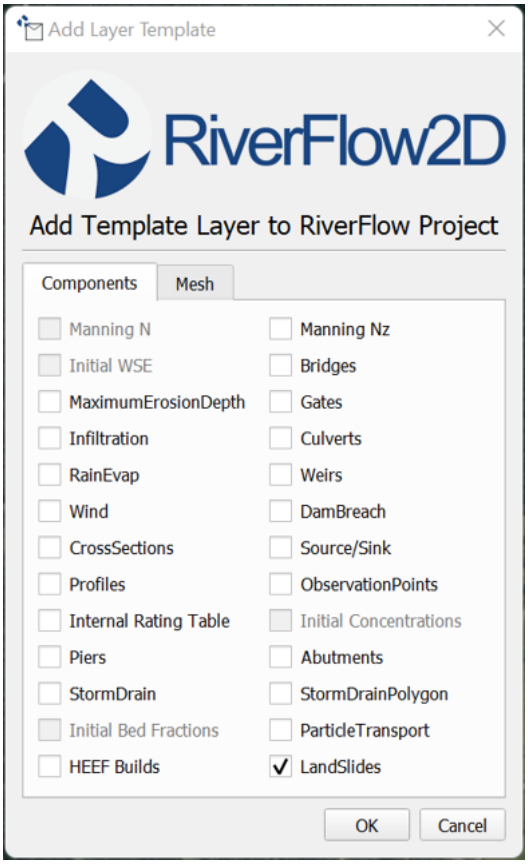


Figure 4.19 – Add LandSlides layer.

Then draw a polygon and enter the depth or volume and sediment class concentrations as indicated in the dialogs :

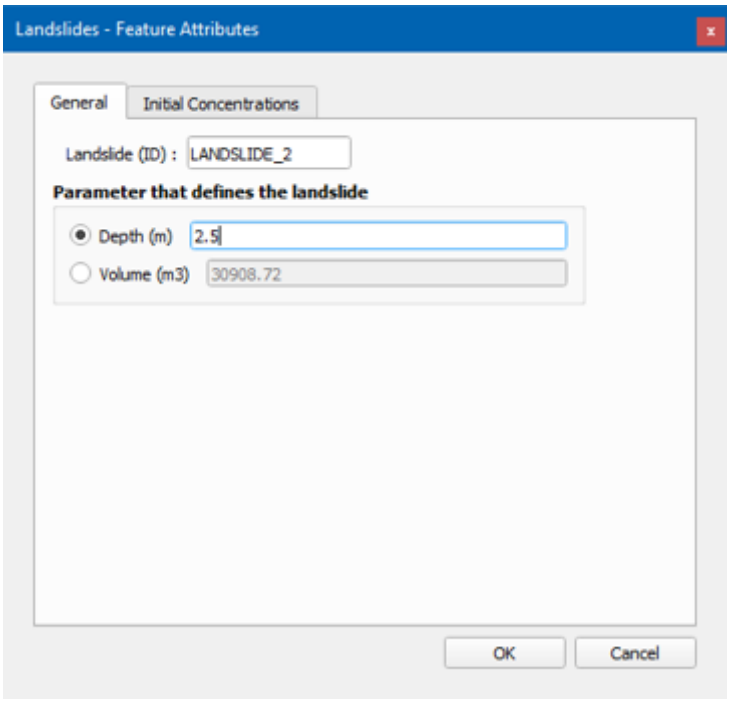


Figure 4.20 – Depth or volume landslide attributes.

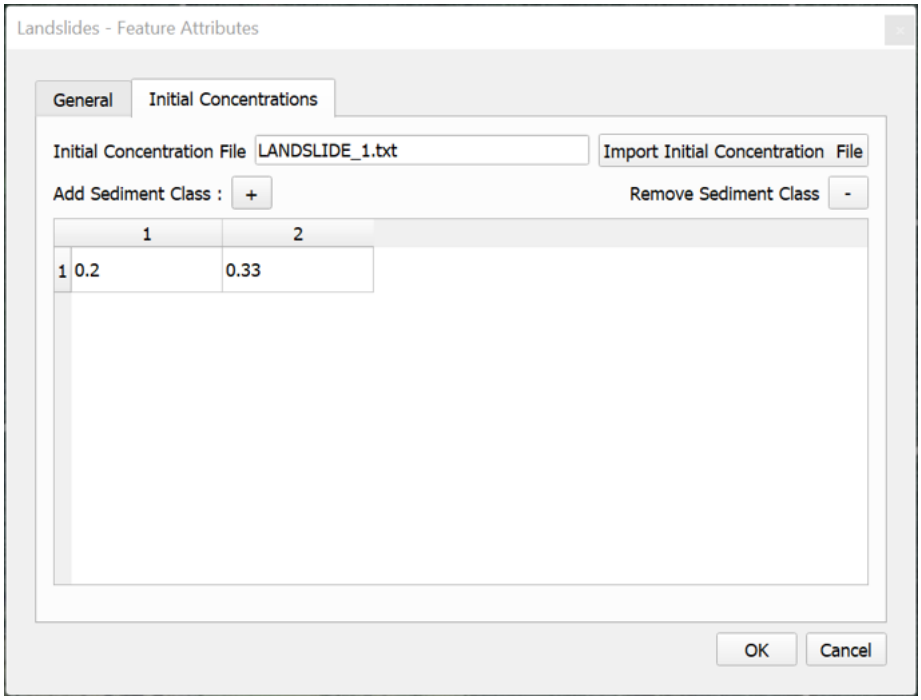


Figure 4.21 – Concentration for all classes forming the landslide material.

When the user provides the depth, all the cells contained in the landslide polygons will changes as follows:

- The cell elevation will be equal to the original terrain elevation minus the given depth (see Figure)
- The initial water or material elevation will be equal to the original terrain elevation.

When the user provides the volume, the model computes an average area of the landslide as depth = Volume/Polygon area, and all the cells contained in the landslide polygons will change as indicated above. All of the terrain changes are transferred to the .FED file that contains the mesh data and cell elevations.
Note that the *Read initial water elevs. from FED file* needs to be selected in the DIP Control Data panel.

5

Oil Spill on Land 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.

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 quiescent water, leading to the notion of well-balanced schemes or C property [, , ,]. 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),]. 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 OilFlow2D and the information can be expanded in the references.

5.1 Assumptions of the Viscous Flow Model

1. OilFlow2D 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.
4. The model can consider heat transfer to calculate the oil temperature as it flows overland, and considers the density, viscosity and yield stress variation in time and space.

5.2 Flow equations considering prescribed temperature variations

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

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

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 **u** along the (x, y) coordinates respectively. The flux vectors are given by:

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

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

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

where the bed slopes of the bottom level z_b are

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

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

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

with n the roughness coefficient.
Using this option, the model can consider the variation of temperature and its effect on density and viscosity, but the oil temperature is assumed to be prescribed by the user and it does not depend on environmental chances during the simulation.

5.3 Equations considering heat transfer

This section presents the equations that OilFlow2D solves to consider the heat transfer from the oil to the atmosphere and soil as it flows overland. The model also determines how the oil density, viscosity and yield stress change as a function of the temperature during the simulation.
The continuity equation for the oil mass is written as

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial}{\partial x}(\rho h u) + \frac{\partial}{\partial y}(\rho h v) = 0$$

and the conservation laws of the bulk linear momentum along the $x-$ and $y-$ axes are be expressed as

$$\frac{\partial(\rho hu)}{\partial t} + \frac{\partial}{\partial x}(\rho hu^2 + \frac{1}{2}g_n\rho h^2) + \frac{\partial}{\partial y}(\rho huv) = -g_n\rho h\frac{\partial z_b}{\partial x} - \tau_{bx}$$
$$\frac{\partial(\rho hv)}{\partial t} + \frac{\partial}{\partial x}(\rho huv) + \frac{\partial}{\partial y}(\rho hv^2 + \frac{1}{2}g_n\rho h^2) = -g_n\rho h\frac{\partial z_b}{\partial x} - \tau_{by}$$

where ρ ois the depth-integrated bulk density [kg/m^3 or lb/ft^3], h the vertical flow depth [m or ft] and $(u,\, v)$ the components of the depth-integrated flow velocity vector **u** [m/s or ft/s], z_b the bed elevation [m or ft], $(\tau_{bx},\, \tau_{by})$ the components of the depth-integrated basal resistance vector τ_b.
The energy equation for temperature is written non conservative form as follows:

$$\frac{\partial T}{\partial t} + u\frac{\partial}{\partial x}T + v\frac{\partial}{\partial y}T = \mathbf{S}_T$$

where $\mathbf{S}_T$ source term considers the heat transfer of the oil with the environment, as a function of the oil depth, h and other factors.
To close the model, the following equation is used to determine the fluid density ρ [kg/m^3 or lb/ft^3]

$$\rho = \rho_0 + \Lambda(T - T_0)$$

where ρ_0 [kg/m^3 or lb/ft^3] is the oil reference density at temperature T_0 [°K] and Λ is an experimental parameter.

5.3.1 Fluid properties and friction laws

Besides the density, the flow temperature also affects the viscosity and yield stress, generating changes in friction stresses between the flow and terrain.
The yield stress [Pa or lb/in^2] dependence on temperature can be entered as a table of experimental stress for each temperature or using regression formulas as the following one

$$Ys = 10^{(AysT^2 + BysT - Cys)},$$

where Ays, Bys and Cys are flow parameters coming from oil characterization.
The oil viscosity, μ [Pa·s or lb·s/in^2], is governed by Andrade formulation

$$\mu = Av\; e^{(Bv/T)}$$

where Av and Bv are flow parameters coming from oil characterization and T is in °K.
The model provides also the option to enter tables that represent the variation of viscosity, yield stress and density as a function of temperature.

5.3.2 Temperature Source Term and Heat Transfer Mechanisms

The temperature source term is computed as

$$\mathbf{S}_T = \frac{Q}{\rho C_p h}$$

where C_p is specific heat [J/kg°C or BTU/lb°F], and Q total heat flux [W/m^2 or BTU/ft^2] that quantifies the heat exchange between the oil and the environment.

OilFlow2D considers the following formulation to represent the heat transfer between the oil and the environment.

- Ambient radiation (oil→environment):
- Received solar thermal radiation (sun→oil): calculated as a function of: extra planetary radiation (R0), which is attenuated by atmospheric transmission (at), cloudiness (ac), shading (cs) and reflection (cr).
- Convection transfer between oil and ground
- Convection transfer between oil and air

The heat transfer model considers air convection, incident radiation and emitted radiation. The heat transfer mechanisms are acting as parallel heat exchanges between the fluid surface and the environment that can be expressed as

$$\mathcal{S}_{\mathcal{T}} = \frac{\dot{Q}}{\rho h C_p} = \frac{\dot{Q}_{rad} + \dot{Q}_{conv} + \dot{Q}_{em}}{\rho h C_p}$$

The radiation heat transfer between the oil and air is computed by following the Stefan-Boltzmann equation

$$\dot{Q}_{em} = \varepsilon\sigma \left(T^4 - T_{air}^4\right)$$

where ε is the oil surface emissivity (assumed as 0.55), $\sigma = 5.67 \times 10^{-8}$ W/(m^{2}K^{-4}) is the Stefan-Boltzmann constant and T_{air} is the air temperature.
The air convection is computed as a function of the difference between the fluid and the air temperatures, T and T_{air} [K], and a convection coefficient, h_c,

$$\dot{Q}_{conv} = h_c \left(T - T_{air}\right)$$

in terms of the convective heat transfer coefficient h_c [Wm^{-2}K] that introduces the effect of air movement. It can be related to the Nusselt number, in order to establish how relevant is the convection heat transfer compared to the heat conduction:

$$Nu = \frac{h_c L}{k} \left(T - T_{air}\right)$$

where L [m] is a characteristic length and k [W m/K] the thermal conductivity.

In the model h_c is calculated as function of the wind velocity

$$h_c = 8.55 + 2.56 \cdot U_w$$

Finally, incident radiation, $\dot{Q}_{rad}$, can be obtained from measured data and is required in the model as an input parameter.

5.4 Oil Retention

Oil retention is a phenomenon in which the velocity of the oil phase is significantly reduced due to its interaction with the soil or vegetation. This retention occurs through a combination of adsorption, capillary trapping in pore spaces, and increased viscous resistance, leading to reduced oil mobility and extended residence time within the subsurface. In this context, the focus is on the viscous resistance between the oil flow and the soil. To model this process, the Simple Shear Infinite Landslide approach is used ,. In this model, the shear stress is balanced by the gravitational force component in the horizontal flow direction (see Figure). This shear stress reduces the momentum in the oil flow direction. However, this shear stress only affects the thin layer closest to the bottom. Therefore, an oil depth, $h_{\text{retention}}$, must be defined as the thickness of this layer. Therefore, the shear stress caused by this phenomenon, $\tau_{\text{retention}}$, at the soil-oil interface can be expressed as:

$$\tau_{\text{retention}} = \rho g h_{\text{retention}} sin(\theta)$$

where μ is the oil viscosity, ρ is the oil density, g is gravitational acceleration, θ is the bottom slope angle (see Figure). Using this expression, the following cases may occur:

- **Case with slope ($S_0 = tg(\theta)$)**: In this case, the shear stress is defined by the expression , and the velocity is considered zero when the oil depth is $h \leq h_{\text{retention}}$.
- **Case without slope ($S_0 = 0,\ \theta = 0$)**: In this case, using the expression , $\tau_{\text{retention}} = 0$. Therefore, oil retention is caused by imposing a oil flow velocity of zero when $h \leq h_{\text{retention}}$.

It is important to note that the oil retention depth, $h_{\text{retention}}$, represents the minimal depth for oil flow. Consequently, the velocity of cells with an oil depth smaller than $h_{\text{retention}}$ will be zero. For this reason, the values for $h_{\text{retention}}$ should be on the order of millimeters.

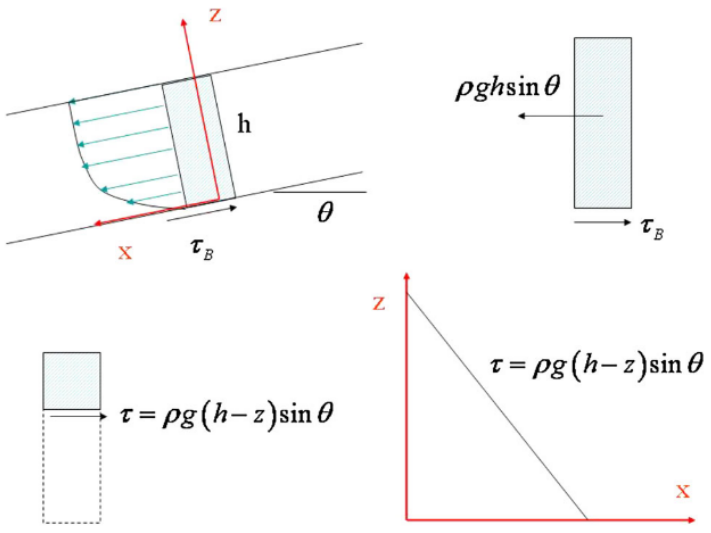


Figure 5.1 – The Shear Infinite Landslide approach scheme.

5.5 Pipeline Break Spill Hydrograph Equations

The OilFlow2D QGIS Oil Pipeline Break Model prepares a multiple-sources file and one discharge hydrograph for each generated pipeline break point. The calculation is performed before the OilFlow2D simulation; the resulting hydrograph files are then read by the model as source time series for the overland oil spill simulation. The following equations summarize the calculation used by the pipeline module.

For a pipe diameter D_p and leak diameter D_l, the pipe and leak areas are

$$A_p = \frac{\pi D_p^2}{4}$$

$$A_l = \frac{\pi D_l^2}{4}$$

The gravitational acceleration is $g = 9.80665$ m/s^2 when the project is in metric units, or $g = 32.174$ ft/s^2 when the project is in English units. The leak loss coefficient associated with the user-entered discharge coefficient C_d is

$$K_l = \frac{1}{C_d^2} - 1$$

5.5.1 Initial Steady Pipeline Head

The module first computes a steady energy head along the pipeline from the initial flow rate Q_0 and the initial pressure head entered at the pipe end. If z_n is the elevation at the downstream pipe end and H_{end} is the entered pressure head at that end, the downstream energy head is

$$h_n = z_n + H_{end}$$

Moving upstream by segment, the energy head is updated as

$$h_i = h_{i+1} + \frac{f_i \Delta x_i}{2 g D_i A_i^2} Q_0^2$$

where h_i is the energy head at node i, Δx_i is the segment length, D_i is the pipe diameter, A_i is the pipe area, and f_i is the Darcy-Weisbach friction factor. The Reynolds number used for each segment is

$$Re_i = \frac{|Q_0| D_i}{A_i \nu}$$

where ν is the kinematic viscosity. For laminar flow the friction factor is

$$f_i = \frac{64}{Re_i}$$

For turbulent flow, the module solves the Colebrook relation by Newton-Raphson iteration:

$$\frac{1}{\sqrt{f_i}} = -0.86 \ln\left(\frac{e_i}{3.71 D_i} + \frac{2.51}{Re_i \sqrt{f_i}}\right)$$

where e_i is the pipe roughness. If the dialog option to calculate the friction coefficient from rugosity is enabled, this friction calculation is also used by the spill-drainage equations. Otherwise, the user-entered friction coefficient is used for the spill-drainage step.

5.5.2 Upstream Leak Flow

For each break, the pressure head used at the break is

$$h_p = h_b - z_b$$

where h_b is the computed energy head at the break and z_b is the break elevation. The upstream leak discharge at each time step is calculated with the orifice equation

$$Q_l = C_d A_l \sqrt{2gh_p}$$

The calculation interval is Δt. The inflow from the pipeline upstream of the break is initially Q_0. If t_s is the valve-closing start time and t_c is the valve-closing duration, the module uses

$$Q_{in}(t) = \begin{cases} Q_0, & t \le t_s \\ Q_0\left(1 - \frac{t-t_s}{t_c}\right), & t_s < t \le t_s + t_c \\ 0, & t > t_s + t_c \end{cases}$$

At each time step the leaked volume, incoming volume, and net outgoing volume are

$$V_l = Q_l \Delta t$$

$$V_{in} = Q_{in} \Delta t$$

$$V_{net} = V_l - V_{in}$$

The available upstream pipe volume V_a is reduced by the net outgoing volume. While available volume remains, the pressure head is reduced in proportion to the remaining volume:

$$V_a^{new} = V_a - V_{net}$$

$$h_p^{new} = h_p \frac{V_a^{new}}{V_a}$$

When the available volume is depleted, the upstream pressure head is set to zero and the upstream contribution ends.

5.5.3 Downstream Gravity Drainage

The downstream portion of the pipe is assumed to drain by gravity from the pipe volume downstream of the break. For the contributing downstream length L_a, the available volume is

$$V_a = L_a A_p$$

The local pipe slope used to lower the oil surface during drainage is

$$m = \frac{|z_s - z_b|}{L_a}$$

where z_s is the elevation at the end of the contributing segment. At each time step the module solves the quadratic form

$$AQ_l^2 + BQ_l + C = 0$$

with

$$A = \frac{8}{g\pi^2}\left(\frac{1 + K_l}{D_l^4} - \frac{1}{D_p^4}\right)$$

$$B = fL_a$$

$$C = \frac{P_l - P_s}{\rho g} + z_b - z_s$$

where ρ is oil density, P_l is the pressure at the leak, and P_s is the pressure at the upstream end of the contributing segment. For downstream gravity drainage the module uses zero gauge pressure, so the pressure term is normally zero. The positive-root discharge is

$$Q_l = \frac{-B + \sqrt{B^2 - 4AC}}{2A}$$

The source discharge written to the spill hydrograph is Q_l. For the internal downstream pipe-volume update, the module also computes a downstream flow term Q_o when an active downstream boundary or valve contribution is present; otherwise $Q_o = 0$. The net volume removed from the downstream contributing pipe volume is

$$V_{net} = (Q_l - Q_o)\Delta t$$

and the remaining downstream volume and contributing length are updated as

$$V_a^{new} = V_a - V_{net}$$

$$L_a^{new} = \frac{V_a^{new}}{A_p}$$

The oil surface elevation used by the next time step is lowered by

$$z_s^{new} = z_s - m\frac{V_{net}}{A_p}$$

The downstream hydrograph ends when the remaining pipe volume no longer changes by more than the module tolerance.

5.5.4 Combined Source Hydrograph

For each break point, the upstream and downstream hydrographs are combined by time step:

$$Q_{source}(t) = Q_{upstream}(t) + Q_{downstream}(t)$$

The combined hydrograph is written as the source time series referenced by the multiple-sources file. Each source is then applied by OilFlow2D as a point inflow at the generated break coordinates.

5.6 Finite-Volume Numerical Solution

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

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

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 the domain is divided into computational cells, Ω_i, using a fixed mesh. Assuming a piecewise representation of the conserved variables (Figure) and an upwind and unified formulation of fluxes and source terms

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

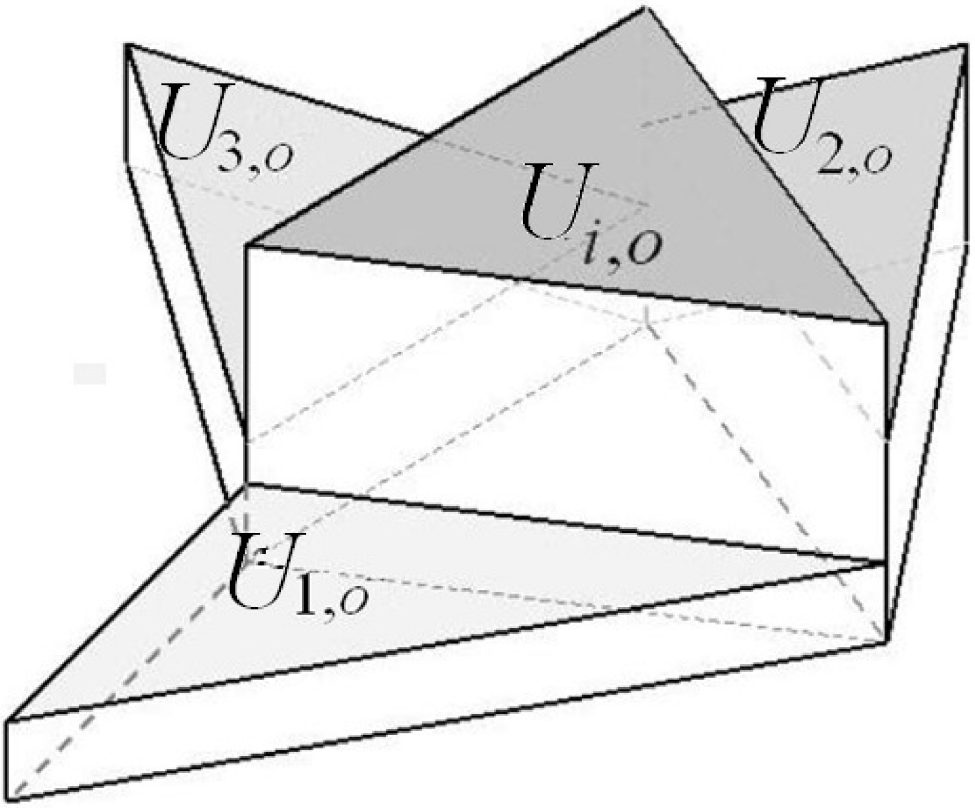


Figure 5.2 – Piecewise uniform representation of the flow variables.

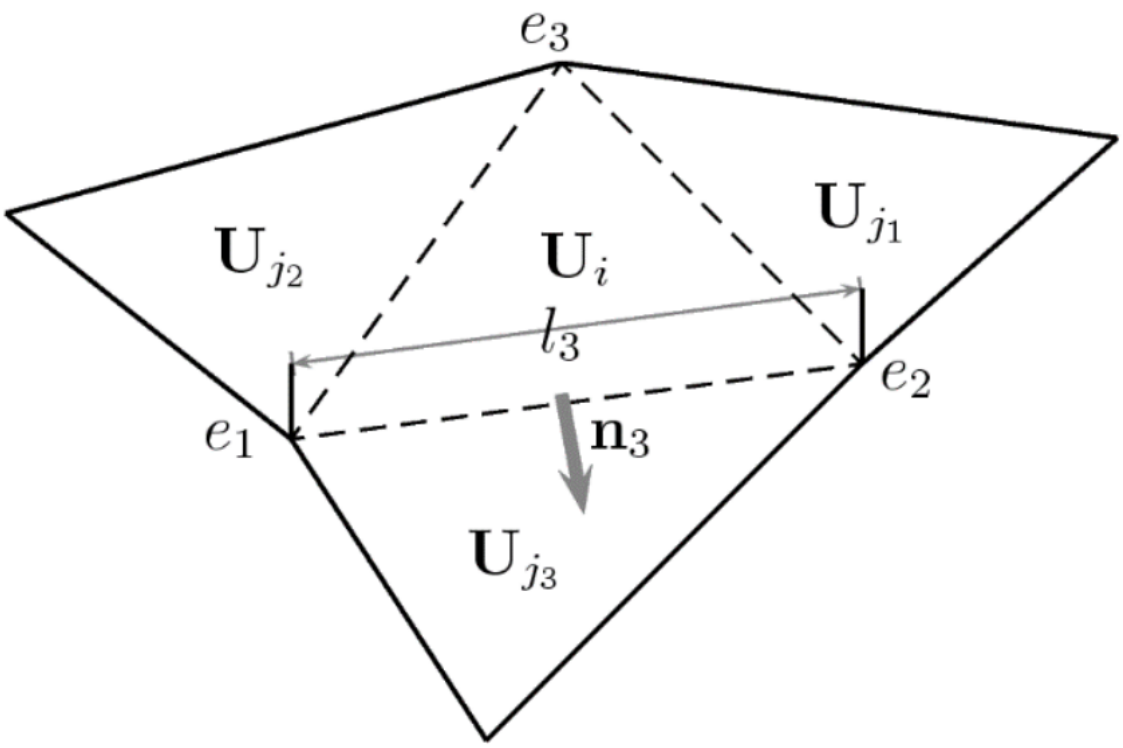


Figure 5.3 – Cell parameters.

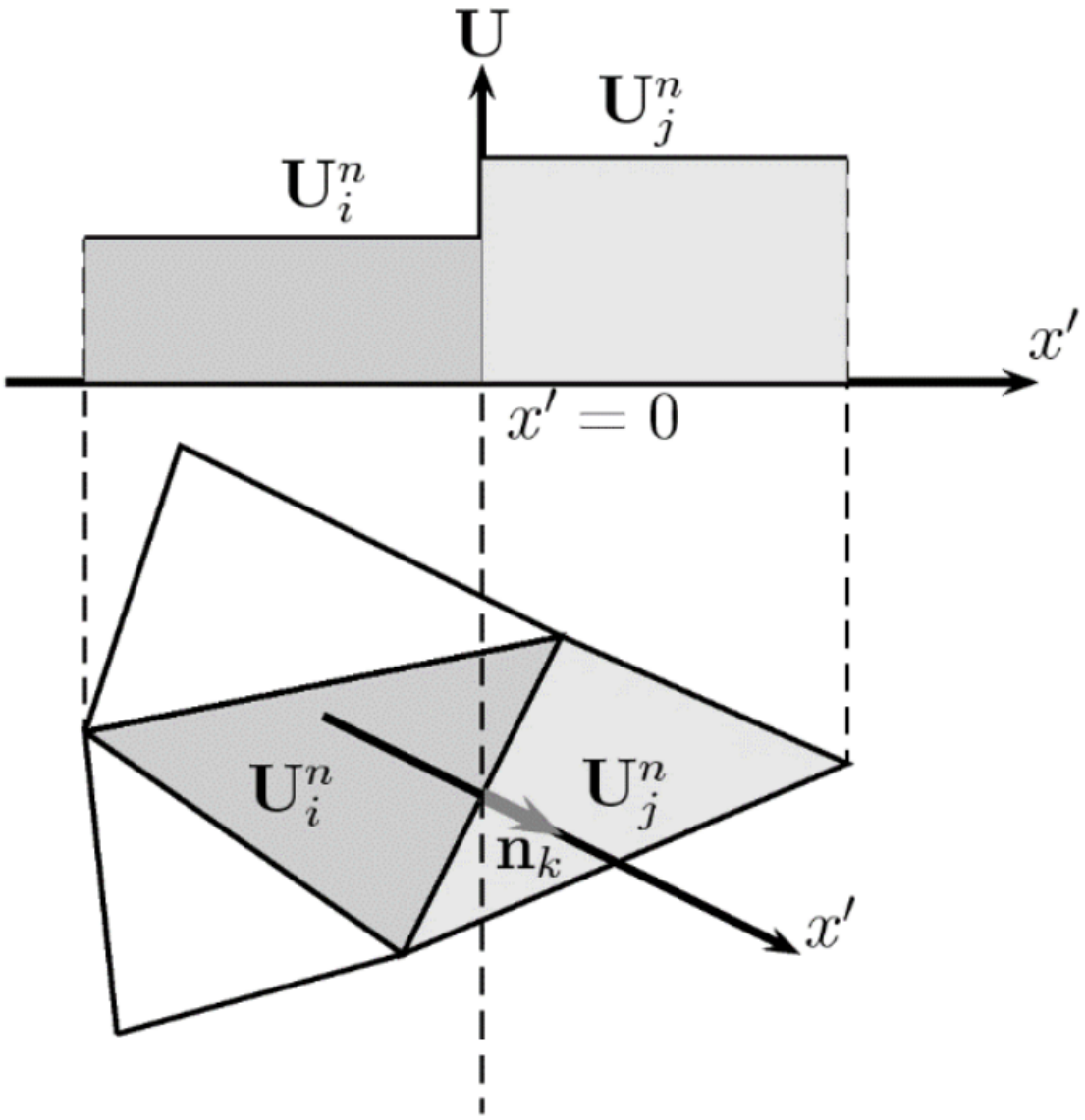


Figure 5.4 – Cell parameters.

The approximate solution can be defined using an approximate Jacobian matrix $\widetilde{\mathbf{J}}_{\mathbf{n},k}$ 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

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

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

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

Both the difference in vector **U** across the grid edge and the the source term are projected onto the matrix eigenvectors basis

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

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. The complete linearization of all terms in combination with the upwind technique allows to define the numerical flux function $(\mathbf{En} - \mathbf{S})_k$ as

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

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 gives an explicit first order Godunov method

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

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

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

can be rewritten as

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

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

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

with

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

The use of is efficient when dealing with boundary conditions. At the same time it ensures conservation. In 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.

5.6.1 Numerical Optimizations

Once wave propagations in $\delta \mathbf{M}_i^-$ in 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 , 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 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 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.

5.6.2 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.

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

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

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

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

OilFlow2D 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 cell size, but also inversely proportional to velocity and depth. Smaller cells 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.

5.7 Open Boundary Conditions

There are two main boundary condition types that can be used in OilFlow2D: Open boundaries where flow can enter of leave the modeling area and closed boundaries that are solid no-flow walls (see Figure). There is no restriction on the number of inlet or outlet boundaries. This section describes the open boundary conditions.

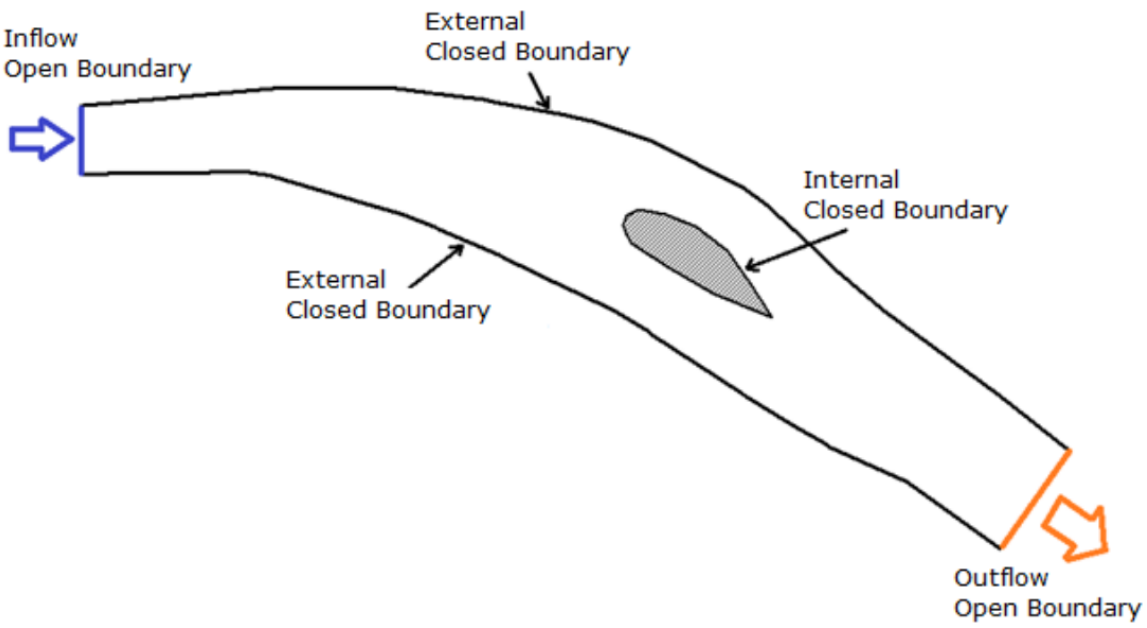


Figure 5.5 – Open and closed boundary conditions.

OilFlow2D 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 OilFlow2D 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.

- **Subcritical:** Q or Velocity; Water Surface Elevation
- **Supercritical:** Q and Water Surface Elevation; None

> **Note**
>
> It is recommended to have at least one boundary where water surface or stage-discharge (e.g. Uniform Flow) is prescribed. Having only discharge and no water surface elevation condition 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.
& Imposes Water Surface Elevation. An associated boundary condition file must be provided.
- **5:** Imposes water discharge and water surface elevation. - **6:** Imposes water discharge inflow. - **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. & Imposes Water Surface Elevation and forces perpendicular velocity directions. An associated boundary condition file must be provided.
- **18:** Imposes Water Surface Elevation and sediment or pollutant concentrations. It also forces

perpendicular velocity directions. An associated boundary condition file must be provided.

> **Note**
>
> If 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 more than one cell (see Figure). Setting two or more open conditions without this separation will lead to incorrect detection of the open boundaries.

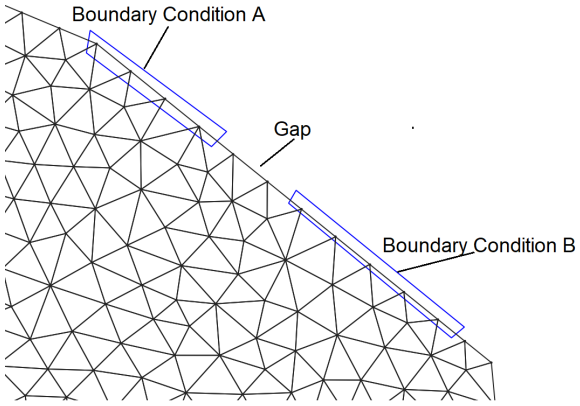


Figure 5.6 – Required gap between adjacent open boundary conditions.

5.7.1 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.

5.7.1.1 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:

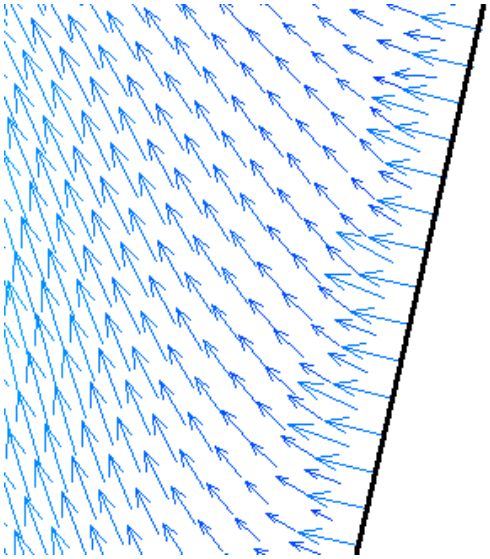


Figure 5.7 – Inflow water discharge imposed as velocities (BCTYPE 6).

5.7.2 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 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.

> **Note**
>
> 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 OilFlow2D version.

5.7.3 “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 cells. 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.

5.7.4 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.

5.7.5 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

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

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

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

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

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

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

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

for supercritical flow, all the following eigenvalues are positive:

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

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:

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

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

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

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.

5.7.5.1 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 analyze 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.

5.7.5.2 Simple cases

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

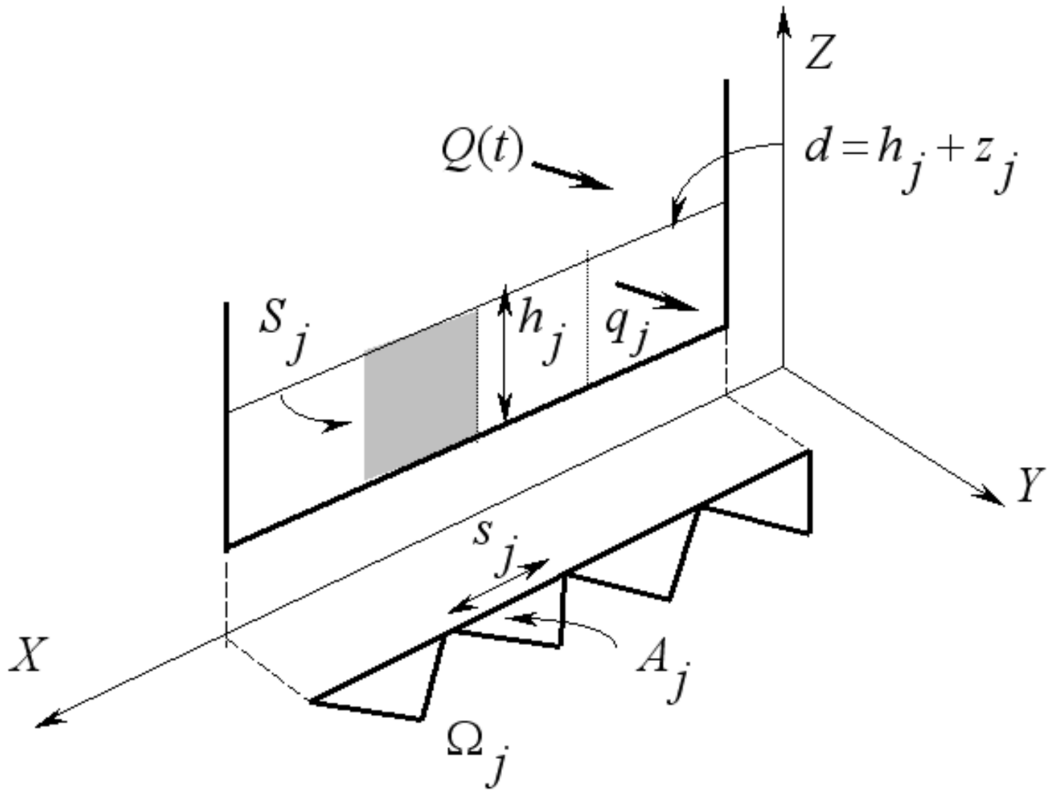


Figure 5.8 – Rectangular inlet cross section.

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

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

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.

5.7.5.3 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).

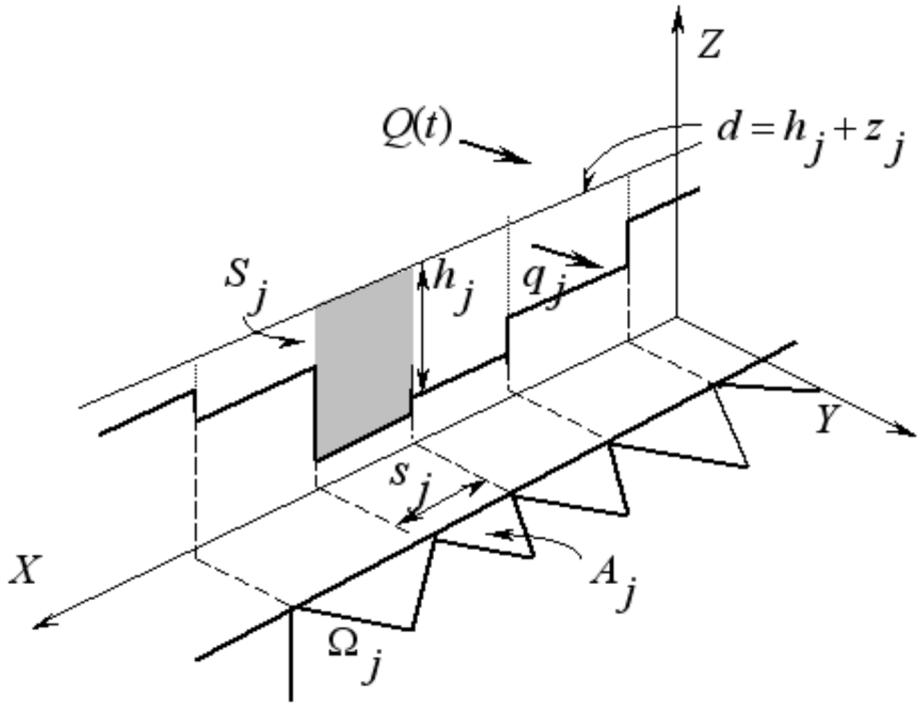


Figure 5.9 – Irregular inlet cross section.

When dealing with inlet sections like that in Figure , a uniform value of q_I as in 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:

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

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) associated, in general, to different water surface levels d_j $d_j = h_j + z_j$.

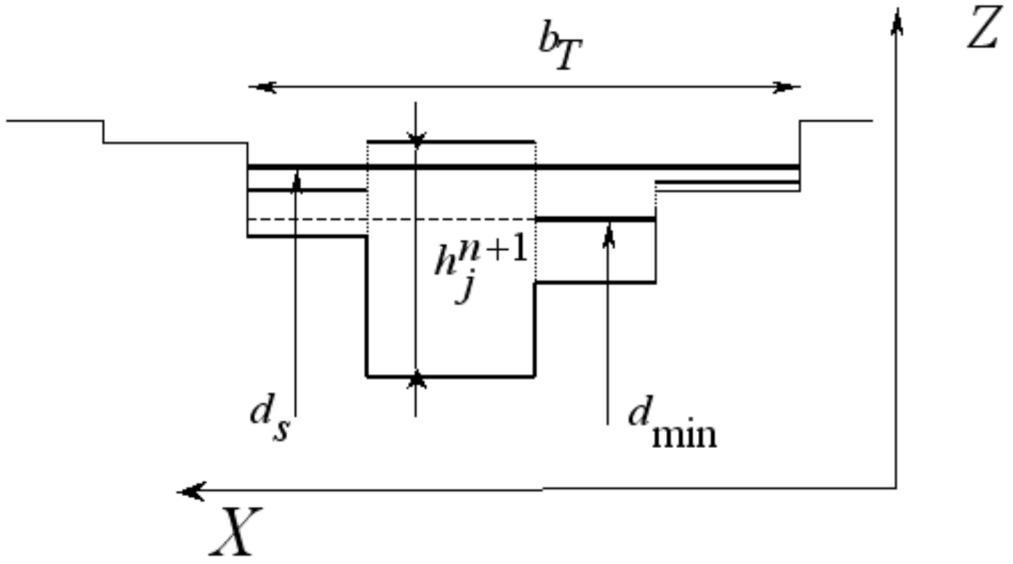


Figure 5.10 – Evaluation of d_{min}.

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

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

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

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

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:

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

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

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

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:

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

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). 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.

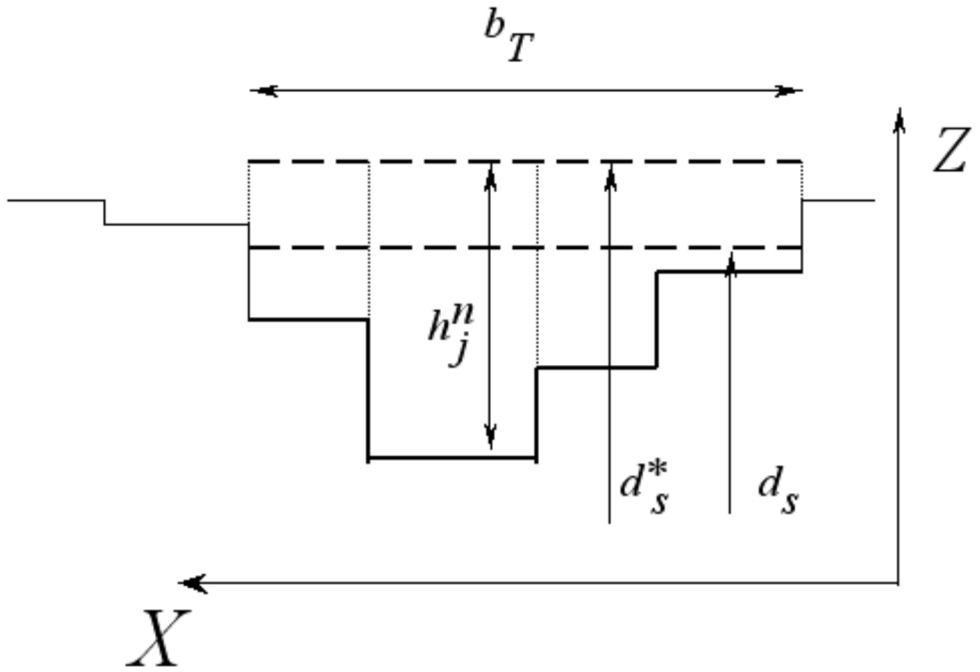


Figure 5.11 – New water level for the inlet section.

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 and imposed in the boundary

cell. Moreover, the water level is computed as a results of the contributions from that other cell edges in when updating the conserved values in the boundary cell at time level $n+1$ and is carefully redistributed as explained before.

5.7.5.4 Outlet boundaries

The analysis of the flow at the outlet boundary is simpler. For supercritical outflow no external conditions have to be imposed. In OilFlow2D, 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.

5.7.6 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 cells to be appropriately resolved. OilFlow2D uses slip condition on closed boundaries and the model will set zero normal flow across the boundary, but tangential velocities are allowed. OilFlow2D 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 **u** after adding all the wave contributions from the rest of the cell edges, where **n** is the solid wall normal (Figure). 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 when updating the conserved values in the boundary cell at time level $n+1$.

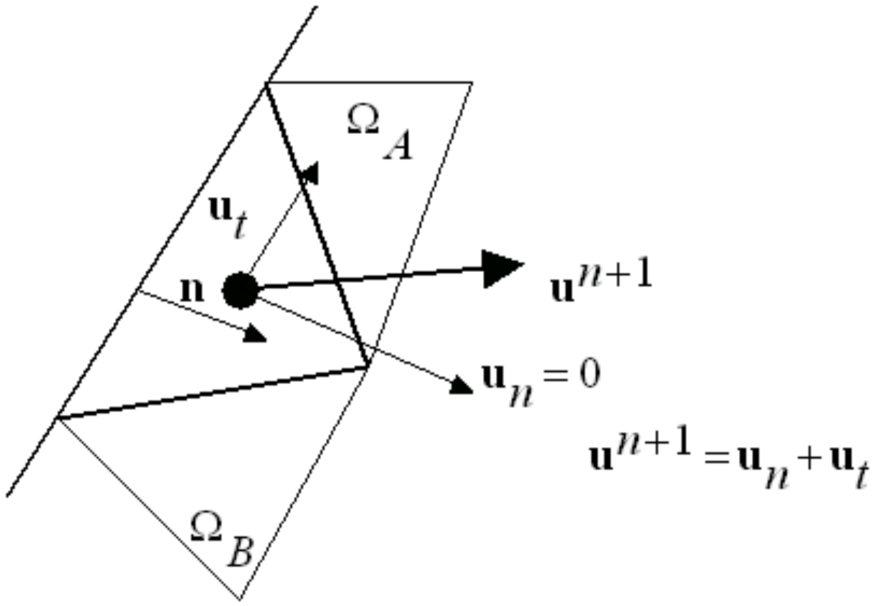


Figure 5.12 – Solid wall condition.

5.8 Dry/Wet Cell Modeling

OilFlow2D 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 OilFlow2D 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 cell classification.

5.8.1 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.
OilFlow2D drying and wetting algorithm is an adaptation of the the one originally proposed by and later improved by and 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.

5.9 Volume Conservation

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

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

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

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

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

$$\Delta M(\Delta t) \neq 0$$

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

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

with the total amount of water existing at time t

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

as follows

$$Error = [Vol(t + \Delta t) - Vol(t)] - \Delta M(\Delta t)$$

This is usually expressed in relative terms as follows:

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

5.10 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.

6

Oil Spill on Water Model

Oil spill simulation is a very complex task where a large number of interacting factors may affect the oil trajectory and fate. Oil trajectory and spreading are two the fundamental processes, since they will determine the areas that will be affected.

The Oil Spills on Water model allows simulation of crude oil spills on the water surface using a 3D particle-tracking approach to represent the oil trajectory. The model also includes algorithms to consider oil evaporation, emulsification, dispersion, dissolution, interaction with shores, and the oil property changes caused by these processes.

Multiple spills can be considered from fixed user selected locations and also from a moving ship. The model can also include containment booms that represent physical barriers with arbitrary levels of efficiency.

6.1 3D Oil Spill Trajectory Algorithm

The trajectory of a particular spill is of fundamental importance since it will determine the oil impact on coastal areas and other sensitive ecosystems. The Lagrangian-particle-tracking approach used in OilFlow2D calculates the oil trajectory assuming that the oil is formed by a relatively large number of particles that move on a pre-calculated velocity field.

Many oil spill models available today use Eulerian schemes that are often accompanied by serious difficulties like spurious oscillations leading to nonphysical representation of the phenomena. Instead of using such Eulerian approach where the pollutant concentration is calculated at certain fixed points, a Lagrangian approach was chosen to model oil spreading and trajectory on water based on experimental results and data reports from many oil spills. In this method the spilled mass is represented by a predetermined number N of tracer particles. This transport model, based on tracer or Lagrangian concepts, calculates the particle trajectories in a turbulent flow field considering advection by the flow and dispersion of the oil in water column.

The main advantage of this technique compared to the traditional Eulerian method is the elimination

of *numerical* diffusion.

In turbulence theory the concentration of a substance may be represented by a finite number of particles moving with the velocity at their instantaneous position. Usually, it is assumed that the water velocity can be expressed as a mean flow *advective* velocity u_a plus a smaller scale random fluctuation due to turbulent diffusion u_d.

To calculate the particle trajectories the model solves the following ordinary differential equations

$$\frac{dx_i}{dt} = u_{ai} + u_{di}$$

$$\frac{dy_i}{dt} = v_{ai} + v_{di}$$

$$\frac{dz_i}{dt} = w_{ai} + w_{di}$$

where x_i, y_i, and z_i are the coordinates of the i particle, t is the time, u_{ai}, v_{ai}, and w_{ai} are the water velocities that advect particle i in x, y, and z directions respectively and u_{di}, v_{di}, and w_{di} are the particle i velocities due to diffusion in the x, y, and z directions respectively.

These equations may be solved using Euler numerical method as follows

$$x_i^{n+1} = x_i^n + \Delta x_i^n + \Delta x_i^{n'}$$

$$y_i^{n+1} = y_i^n + \Delta y_i^n + \Delta y_i^{n'}$$

$$z_i^{n+1} = z_i^n + \Delta z_i^n + \Delta z_i^{n'}$$

where Δt is the time interval, x_i^{n+1}, y_i^{n+1}, and z_i^{n+1} are the i particle coordinates for time $(n+1)\Delta t$, x_i^n, y_i^n, and z_i^n are the i particle coordinates for time $n\Delta t$, and Δx_i^n, Δy_i^n, and Δz_i^n are the advective particle displacements defined by

$$\Delta x_i^n = u_{ai}\Delta t$$

$$\Delta y_i^n = v_{ai}\Delta t$$

$$\Delta z_i^n = w_{ai}\Delta t$$

and Δt is the time interval, $\Delta x_i^{n'}$, $\Delta y_i^{n'}$, and $\Delta z_i^{n'}$ are the i particle displacements due to the random velocity fluctuations defined as

$$\Delta x_i^{n'} = u_{di}\Delta t$$

$$\Delta y_i^{n'} = v_{di}\Delta t$$

$$\Delta z_i^{n'} = w_{di}\Delta t$$

The horizontal advective field (u_a, v_a) is calculated by the water current model and the vertical velocity w_d considers the buoyancy forces due to differences in densities between water and oil through the Stokes law a logarithmic velocity profile described below.
The random velocities (u_d, v_d, w_d) due to diffusion are obtained using Montecarlo sampling in a range of velocities $[-U_r, U_r]$ proportional to the diffusion coefficient. U_r is calculated as follows. Assuming large-scale Brownian motion the isotropic diffusion coefficient D may be expressed as

$$D = \left(\frac{1}{2\tau}\int_{-\infty}^{+\infty} x^2 f(x)dx\right) \Big/ \left(\int_{-\infty}^{+\infty} f(x)dx\right)$$

where $f(x)$ is a probability distribution that determines the particle displacement x due to the random fluctuations in time τ. It may be proved that equation depends mainly on the variance of $f(x)$ and not on its specific form. Therefore, this model adopts a *tophat* distribution that, introduced in determine the relationship between ϵ and the diffusion velocity fluctuation range $[-U_r, U_r]$ in the following way

$$U_r = \sqrt{\frac{6D_x}{\Delta t}}$$

where D_x is the diffusion coefficient in x direction.
An analogue deduction may be performed to obtain the random velocity fluctuation ranges in y and z directions ($[-V_r, V_r], [-W_r, W_r]$).
To calculate oil concentration at a particular point (x, y, z), the model counts the number of particles inside a control cubic volume around the point. The concentration corresponding to each particle is the initial concentration divided by the number of particles used to represent the spill.
To calculate oil thickness Z, the following expression is used:

$$Z = N(\Delta A)Z_p$$

where $N(\Delta A)$ is the number of particles contained in the reference surface area ΔA, and Z_p is the *particle thickness* defined as:

$$Z_p = \frac{V}{\Delta A N_p}$$

V is the initial spill volume, and N_p is the total number of particles used to represent the oil.

6.1.1 Three-dimensional 3D Flow Field

Since the OilFlow2D hydrodynamic component is based on the two-dimensional shallow water equations that provide the depth average velocities, in OilFlow2D oil spills on water model the vertical distribution of the streamwize velocity is obtained assuming the logarithmic velocity profile. That assumption allows representing the three dimensional flow field in an approximated manner.

$$U(z) = \frac{U_*}{\kappa} \ln\left(\frac{z}{z_0}\right)$$

where $U(z)$ is streamwize velocity at elevation z
U_* is the shear velocity = $\sqrt{\tau_0/\rho}$
τ_0 is the bottom shear
ρ is the water density

κ is the von Karman constant $\approx= 0.41$

z_0 is the bottom friction length

6.1.2 Dispersion Coefficients

Adequate estimation of the dispersion coefficients is one of the basic factors affecting oil spills, since they determine the random velocity ranges and consequently the oil trajectories.

The complexity of the dynamic process that govern turbulent dispersion in water and the spreading dynamics justify an empirical approach to estimate these coefficients. OilFlow2D can estimate horizontal dispersion coefficients such that the particle spreading calculated with the Lagrangian scheme would be equivalent to the solution given by the modified Mackay formulation .

With this assumption, the spreading coefficient can be obtained knowing that $d\sigma^2/dt \propto dR/dt$, where R represents the radius of the circular slick. In addition, for spreading in 2D, the variance and its derivative with respect to time are $\sigma^2 = 4Dt$, $d\sigma^2/dt = 4D$, and $d\sigma^2/dt = dA/dt$.

The resulting expression gives a relationship between the oil spreading law and the dispersion coefficient D as shown in the table below, where k_1, k_2, and k_3 are constants that depend on the spreading regime, according to Fay: $k_1 = 1.14$, $k_2 = 1.45$, and $k_3 = 1.0$.

Regime	Slick radius	Dispersion coefficient
Gravity-Inertia	$\frac{k_1}{2}(\Delta g V t^2)^{1/4}$	$\frac{\pi k_1^2}{16}(\Delta g V)^{1/2}$
Gravity-Viscous	$\frac{k_2}{2}\left(\frac{\Delta g V^2 t^{3/2}}{\nu_w^{1/2}}\right)^{1/6}$	$\frac{\pi k_2^2}{32}\left(\frac{\Delta g V^2}{\nu_w^{1/2}}\right)^{1/3}\frac{1}{\sqrt{t}}$
Surface tension-Viscous	$\frac{k_3}{2}\left(\frac{\sigma^2 t^3}{\rho_w^2 \nu_w}\right)^{1/4}$	$\frac{3\pi k_3^2}{16}\left(\frac{\sigma}{\rho_w \nu_w^{1/2}}\right)\sqrt{t}$

6.1.3 Comparison with Analytical Solution

The oil spill trajectory model was compared with a 3-D analytical solution. The simplest solution corresponds to the instantaneous release of mass M of a solute in an unconfined static fluid at $t = 0$. The resulting concentration distribution is given by (Crank):

$$c = \frac{M}{(4\pi t)^{3/2}(D_x D_y D_z)^{1/2}} exp\left(-\frac{x^2}{4D_x t} - \frac{y^2}{4D_y t} - \frac{z^2}{4D_z t}\right)$$

Figure 1.1 shows the comparison of the model with the analytical solution at point (0,5,0) for an spill of of $1000\,m^3$ of at the origin of coordinates (0,0,0). The diffusion coefficients were: $D_x = D_y = D_z = 1\times10^{-2}\,m^2/s$. Different runs were done for number of particles ranging from 5.000 to 100.000. Note how as the number of particles is increased the numerical solution approaches the analytical one.

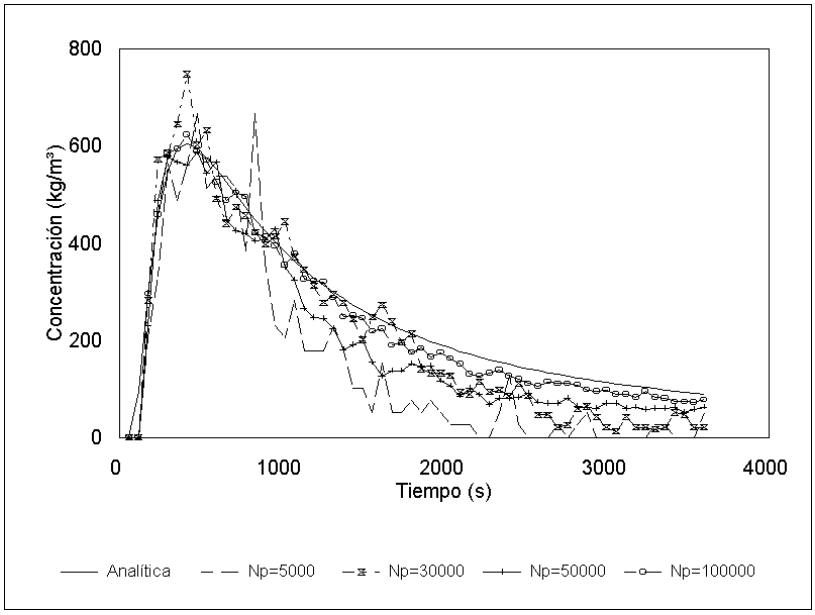


Figure 6.1 – Comparison of numerical and analytical solution for instantaneous spill for different number of particles Np.

6.1.4 Comparison with Experiments

To test the validity of the model and the expressions deduced previously for A_k and B_k, a set of oil spreading experiments were performed in a laboratory wave tank 14 m long, 9 m wide and 0 .60 m deep, varying the oil types, spilled volume, wave period and wave height. A total of thirty four oil spills were tested with Victoria, Mesa, and Lago Medio crude oils. The oil-water and oil-air interfacial tensions was measured in the laboratory for each oil sample using a Fisher ring tensiometer. Density and viscosity were measured before each test. Details of these tests are reported in .

Figure 1.2 presents the time variation of the radius of the circular slick measured with Mesa oil as well as the calculations performed with various formulations. It may be seen that the proposed formulation (Rf2) compares well with the experimental data. Mackay thick and thin formulas differ with the experiments and with the other formulations.

Figure 1.3 presents similar results for Lago Medio oil. It is also observed that the proposed formulation (Rf2) in this case also compares very well the experimental data, while the Mackay thick and thin formulas deviates from the experimental results. Also Mackay's thick slick formulation underestimates the spreading radius in all tests performed in this study.

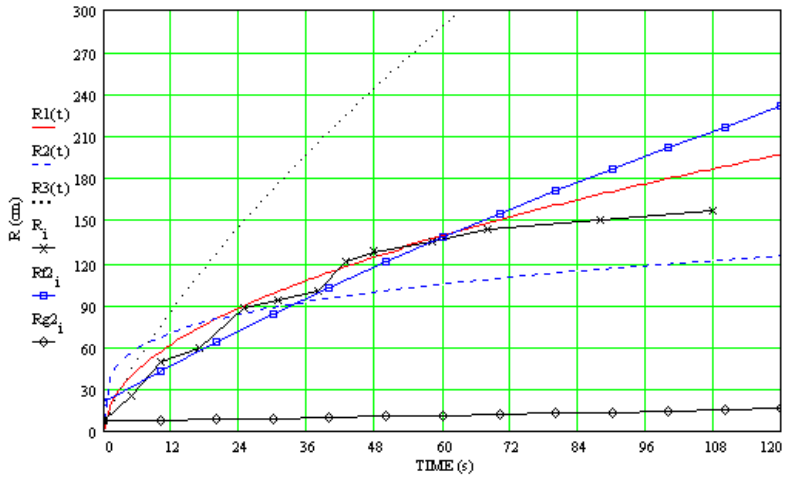


Figure 6.2 – Mesa oil spreading experiments. Mackay and OilFlow2D formulations.

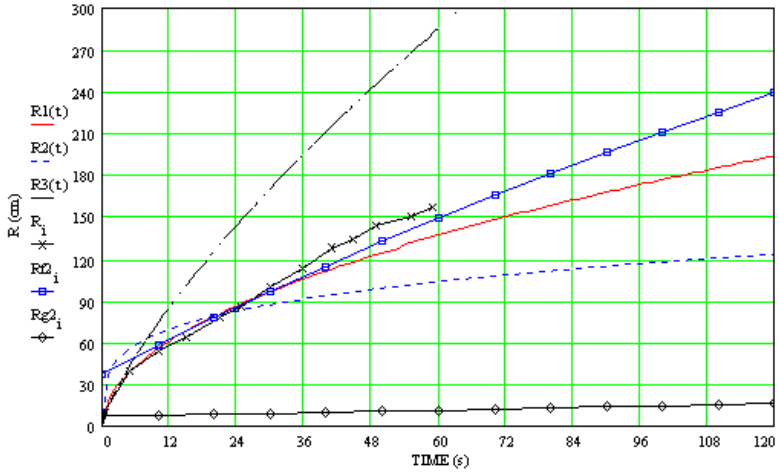


Figure 6.3 – Lago Medio oil spreading experiments. Mackay and OilFlow2D formulation.

6.2 Evaporation

Immediately after an oil spill occurs, hydrocarbons begin to evaporate, thus reducing the volume of spilled oil. To consider this phenomenon, it is necessary calculate the fraction of oil that has evaporated at each time interval depending on the crude oil properties. OilFlow2D offers two formulations to calculate the evaporated oil fraction, one proposed by Fingas (2010) and another that of Stiver-Mackay (1984).

6.2.1 Stiver-Mackay formulation

The Stiver-Mackay formulation is as follows:

$$\frac{dF_e}{dt} = \Theta \left[6.3 - \left(\frac{10.3}{T_{oil}} \right) \left(T_0 + P\ F_e \right) \right]$$

where F_e is the fraction of the total volume of the slick that has been evaporated.
Θ is the so-called *exposure coefficient* and is defined as:

$$\Theta = \frac{K_m A}{V_o}$$

being K_m the mass transfer coefficient which can be written as:

$$K_m = 0.0025\, W^{0.78}$$

W is the wind velocity at 10 m above the surface in m/s, A the slick area expressed in m^2, t is the spill time in seconds, $V$is the initial volume of the spill in m^3, T, is oil temperature in K, $T_0 = 532.95 - 3.125API$, is the initial boiling point for oil initial air temperature in K, $P = 985.62 - 13.597API$ is represents the slope of the oil standard distillation (Temperature vs. Evaporated fraction).

6.2.2 Fingas formulation

According to Fingas (2010), the evaporation of oils and petroleum products can be described using the concept of diffusion-controlled regulation through the oil layer and the oil-air surface interface. This mechanism differs fundamentally from air-boundary-layer regulation, which applies to pure, rapidly evaporating liquids such as water.
Evaporation models are generally classified into two categories: those based on air-boundary-layer-controlled evaporation and those based on diffusion-regulated evaporation. Experimental studies demonstrate that oil evaporation does not follow air-boundary-layer control. Unlike water, where evaporation rates are strongly influenced by atmospheric conditions, oil evaporation is governed primarily by diffusion processes within the oil layer itself.
Because according to Fingas, oil evaporation is not air-boundary-layer regulated, a relatively simple evaporation formulation is sufficient to describe the process. Factors such as wind speed, turbulence intensity, surface area, and scale effects do not need to be explicitly considered. Instead, the dominant variables controlling evaporation are time and temperature. Oil thickness also plays a secondary role and is therefore included in diffusion-regulated evaporation formulations.
Fingas (2010) developed a simplified, empirically based modeling framework that provides evaporation equations for more than 200 common oils and petroleum products. These formulations were derived from experimental studies of oil evaporation behavior. For most oils, evaporation follows a functional relationship of the form $F_e = 0.01(a + b\,T)\ln(t)$, where a and b are empirical constants, T represents temperature, and t denotes time.
Some oils, such as diesel fuels, exhibit a different evaporation behavior, following a relationship of the form $F_e = 0.01(a + b\,T)\sqrt{(t)}$. Diesel and similar fuels therefore display a distinct temporal curvature in their evaporation profiles, particularly during the early stages of evaporation, when compared to most other oils.
The resulting Fingas (2010) formulations to determine the evaporation fraction are summarized in the following tables:

Oil (Group 1)	$F_e \times 100$	Oil (Group 2)	$F_e \times 100$
Adgo	$(0.11 + 0.013\,T)\sqrt{t}$	Chavyo	$(3.52 + 0.045\,T)\ln(t)$
Adgo-long term	$(0.68 + 0.045\,T)\ln(t)$	Combined oil/gas	$(-0.08 + 0.013\,T)\sqrt{t}$

Oil (Group 1)	$F_e \times 100$	Oil (Group 2)	$F_e \times 100$
Alberta Sweet Mixed Blend	$(3.24 + 0.054\,T)\,\ln(t)$	Compressor Lube Oil-new	$(-0.68 + 0.045\,T)\,\ln(t)$
Amauligak	$(1.63 + 0.045\,T)\,\ln(t)$	Cook Inlet Trading Bay	$(3.15 + 0.045\,T)\,\ln(t)$
Amauligak-f24	$(1.91 + 0.045\,T)\,\ln(t)$	Cook Inlet-Granite Point	$(4.54 + 0.045\,T)\,\ln(t)$
Arabian Heavy	$(1.31 + 0.045\,T)\,\ln(t)$	Cook Inlet-Swanson River	$(3.58 + 0.045\,T)\,\ln(t)$
Arabian Heavy	$(2.71 + 0.045\,T)\,\ln(t)$	Corrosion Inhibitor Solvent	$(-0.02 + 0.013\,T)\sqrt{t}$
Arabian Light	$(2.52 + 0.037\,T)\,\ln(t)$	Cusiana	$(3.39 + 0.045\,T)\,\ln(t)$
Arabian Light	$(3.41 + 0.045\,T)\,\ln(t)$	Delta West Block 97	$(6.57 + 0.045\,T)\,\ln(t)$
Arabian Light (2001)	$(2.4 + 0.045\,T)\,\ln(t)$	Diesel (regular stock)	$(0.31 + 0.018\,T)\sqrt{t}$
Arabian Medium	$(1.89 + 0.045\,T)\,\ln(t)$	Diesel Anchorage-Long	$(4.54 + 0.045\,T)\,\ln(t)$
ASMB (offshore)	$(2.2 + 0.045\,T)\,\ln(t)$	Diesel Anchorage-Short	$(0.51 + 0.013\,T)\sqrt{t}$
ASMB-Standard #5	$(3.35 + 0.045\,T)\,\ln(t)$	Diesel fuel-Southern-long term	$(2.18 + 0.045\,T)\,\ln(t)$
Av Gas 80	$(15.4 + 0.045\,T)\,\ln(t)$	Diesel fuel-Southern-short term	$(-0.02 + 0.013\,T)\sqrt{t}$
Avalon	$(1.41 + 0.045\,T)\,\ln(t)$	Diesel Mobile 1997	$(0.03 + 0.013\,T)\sqrt{t}$
Avalon J-34	$(1.58 + 0.045\,T)\,\ln(t)$	Diesel Mobile 1997 long-term	$(-0.02 + 0.013\,T)\sqrt{t}$
Aviation Gasoline 100 LL	$(0.5 + 0.045\,T)\,\ln(t)$	Diesel-long term	$(5.8 + 0.045\,T)\,\ln(t)$
Barrow Island	$(4.67 + 0.045\,T)\,\ln(t)$	Dos Cuadros	$(1.88 + 0.045\,T)\,\ln(t)$
BCF-24	$(1.08 + 0.045\,T)\,\ln(t)$	Ekofisk	$(4.92 + 0.045\,T)\,\ln(t)$
Belridge Crude	$(0.03 + 0.013\,T)\sqrt{t}$	Empire Crude	$(2.21 + 0.045\,T)\,\ln(t)$
Bent Horn A-02	$(3.19 + 0.045\,T)\,\ln(t)$	Endicott	$(0.9 + 0.045\,T)\,\ln(t)$
Beta	$(-0.08 + 0.013\,T)\sqrt{t}$	Esso Spartan EP-680 Industrial Oil	$(-0.66 + 0.045\,T)\,\ln(t)$
Beta-long term	$(0.29 + 0.045\,T)\,\ln(t)$	Eugene Island 224-condensate	$(9.53 + 0.045\,T)\,\ln(t)$
Boscan	$(-0.15 + 0.013\,T)\sqrt{t}$	Eugene Island Block 32	$(0.77 + 0.045\,T)\,\ln(t)$
Brent	$(3.39 + 0.048\,T)\,\ln(t)$	Eugene Island Block 43	$(1.57 + 0.045\,T)\,\ln(t)$
Bunker C Anchorage	$(-0.13 + 0.013\,T)\sqrt{t}$	Evendell	$(3.38 + 0.045\,T)\,\ln(t)$
Bunker C Anchorage (long term)	$(0.31 + 0.045\,T)\,\ln(t)$	FCC Heavy Cycle	$(0.17 + 0.013\,T)\sqrt{t}$
Bunker C-Light (IFO-250)	$(0.0035 + 0.0026\,T)\sqrt{t}$	FCC Light	$(-0.17 + 0.013\,T)\sqrt{t}$
Bunker C-long term	$(-0.21 + 0.045\,T)\,\ln(t)$	FCC Medium Cycle	$(-0.16 + 0.013\,T)\sqrt{t}$
Bunker C-short term	$(0.35 + 0.013\,T)\sqrt{t}$	FCC-VGO	$(2.5 + 0.013\,T)\sqrt{t}$
California API 11	$(-0.13 + 0.013\,T)\sqrt{t}$	Federated	$(3.47 + 0.045\,T)\,\ln(t)$
California API 15	$(-0.14 + 0.013\,T)\sqrt{t}$	Federated (new-1999)	$(3.45 + 0.045\,T)\,\ln(t)$
Cano Limon	$(1.71 + 0.045\,T)\,\ln(t)$	Garden Banks 387	$(1.84 + 0.045\,T)\,\ln(t)$
Carpenteria	$(1.68 + 0.045\,T)\,\ln(t)$	Garden Banks 426	$(3.44 + 0.045\,T)\,\ln(t)$

Oil (Group 1)	$F_e \times 100$	Oil (Group 2)	$F_e \times 100$
Cat cracking feed	$(-0.18 + 0.013\,T)\sqrt{t}$	Gasoline	$(13.2 + 0.21\,T)\ln(t)$

Evaporated fraction as a function of temperature T and time t according to the Fingas (2010) formulation

Oil (Group 3)	$F_e \times 100$	Oil (Group 4)	$F_e \times 100$
Genesis	$(2.12 + 0.045\,T)\ln(t)$	Maya	$(1.38 + 0.045\,T)\ln(t)$
Green Canyon Block 109	$(1.58 + 0.045\,T)\ln(t)$	Mayan crude	$(1.45 + 0.045\,T)\ln(t)$
Green Canyon Block 184	$(3.55 + 0.045\,T)\ln(t)$	Mississippi Canyon Block 194	$(2.62 + 0.045\,T)\ln(t)$
Green Canyon Block 65	$(1.56 + 0.045\,T)\ln(t)$	Mississippi Canyon Block 72	$(2.15 + 0.045\,T)\ln(t)$
Greenplus Hydraulic Oil	$(-0.68 + 0.045\,T)\ln(t)$	Mississippi Canyon Block 807	$(2.05 + 0.045\,T)\ln(t)$
Gullfaks	$(2.29 + 0.034\,T)\ln(t)$	Nektoralik	$(0.62 + 0.045\,T)\ln(t)$
Heavy Reformate	$(-0.17 + 0.013\,T)\sqrt{t}$	Neptune Spar (Viosca Knoll 826)	$(3.75 + 0.045\,T)\ln(t)$
Hebron MD-4	$(1.01 + 0.045\,T)\ln(t)$	Nerlerk	$(2.01 + 0.045\,T)\ln(t)$
Heidrun	$(1.95 + 0.045\,T)\ln(t)$	Ninian	$(2.65 + 0.045\,T)\ln(t)$
Hibernia	$(2.18 + 0.045\,T)\ln(t)$	Norman Wells	$(3.11 + 0.045\,T)\ln(t)$
High Viscosity Fuel Oil	$(-0.12 + 0.013\,T)\sqrt{t}$	North Slope-Middle Pipeline	$(2.64 + 0.045\,T)\ln(t)$
Hondo	$(1.49 + 0.045\,T)\ln(t)$	North Slope-Northern Pipeline	$(2.64 + 0.045\,T)\ln(t)$
Hout	$(2.29 + 0.045\,T)\ln(t)$	North Slope-Southern Pipeline	$(2.47 + 0.045\,T)\ln(t)$
IFO-180	$(-0.12 + 0.013\,T)\sqrt{t}$	Nugini	$(1.64 + 0.045\,T)\ln(t)$
IFO-30 (Svalbard)	$(-0.04 + 0.045\,T)\ln(t)$	Odoptu	$(4.27 + 0.045\,T)\ln(t)$
IFO-300 (old Bunker C)	$(-0.15 + 0.013\,T)\sqrt{t}$	Oriente 1	$(1.32 + 0.045\,T)\ln(t)$
Iranian Heavy	$(2.27 + 0.045\,T)\ln(t)$	Oriente 2	$(1.57 + 0.045\,T)\ln(t)$
Issungnak	$(1.56 + 0.045\,T)\ln(t)$	Orimulsion 400-dewater	$3.6\ln(t)$
Isthmus	$(2.48 + 0.045\,T)\ln(t)$	Orimulsion plus water	$(3 + 0.045\,T)\ln(t)$
Jet 40 Fuel	$(8.96 + 0.045\,T)\ln(t)$	Oseberg	$(2.68 + 0.045\,T)\ln(t)$
Jet A1	$(0.59 + 0.013\,T)\sqrt{t}$	Panuke	$(7.12 + 0.045\,T)\ln(t)$
Jet Fuel (Anch)	$(7.19 + 0.045\,T)\ln(t)$	Pitas Point	$(7.04 + 0.045\,T)\ln(t)$
Jet Fuel (Anch) short term	$(1.06 + 0.013\,T)\sqrt{t}$	Platform Gail (Sockeye)	$(1.68 + 0.045\,T)\ln(t)$
Komineft	$(2.73 + 0.045\,T)\ln(t)$	Platform Holly	$(1.09 + 0.045\,T)\ln(t)$
Lago	$(1.13 + 0.045\,T)\ln(t)$	Platform Irene-long term	$(0.74 + 0.045\,T)\ln(t)$

Oil (Group 3)	$F_e \times 100$	Oil (Group 4)	$F_e \times 100$
Lago Treco	$(1.12 + 0.045\,T)\ln(t)$	Platform Irene-short term	$(-0.05 + 0.013\,T)\sqrt{t}$
Lucula	$(2.17 + 0.045\,T)\ln(t)$	Point Arguello Heavy	$(0.94 + 0.045\,T)\ln(t)$
Main Pass Block 306	$(2.86 + 0.045\,T)\ln(t)$	Point Arguello Light	$(2.44 + 0.045\,T)\ln(t)$
Main Pass Block 37	$(3.04 + 0.045\,T)\ln(t)$	Point Arguello Light-b	$(2.3 + 0.045\,T)\ln(t)$
Malongo	$(1.67 + 0.045\,T)\ln(t)$	Point Arguello-co-mingled	$(1.43 + 0.045\,T)\ln(t)$
Marinus Turbine Oil	$(-0.68 + 0.045\,T)\ln(t)$	Polypropylene Tetramer	$0.25\,t$
Marinus Valve Oil	$(-0.68 + 0.045\,T)\ln(t)$	Port Hueneme	$(0.3 + 0.045\,T)\ln(t)$
Mars TLP	$(2.18 + 0.045\,T)\ln(t)$		
Maui	$(-0.14 + 0.013\,T)\sqrt{t}$		

Evaporated fraction as a function of temperature T and time t according to the Fingas (2010) formulation

Oil (Group 5)	$F_e \times 100$	Oil (Group 6)	$F_e \times 100$
Prudhoe Bay (new stock)	$(2.37 + 0.045\,T)\ln(t)$	Vasconia	$(0.84 + 0.045\,T)\ln(t)$
Prudhoe Bay (old stock)	$(1.69 + 0.045\,T)\ln(t)$	Viosca Knoll Block 826	$(2.04 + 0.045\,T)\ln(t)$
Prudhoe stock b	$(1.4 + 0.045\,T)\ln(t)$	Viosca Knoll Block 990	$(3.16 + 0.045\,T)\ln(t)$
Rangely	$(1.89 + 0.045\,T)\ln(t)$	Voltesso 35	$(-0.18 + 0.013\,T)\sqrt{t}$
Sahara Blend	$(0.001 + 0.013\,T)\sqrt{t}$	Waxy Light and Heavy	$(1.52 + 0.045\,T)\ln(t)$
Sahara Blend (long term)	$(1.09 + 0.045\,T)\ln(t)$	West Delta Block 30 w/water	$(-0.04 + 0.013\,T)\sqrt{t}$
Sakalin	$(4.16 + 0.045\,T)\ln(t)$	West Texas Intermediate 1	$(2.77 + 0.045\,T)\ln(t)$
Santa Clara	$(1.63 + 0.045\,T)\ln(t)$	West Texas Intermediate 2	$(3.08 + 0.045\,T)\ln(t)$
Scotia Light 1	$(6.92 + 0.045\,T)\ln(t)$	West Texas Sour	$(2.57 + 0.045\,T)\ln(t)$
Scotia Light 2	$(6.87 + 0.045\,T)\ln(t)$	White Rose	$(1.44 + 0.045\,T)\ln(t)$
Ship Shoal Block 239	$(2.71 + 0.045\,T)\ln(t)$	Zaire	$(1.36 + 0.045\,T)\ln(t)$
Ship Shoal Block 269	$(3.37 + 0.045\,T)\ln(t)$		
Sockeye	$(2.14 + 0.045\,T)\ln(t)$		
Sockeye Co-mingled	$(1.38 + 0.045\,T)\ln(t)$		
Sockeye Sour	$(1.32 + 0.045\,T)\ln(t)$		
Sockeye Sweet	$(2.39 + 0.045\,T)\ln(t)$		
South Louisiana	$(2.39 + 0.045\,T)\ln(t)$		
South Pass Block 60	$(2.91 + 0.045\,T)\ln(t)$		
South Pass Block 67	$(2.17 + 0.045\,T)\ln(t)$		

Oil (Group 5)	$F_e \times 100$	Oil (Group 6)	$F_e \times 100$
South Pass Block 93	$(1.5 + 0.045\,T)\ln(t)$		
South Timbalier Block 130	$(2.77 + 0.045\,T)\ln(t)$		
Statfjord	$(2.67 + 0.06\,T)\ln(t)$		
Sumatran Heavy	$(-0.11 + 0.013\,T)\sqrt{t}$		
Sumatran Light	$(0.96 + 0.045\,T)\ln(t)$		
Taching	$(-0.11 + 0.013\,T)\sqrt{t}$		
Takula	$(1.95 + 0.045\,T)\ln(t)$		
Tapis	$(3.04 + 0.045\,T)\ln(t)$		
Tchatamba Crude	$(3.8 + 0.045\,T)\ln(t)$		
Terra Nova	$(1.36 + 0.045\,T)\ln(t)$		
Terresso 150	$(-0.68 + 0.045\,T)\ln(t)$		
Terresso 220	$(-0.66 + 0.045\,T)\ln(t)$		
Terresso 46	$(-0.67 + 0.045\,T)\ln(t)$		
Industrial oil Thevenard Island	$(5.74 + 0.045\,T)\ln(t)$		
Turbine Oil STO 120	$(-0.68 + 0.045\,T)\ln(t)$		
Turbine Oil STO 90	$(-0.68 + 0.045\,T)\ln(t)$		
Udang	$(-0.14 + 0.013\,T)\sqrt{t}$		
Udang (long term)	$(0.06 + 0.045\,T)\ln(t)$		

Evaporated fraction as a function of temperature T and time t according to the Fingas (2010) formulation

6.3 Emulsification

This is one of the least understood yet most critical processes influencing the evolution of an oil slick, particularly with respect to the phenomenon of oil properties changes and oil sinking. A careful analysis is required due to its significant impact on the physical behavior and fate of spilled oil. The process involves the formation of a water-in-oil emulsion, which transforms the spilled oil into a highly viscous mixture. Owing to its dark color and thick, semi-solid appearance, this emulsion is commonly referred to as *chocolate mousse*.

Oil emulsification occurs as the oil spreads over the water surface. Certain oil components tend to accumulate at the oil-water interface, forming a stabilizing layer composed primarily of asphaltenes, resins, and waxes. This interfacial layer inhibits the coalescence of individual oil droplets. Nevertheless, partial aggregation still occurs, and as oil fragments merge, water becomes entrapped between them, forming thin films that evolve into small, stable emulsified structures. The retained water provides structural support to the emulsion, thereby promoting and sustaining the water-in-oil emulsification process.

The propensity of crude oil to emulsify depends strongly on the oil's composition as well as on the environmental conditions prevailing at the spill site, such as temperature, wave energy, and salinity. The incorporated water fraction into the oil slick due to emulsion formation can be calculated using

the Macay's formulation (Sebastiao and Soares, 1995):

$$\frac{dF_{em}}{dt} = K_{em}(1 + W^2)(1 - \frac{F_{em}}{F_{max}})$$

where F_{em} is the water fraction in the oil with respect to the total oil slick volume, F_{max} is the maximum water content, and W is the wind velocity at 10 m above the surface in m/s.

6.4 Density and Viscosity Change

The density and viscosity of an oil slick on the sea surface are primarily governed by oil weathering processes. Among these, evaporation and emulsification are the dominant mechanisms influencing viscosity evolution. Although evaporation alters oil properties, its impact on viscosity has been reported to be less significant than that of emulsification. Emulsification leads to a rapid increase in slick viscosity. Therefore, the effect of emulsion formation must be explicitly accounted for when estimating slick viscosity.
To estimate the oil density change the model uses the following formula:

$$\rho_o = F_{em}\rho_w + \rho_{oi}(1 - F_{em})(1 + 0.18F_e)$$

where ρ_o is the oil density at a given time (kg/m^3), ρ_{oi} is the initial oil density (kg/m^3), and ρ_w is the water density (kg/m^3).
To determine the viscosity change the model uses the following formula:

$$\mu_{ow} = \mu_{oi} \exp(10F_e) \exp\left(\frac{2.5F_{em}}{1 - 0.65F_{em}}\right) \exp\left(-63.16\frac{T_{ref} - T_o}{T_o T_{ref}}\right)$$

where μ_{ow} is the oil-water mixture viscosity (cP), μ_{oi} is the initial oil viscosity (cP), T_o is the oil temperature (K), and T_{ref} is the initial oil temperature (K).

6.5 Oil Spreading Formulation

Many authors have treated the theme among which the works of Fay and Hoult may be considered as the first attempts to formulate relatively simple formulas useful to simulate oil spreading. In 1980, Mackay et al.[1] presented the now landmark work *Oil Spill Processes and Models* , where the most important phenomena that determine oil spills were formulated in a simple manner. In particular, the spreading formulation, based on Fay formulas, applied the concept of thick and thin slick that has had wide acceptance. It appears that most research work since 1980 has been dedicated to study other important processes with the assumption that the spreading process was correctly modeled. Mackay spreading formulation for the thin slick is based on the Fay's surface tension spreading regime. The time evolution of oil area is written:

$$A_{tn} = K_{tn}t^{3/2}$$

where A_{tn} is the oil thin slick area, t is the time, and K_{tn} is assumed to be constant.
Differentiating the previous equation with respect to time:

[1] In this manual Mackay et al. report will be called indistinctly Mackay report

$$\frac{dA_{tn}}{dt} = \frac{3}{2} K_{tn} t^{1/2}$$

Substituting t from the area equation into the derivative and simplifying, the following expression is obtained:

$$\frac{dA_{tn}}{dt} = K' {A_{tn}}^{1/3}$$

where $K' = 3/2 K_{tn}^{2/3}$
Applying Euler method to the ordinary differential equation and assuming K' independent of time (Mackay assumes K_{tn} is constant), it is approximated as:

$$\Delta A_{tn} = A_k {A_{tn}}^{1/3} \Delta t$$

where ΔA_{tn} is the thin slick area change in the time interval Δt, and $A_k = K'$.
Mackay assumed that the thin slick is of constant thickness ($1 \mu m$), and as it spreads, *draws* oil from the thick slick. To account for the retardation of this *feeding* mechanism, Mackay introduces an empirical exponential factor:

$$\Delta A_{tn} = A_k {A_{tn}}^{1/3} \exp(\frac{-C_k}{Z}) \Delta t$$

where C_k is a constant to be adjusted, and Z is the thick slick thickness.
To simulate the thick slick, Mackay derivation starts from the Fay's gravity-viscous equation:

$$A_{tk} = K_{tk} V^{2/3} t^{1/2}$$

where A_{tk} is the oil thick slick area, and K_{tk} is assumed to be constant.
Differentiating the previous equation with respect to time:

$$\frac{dA_{tk}}{dt} = \frac{1}{2} K_{tk} V^{2/3} t^{-1/2}$$

Substituting t from the thick slick area equation into its derivative, assuming constant thick slick thickness ($V = A_{tk} Z$), and simplifying, the following expression is obtained:

$$\frac{dA_{tk}}{dt} = K'_{tk} Z^{4/3} {A_{tk}}^{1/3}$$

where $K'_{tk} = 1/2 K_{tk}^2$.
Applying Euler method to the ordinary differential equation, assuming K'_{tk} independent of time (remember again that Mackay assumes K_{tk} is constant) and that Z does not change significantly in the Δt time interval, it is approximated as:

$$\Delta A_{tk} = B_k Z^{4/3} {A_{tk}}^{1/3} \Delta t$$

where ΔA_{tk} is the thin slick area change in the time interval Δt, and $B_k = K'_{tk}$.
The final equation for thick slick area evolution taking into account loss of volume due to the *feeding* of the thin slick is according to Mackay

$$\Delta A_{tk} = B_k Z^{4/3} {A_{tk}}^{1/3} \Delta t - \left(10^{-6} \frac{\Delta A_{tn}}{Z} \right)$$

Mackay determines A_k, B_k, and C_k constants by adjusting them so that the thin- and thick-slick equations fit a particular spreading field experiment. The values obtained were:

$$A_k = 1.0$$

$$B_k = 150.0$$

$$C_k = 0.0015$$

Although these values were fitted to a particular set of field tests, most oil spill models available today use the same constants disregarding the influence of the oil properties in its determination.

6.5.1 A_k and B_k Formulation

To analyze the Mackay equations and to determine the correct expressions of A_k and B_k, we will start with the Fay and Hoult original surface tension spreading formulation. Assuming circular spreading we have:

$$A_{tn} = \frac{\pi}{4} \frac{\sigma}{\rho_w \nu_w^{1/2}} t^{3/2}$$

where ρ_w is the water density, ν_w is the water kinematic viscosity, and σ is the net spreading coefficient defined as:

$$\sigma = \sigma_{aw} - \sigma_{ow} - \sigma_{oa}$$

where σ_{aw}, σ_{ow} and σ_{oa} are the air-water, oil-water and oil-air interfacial tensions respectively.
Comparing the Fay and Hoult expression with Mackay's thin slick expression, it is observed that K_{tn} is not a constant but actually:

$$K_{tn} = \frac{\pi}{4} \frac{\sigma}{\rho_w \nu_w^{1/2}}$$

Substituting this value into A_k the following expression is obtained:

$$A_k = \frac{3}{2} \left(\frac{\pi}{4}\right)^{2/3} \left(\frac{\sigma}{\rho_w \nu_w^{1/2}}\right)^{2/3}$$

Note that A_k has dimensions $[L^{4/3}/T]$.
The previous analysis shows that A_k is not a constant but depends on the net oil spreading coefficient and on the water properties.
To analyze Mackay thick slick equation, we start with Fay and Hoult gravity-viscous spreading formulation. Assuming again circular spreading we have:

$$A_{tk} = \frac{\pi}{4} \left(\frac{\Delta g}{\nu_w^{1/2}}\right)^{1/3} V^{2/3} t^{1/2}$$

where g is the gravitational acceleration, $\Delta = 1 - \rho_o/\rho_w$, and ρ_o is the oil density.

Comparing the Fay and Hoult expression with Mackay's thick slick expression, it is observed that K_{tk} is not a constant but actually:

$$K_{tk} = \frac{\pi}{4}\left(\frac{\Delta g}{\nu_w^{1/2}}\right)^{1/3}$$

Substituting this value into B_k the following expression is obtained:

$$B_k = \frac{1}{2}\left(\frac{\pi}{4}\right)^2\left(\frac{\Delta g}{\nu_w^{1/2}}\right)^{2/3}$$

Note that B_k has dimensions $[1/T]$.

Therefore, B_k is not a constant but depends on the oil and water physical properties.

Figure 1.4 shows the dependence of A_k on the spreading coefficient σ. Mackay's proposed value of $A_k = 1m^{4/3}/s = 464.16cm^{4/3}/s$ corresponds to $\sigma \approx 714$ dynes/cm. It is well known that the σ value for existing oils is around 25 dynes/cm. There is not any oil with surface tension values in the range of that implied by Mackay's constant.

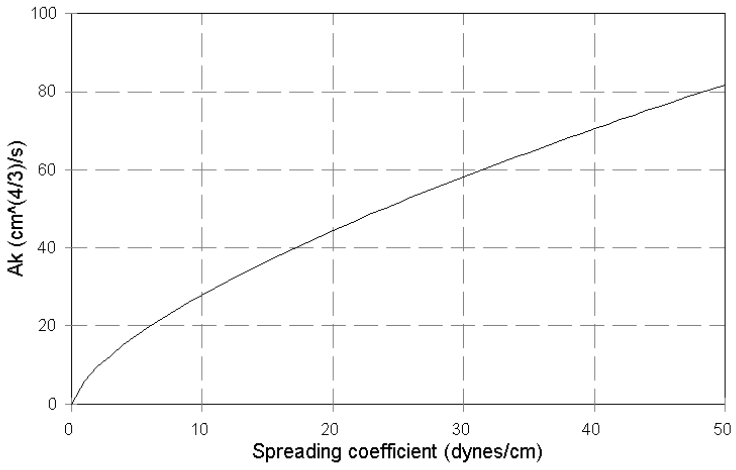


Figure 6.4 – A_k vs. oil spreading coefficient σ in sea water

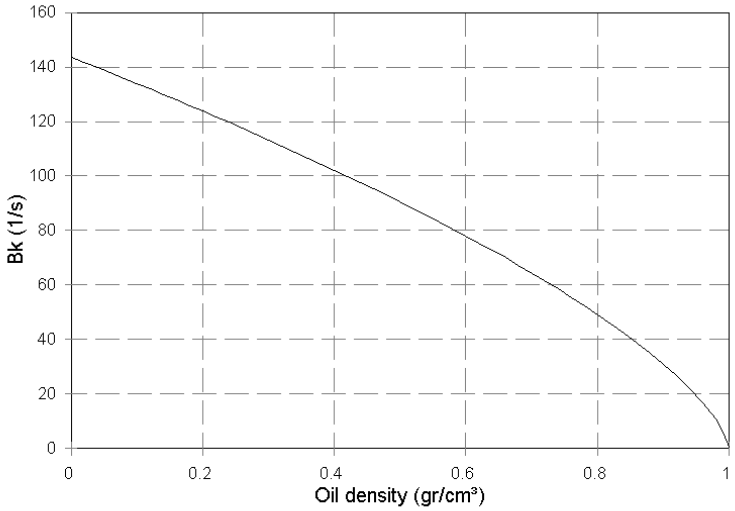


Figure 6.5 – B_k as a function of oil density ρ_o

Figure 1.5 shows the dependence of B_k on oil density. Note that the Mackay's suggested value of $B_k = 150$ is out of range and does not correspond to any existing oil. The appropriate values should be around $B_k = 30$.
The previous analysis demonstrated that errors may be obtained if the values of A_k and B_k originally suggested by Mackay are used for all oil spill simulations without taking into consideration oil and water properties.

6.6 Containment Booms

OilFlow2D can consider the effect of containment booms in the trajectory of oil spills using the *Booms* component. Booms are entered as polylines in the QGIS *SpillBooms* layer and are assumed to be physical barriers to the particles that represent the oil. These barriers can be partial depending on an *Oil Trapping Fraction (OTF)* parameter assigned to each boom. For instance if the OTF is 1, all of the oil particles will be trapped by the boom, but if OTF is set to 0.1, only 10% of the particles will be contained and 90% will pass through the boom.

6.7 Setting up a Oil Spill on Water Simulation

Please, consult the Tutorials document for step-by-step explanation about setting up a complete oil spill on water simulation.

7

Pollutant Transport Model: PL

The study of solute transport phenomena and river mixing has become a great concern in hydraulic and environmental problems. OilFlow2D 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 of simulating pollutant transport is usually focused around determining 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

7.1 Model Equations

Although OilFlow2D 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 pollutant 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:

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

where

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

$$\mathbf{F} = \left(q_x,\, \tfrac{q_x 2}{h} + \tfrac{1}{2}gh2,\, \tfrac{q_x q_y}{h},\, h\phi u\right)^T, \qquad \mathbf{G} = \left(q_y,\, \tfrac{q_x q_y}{h},\, \tfrac{q_y 2}{h} + \tfrac{1}{2}gh2,\, h\phi v\right)^T$$

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

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

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

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

7.2 Pollutant Transport Finite-Volume Numerical Solution

In OilFlow2D, 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 , 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, OilFlow2D 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 was implemented in OilFlow2D.

According to , 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,

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

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

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

where

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

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

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). 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).

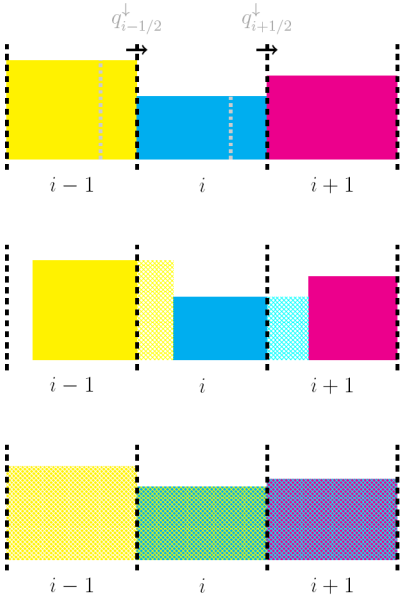


Figure 7.1 – Physical representation of solute mass exchange between cells with $q^{\downarrow}_{i-1/2}, q^{\downarrow}_{i+1/2} > 0$.

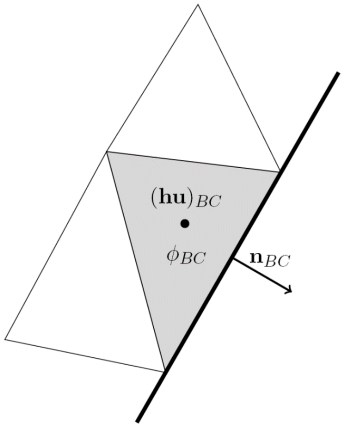


Figure 7.2 – Extraction of mass solute in an outlet boundary cell.

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

7.3 Entering Data for the Pollutant Transport Model

To enter data for a pollutant transport simulation use the *Pollutant Transport* panel. Also make sure that the Pollutant Transport check box is active in thee Control Data tab.

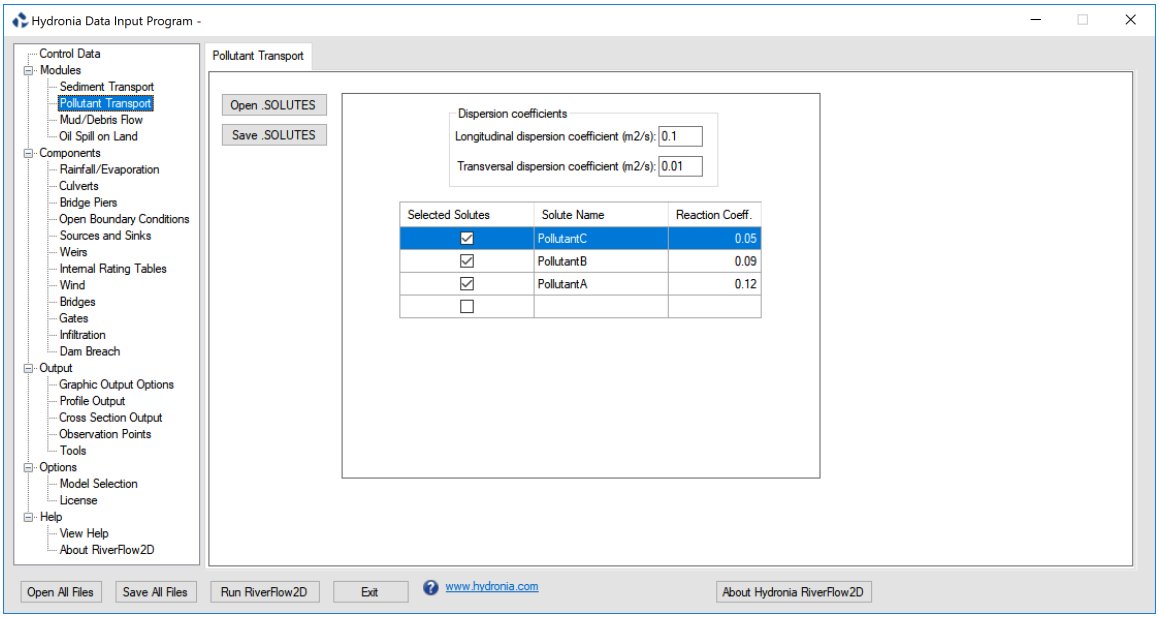


Figure 7.3 – DIP *Pollutant Transport* panel.

7.4 Assumptions of the Pollutant Transport Model

The main assumptions involved in the present version of OilFlow2D model are:

1. There is no predetermined limit to the number of pollutants.
2. The pollutant concentration units are arbitrary. The user can use volume concentration, mg/l, ppt, ppm, or any other suitable units, provided that the inflow boundary conditions are consistent.
3. Interaction between solutes and between each solute and water are assumed to be first order reactions.
4. All inflow boundaries where either discharge or water elevation is imposed must provide a concentration time series for each pollutant.

8

Code Parallelization

8.1 OilFlow2D CPU

OilFlow2D code has been parallelized using OpenMP directives available in the Intel C++ compiler. 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 OilFlow2D 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 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.

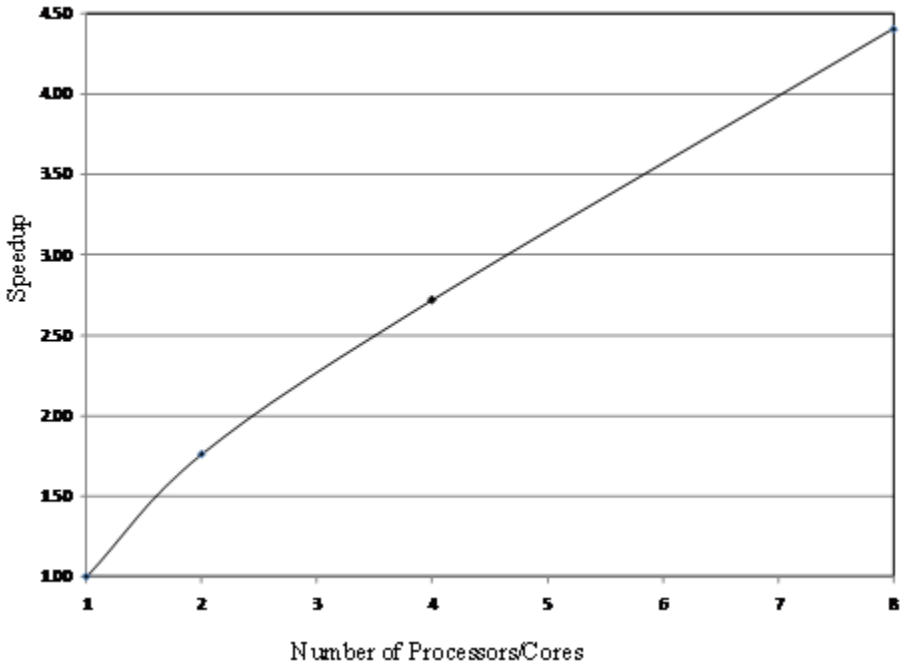


Figure 8.1 – Speed up using OilFlow2D parallelized code as a function of number of processor cores.

8.2 OilFlow2D GPU

The GPU version of the OilFlow2D model offers amazing speedups that considerably reduce run times. OilFlow2D GPU implements two strategies: OpenMP parallelization and GPU techniques. Since dealing with transient inundation flows the number of wet calls 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. OilFlow2D 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 700X when compared with non-parallelized CPU (1-core) runs. Figure shows performance tests using recent GPU hardware technology, that demonstrate that the parallelization techniques implemented in OilFlow2D GPU can significantly reduce the computational cost.

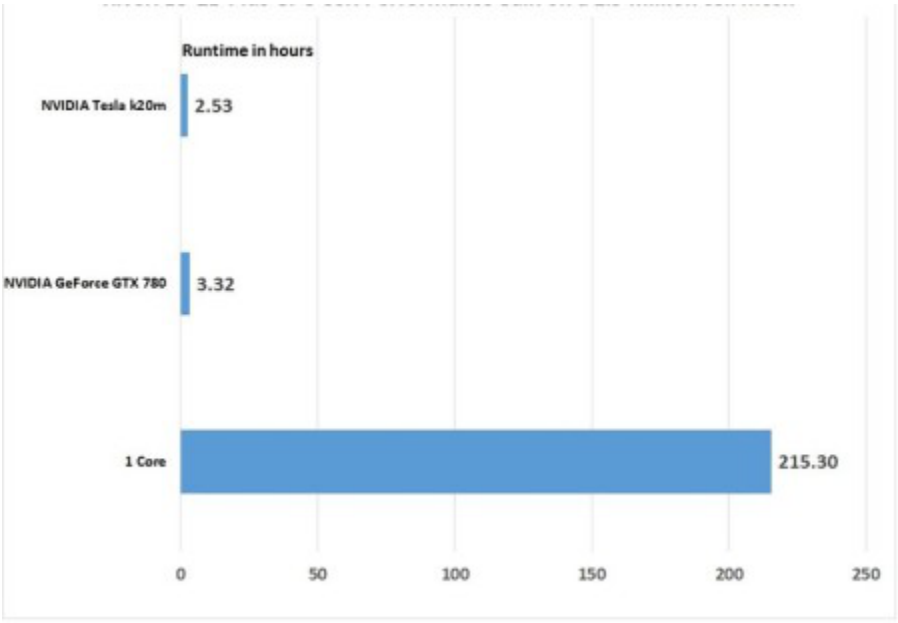


Figure 8.2 – Speed up using OilFlow2D parallelized code as a function of number of processor cores.

9

Components to Model Internal Structures

OilFlow2D 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 components are set over polygons:

- **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.
- **Dam Breach:** accounts for internal dams or levees that can break.
- **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.

9.1 Bridges Component

OilFlow2D 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 data file which is generated by OilFlow2D model based on the user defined data in DIP. To run a simulation with the bridges component, you need to select the option in the *Control Data* panel as shown in Figure.

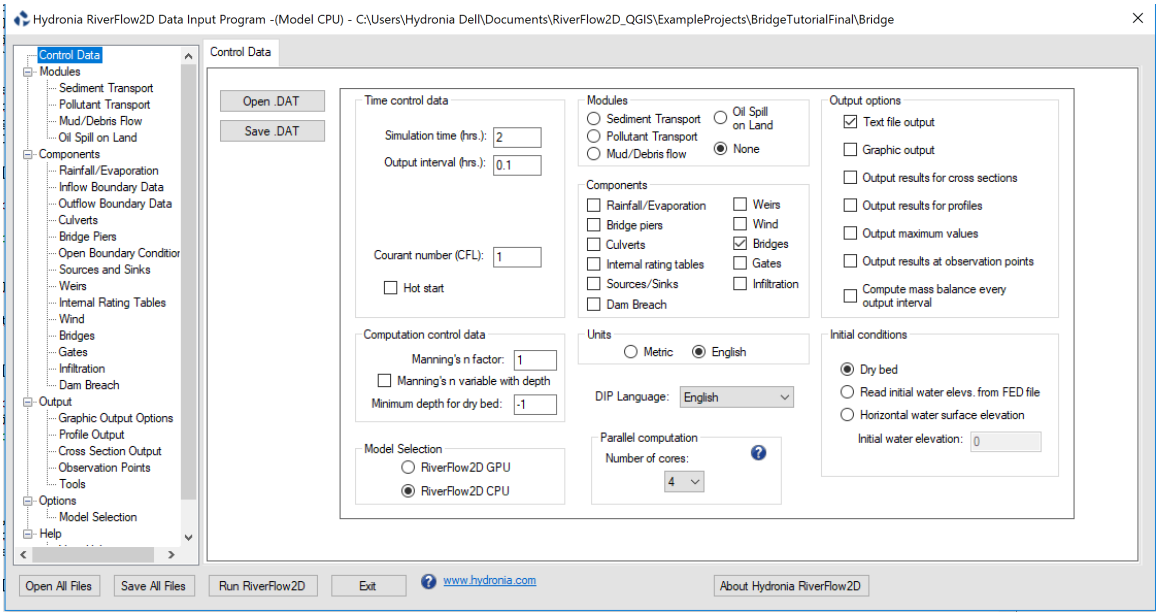


Figure 9.1 – DIP *Control Data* panel with the Bridges component selected.

The bridge plan data is entered in OilFlow2D *Bridges* layer. To create a bridge, please consult the *Simulating bridges* tutorial in the Tutorials document.

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

9.1.1 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:

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.**

9.1.1.1 Example of the Bridge Cross Section File

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

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

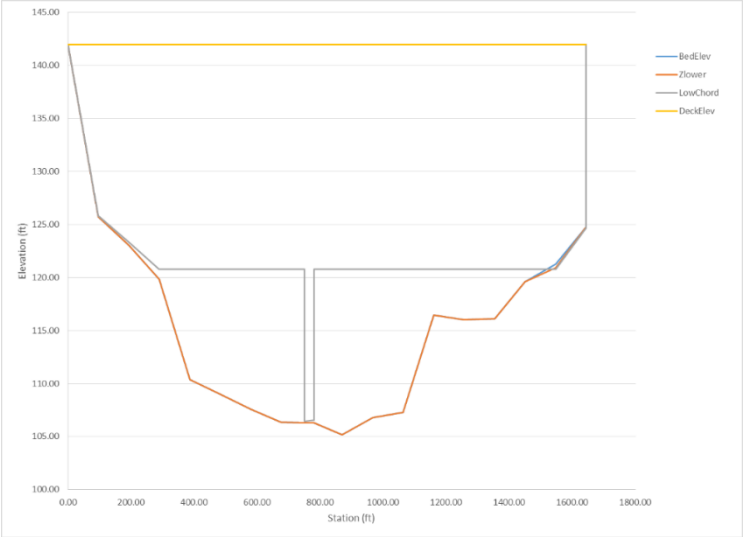


Figure 9.2 – Front view of a bridge cross section.

- **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 an area blocked 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 an area blocked to the flow. The space between ZLOWER and LOWCHORD is open to the flow. If the bridge has no holes, ZLOWER must be identical to BEDELEV.

9.1.2 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

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

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 $\mathbf{S}_f$

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

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

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

and is used to represent bridges.

The description of energy losses for the friction term associated to the bed stress in equation is commonly formulated as an extension of a 1D formulation. The same approach is applied in OilFlow2D 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.

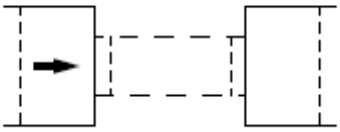


Figure 9.3 – Top view of a bridge showing the cross sections of interest. Only two piers are depicted for simplicity.

9.1.2.1 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. 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

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

where ΔH_c and ΔH_e are the contraction and expansion losses respectively

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

where m is a typical value for the contraction coefficient, $m = 0.62$ and the areas A_1 to A_4 refer to effective cross sectional flow area. The numbering of areas is shown in Figures. 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.

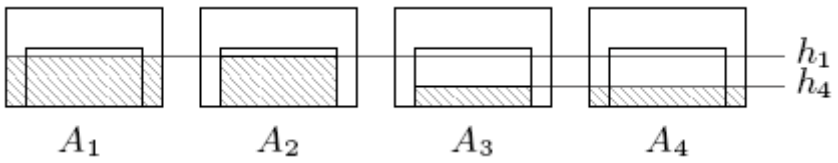


Figure 9.4 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in free surface bridges.

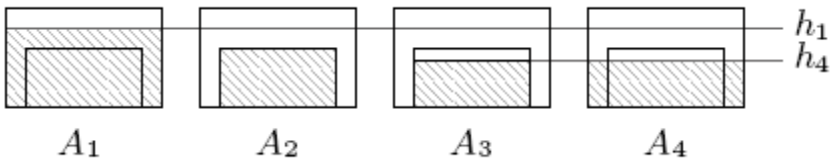


Figure 9.5 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in a partially submerged bridges.

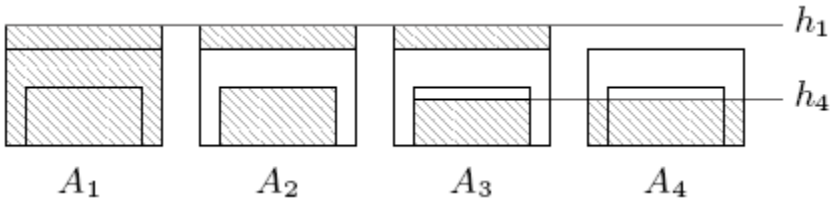


Figure 9.6 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in fully submerged bridges.

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

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

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 shows a sketch of the areas considered in the free surface case, Figure shows the equivalent areas for partially submerged bridges and Figure for fully submerged bridges.

9.1.2.2 Integration of the energy losses generated by bridges

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

$$(\mathbf{S_n}_b)_k = \begin{pmatrix} 0 \\ -g\widetilde{h}\,\delta H n_x \\ -g\widetilde{h}\,\delta H n_y \end{pmatrix}_k$$

with

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

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), 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). Note that it is possible to define bridges in arbitrary orientations and in structured/unstructured meshes.

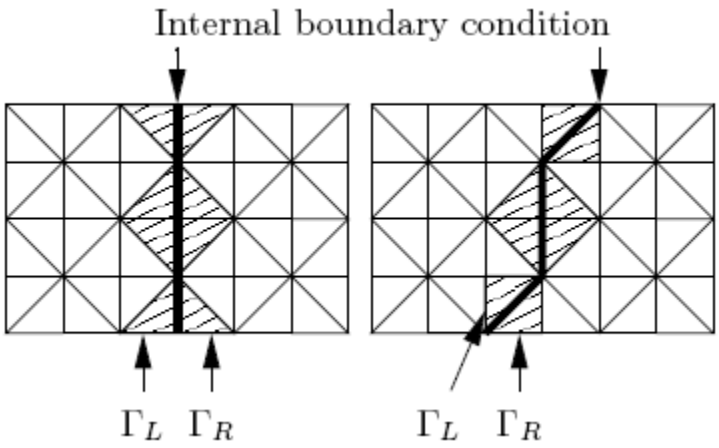


Figure 9.7 – Application of the scheme in triangular structured meshes. Normal bridge (left) and oblique bridge (right).

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. The discharge is computed as

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

and the cross sectional average water level surface is estimated as

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

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.

9.1.2.3 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 and , 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. 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).

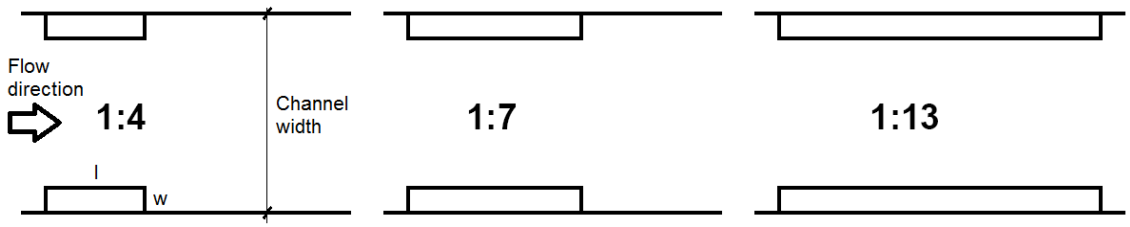


Figure 9.8 – Bridge pier proportions used to asses the influence of the structure width.

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

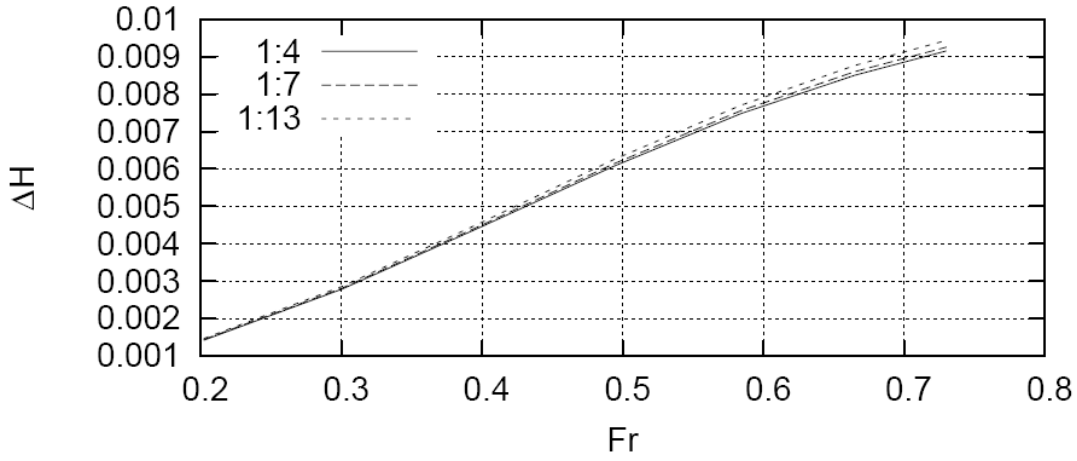


Figure 9.9 – Influence of the structure width on the total head change (ΔH) across the bridge as a function of the Froud number downstream.

9.2 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 DIP as shown in Figure.

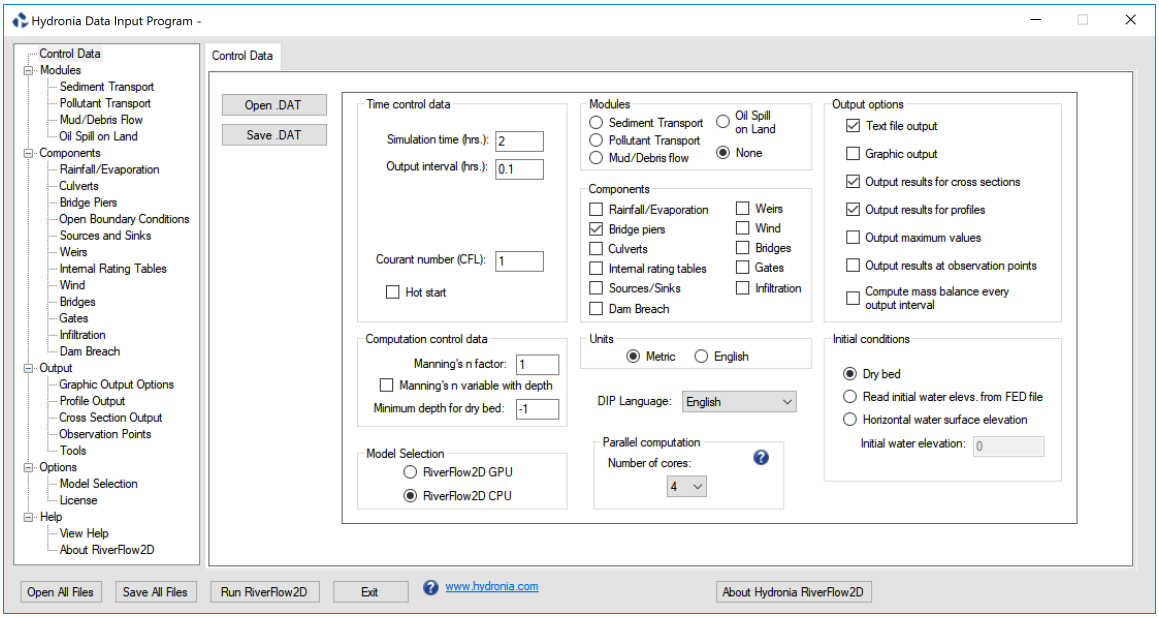


Figure 9.10 – DIP *Control Panel* dialog with the Bridge Piers component selected.

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

9.2.1 Bridge Pier Calculation

The Bridge Pier component can be used when the pier plan area is small compared to the cell 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. :

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

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

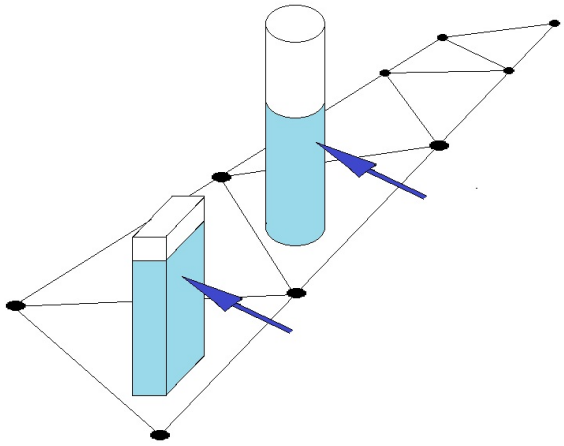


Figure 9.11 – Piers inside cells.

Piers can be circular or rectangular in plan. Rectangular piers are located on cells 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.

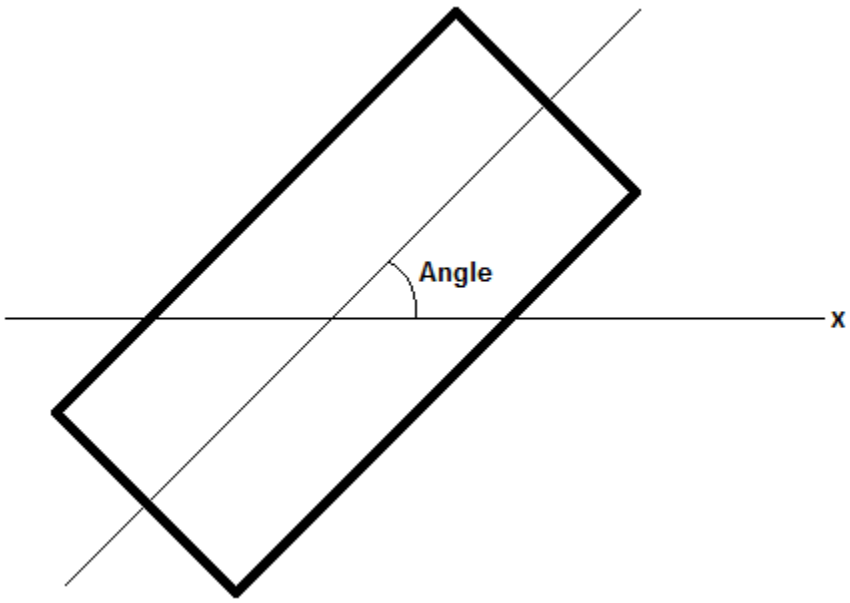


Figure 9.12 – Schematic view of a rectangular pier.

To represent circular piers enter the with and length equal to the pier diameter and the corresponding drag coefficient.

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 the pier exerts on the flow, OilFlow2D converts it to the distributed shear stress on the cell where the pier centroid coordinate is located. The resulting pier shear stress expressions in x and y directions are as follows:

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

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

where A_e is the cell area.

9.3 Culverts Component

The culvert component in OilFlow2D 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 DIP dialog as shown in Figure.

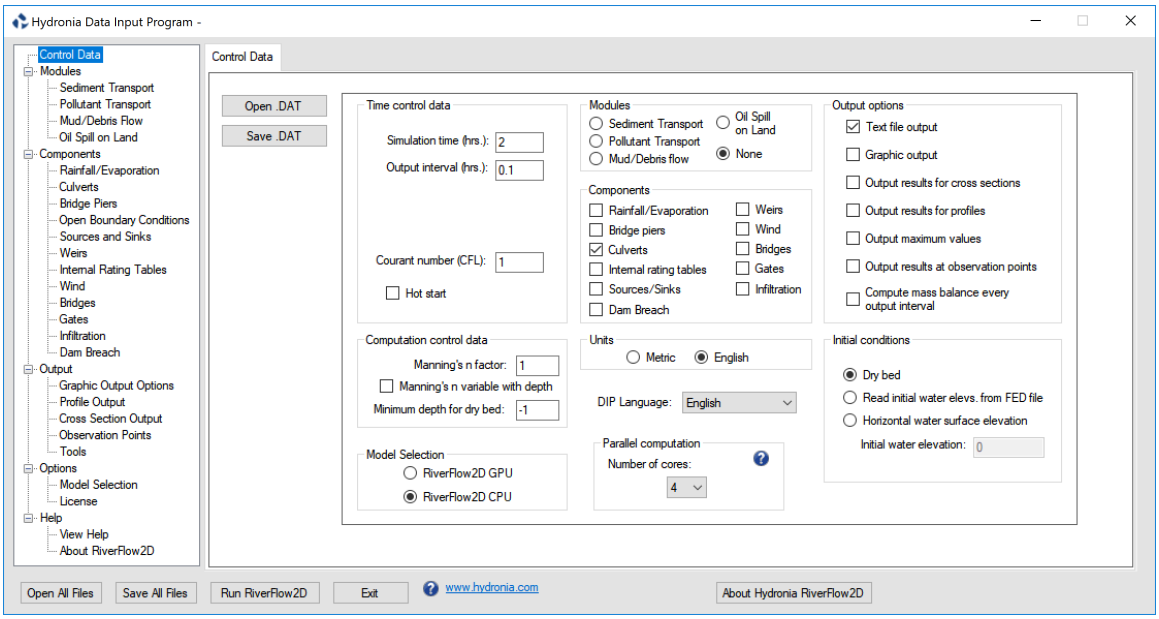


Figure 9.13 – DIP *Global Parameters* dialog with the Culverts Component selected.

There are two options to compute culvert discharge in OilFlow2D. 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.

9.3.1 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 cell and added to the outlet cell assuming instantaneous water volume transmission.

9.3.2 Culvert Calculation using 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:

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

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, $WSEL_h$ 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,

$$C_c = Min \begin{cases} \sqrt{\frac{1-\frac{D_c}{H_h}(Y+mS_0)}{2c'}} \\ \frac{1}{\sqrt{2K'(1/M)}} \left(\frac{H_h}{D_c}\right)^{\left(\frac{1}{M}-0.5\right)} \end{cases}$$

where $H_h = WSEL_h - Z_{bi}$ is the headwater depth. D_c is the culvert diameter for circular culverts and the hight 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:

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

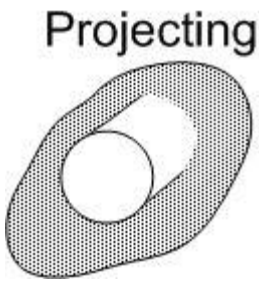
where R_c is the culvert hydraulic radius, K_e is the entrance loss coefficient that can be obtained from Table , n_c is the Manning's n obtained from Table , L_c is the culvert length, and Y, K', M, c' are inlet control coefficients (see Table).

- **Concrete:** Good joints, smooth walls; 0.012
- **Projecting from fill, square-cut end:** 0.015
- **Poor joints, rough walls:** 0.017

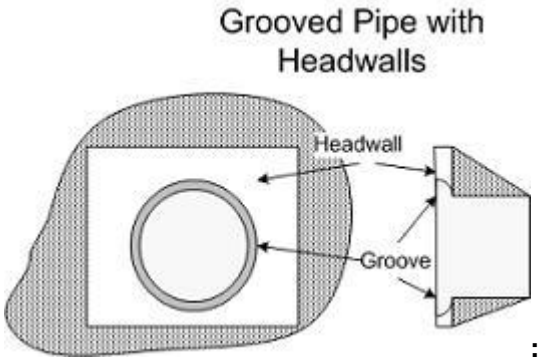
- **Corrugated metal:** 2-2/3 inch × 1/2 inch corrugations; 0.025
- **6 inch × 1 inch corrugations:** 0.024
- **5 inch × 1 inch corrugations:** 0.026
- **3 inch × 1 inch corrugations:** 0.028
- **6 inch × 2 inch corrugations:** 0.034
- **9 inch × 2-1/2 inch corrugations:** 0.035
- **Concrete pipe:** 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
- **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

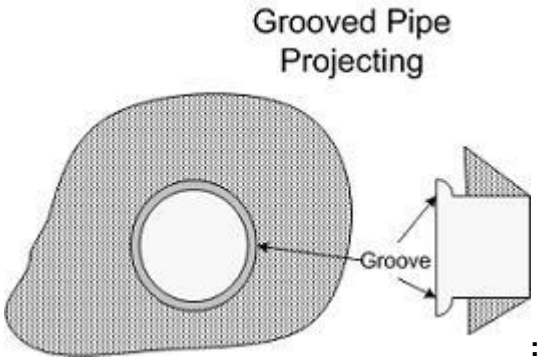
- **Concrete:** Rectangular; Wingwalls; offset 33.7°/flares; beveled top edge; 1.5752; 0.6670; 0.8107; 0.8810
- **Concrete:** Rectangular; Wingwalls; offset 18.4°/flares; top edge bevel; 1.5689; 0.6670; 0.7303; 0.8870
- **Cor. metal:** Rectangular; Headwall; 0.2670; 2.0000; 1.2192; 0.6900
- **Cor. metal:** 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
- **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

- 

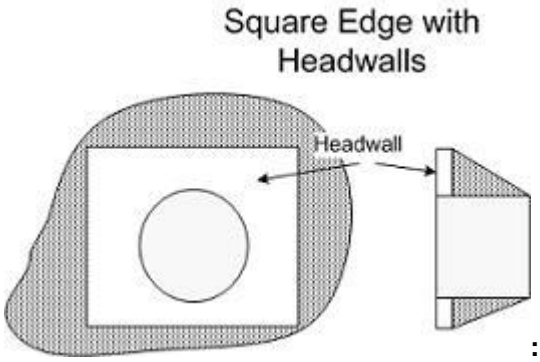
: End of the culvert barrel projects out of the embankment.

- 

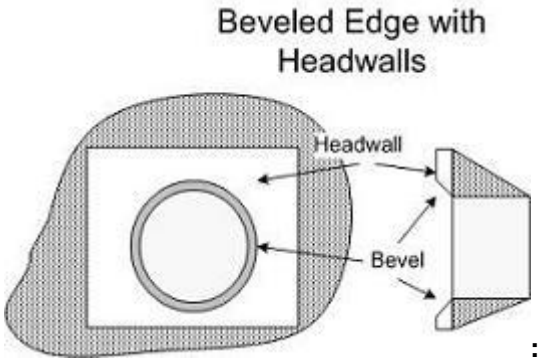
: Grooved pipe for concrete culverts decreases energy losses through the culvert entrance.

- 

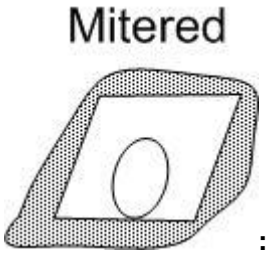
: This option is for concrete pipe culverts.

- 

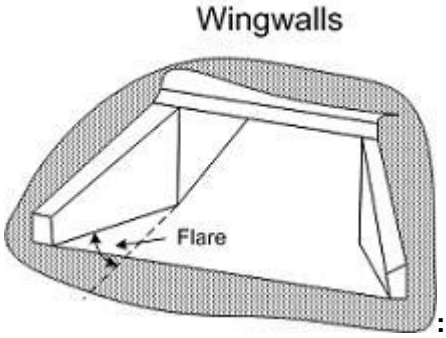
: Square edge with headwall is an entrance condition where the culvert entrance is flush with the headwall.

- 

: '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

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 cell and added to the outlet cell assuming instantaneous water volume transmission.

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

9.3.3 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 cell A to cell 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 CulvertTypes 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.

9.3.4 Culvert Multiple-Cell Volume Exchange Tool in QGIS

The purpose of this option is to increase the number of computational cells involved in the volume exchange between the inlet and outlet. Expanding the selection of cells typically results in a smoother and more stable volume-transfer process effectively reducing oscillations while maintaining accurate computation of exchange discharges.
Using this option, the inlet water elevation is calculated as the average across all wet cells contained within the defined polygon. If the exchange volume for a given time step exceeds the available volume in the inlet cell, the additional volume is drawn from the surrounding cells within the polygon. The primary impact of the new multiple-cell method is the prevention of inlet-cell drying. This significantly reduces the oscillations that commonly occurred when only a single inlet cell was used.
This section describes the tool used to generate and edit polygons for culvert types that rely on multiple cells for volume exchange. The tool automates the creation of inlet and outlet polygons and provides functions to adjust their orientation and size.

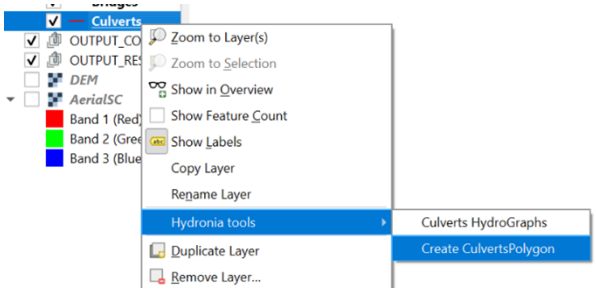
9.3.4.1 Overview

For culverts with BCType = 11, 12, 14, and 15, the modeling framework requires defining inlet and outlet polygons to specify the grid cells involved in the volume-exchange process. To facilitate this, the software includes a dedicated tool that generates a layer called CulvertsPolygon, containing predefined trapezoidal polygons located at each culvert's entrance and exit.
Before using this tool, ensure that the Culverts layer has been added to the project and that all culvert alignments have been properly drawn.

9.3.4.2 Creating the CulvertsPolygon Layer

Follow the steps below to create the CulvertsPolygon layer:

1. In the Layers panel, right-click the Culverts layer.
2. Select Hydronia Tools from the context menu.
3. Click Create CulvertPolygon.



After completion, a new layer named CulvertsPolygon will be added to the project. This layer automatically contains a pair of trapezoidal polygons - one at the inlet and one at the outlet of each culvert. These polygons represent the areas used for identifying neighboring cells.

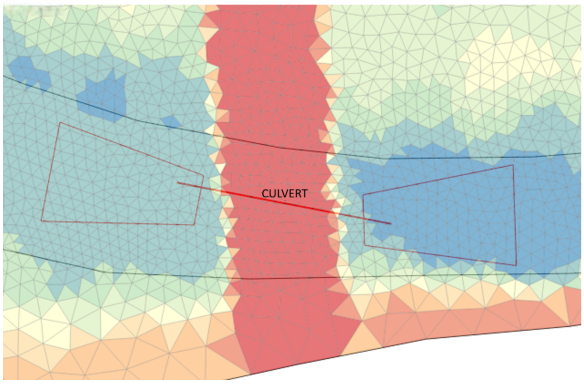


Figure 9.14 – Inlet and outlet culvert polygons.

9.3.4.3 Editing the CulvertsPolygon Layer

The polygons generated by the tool may require adjustment to better match the culvert geometry or local topography. The following tools and procedures are available for polygon editing.

Adjusting Polygon Orientation and Position

To modify the alignment of the polygons:

1. Activate the Advanced Digitizing toolbar.
2. Switch the CulvertsPolygon layer to editing mode.
3. Use the following tools to make the necessary adjustments.
    - Rotate Object: Rotates the polygon around its centroid or a selected reference point.
    - Move Object: Translates the polygon to a new position without altering its shape.
    - To adjust the polygons' vertices, activate the vertex tool 

These tools allow you to align the polygons with the culvert axis accurately.

Adjusting Polygon Size

To modify the dimensions of the polygons:

1. Activate the Node Tool.

2. Select the polygon to be edited.
3. Drag the individual nodes or vertices to reshape or resize the polygon.

Adjusting polygon size ensures proper coverage of the neighboring cells used for flow exchange.

9.3.4.4 Notes and Recommendations

- Always verify that polygons align correctly with the culvert and fully cover the intended computational cells.
- After editing, remember to save the layer's edits before proceeding with the setup.
- Avoid excessive stretching or distortion of polygons, as this may affect model behavior.

9.4 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). Figure shows the flow modes that can be calculated through gates that include submergence and overtopping.

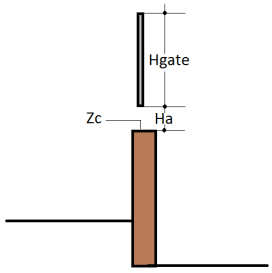


Figure 9.15 – Schematic cut view perpendicular to a gate structure.

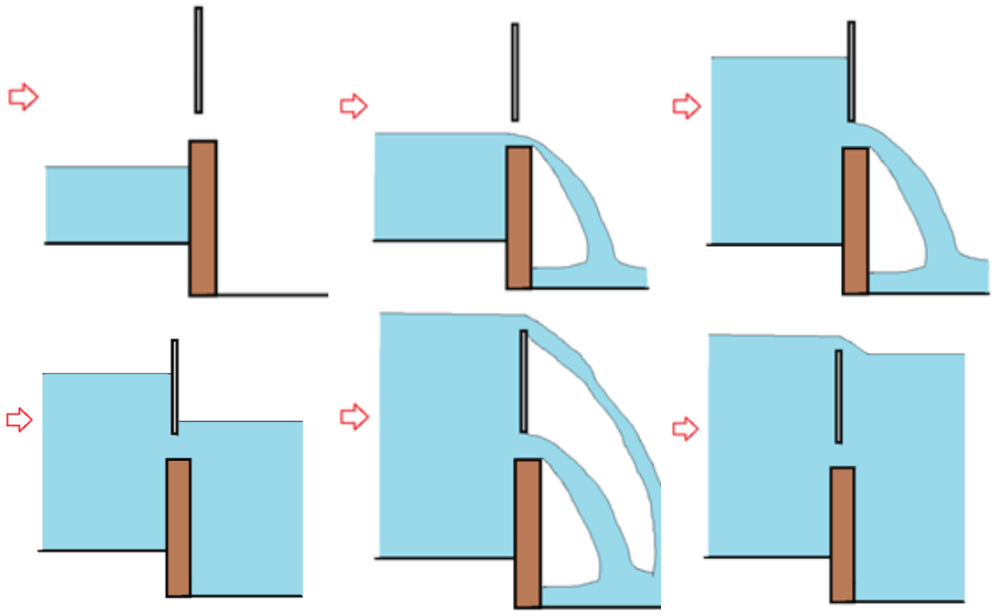


Figure 9.16 – Flow modes across gates.

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

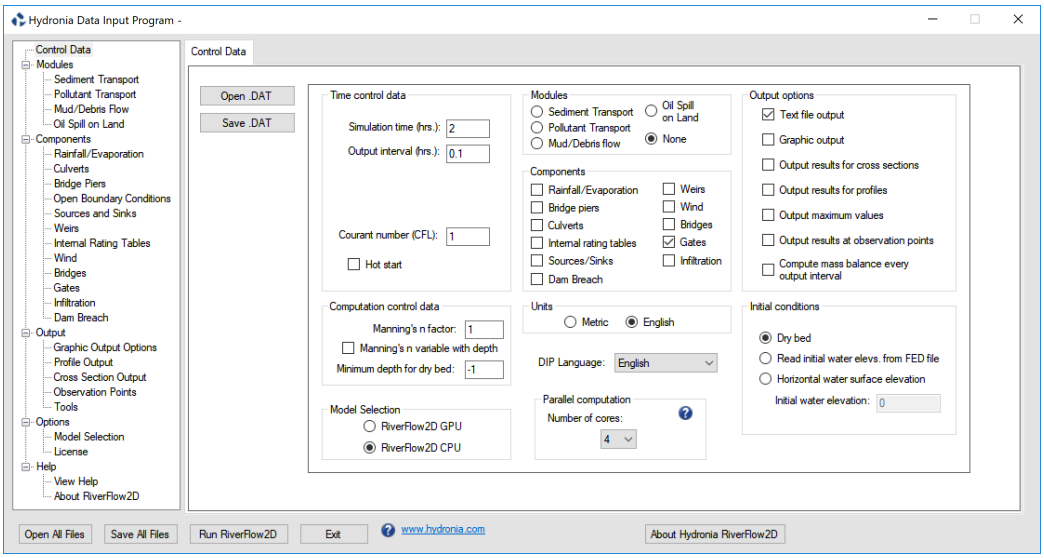


Figure 9.17 – DIP *Control Data* panel with the Gates Component selected.

The gate plan data is entered in the QGIS Gates layer.

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.

Note

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

9.4.1 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 , the discharge is given by

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

with K_1 an energy loss coefficient. In OilFlow2D K_1=3.33 .
When $G_o + max(z_l, z_r) > min(d_l, d_r)$, Figure , the discharge is given by

$$q = C_d (2/3) \sqrt{2g} \, [d_r - max(z_l, z_r)]^{1/2}$$

with C_d being the non-dimensional discharge coefficient that takes values around 0.6.

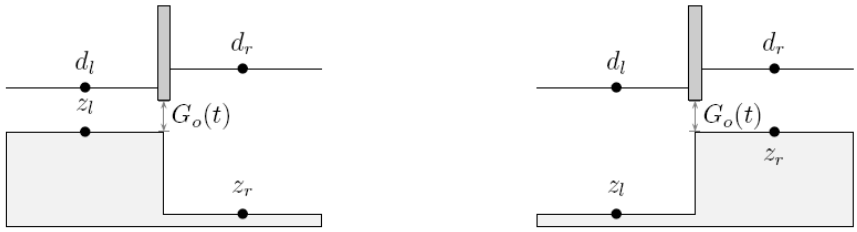


Figure 9.18 – Water levels for discharge under a gate in submerged conditions formulated as in (G1).

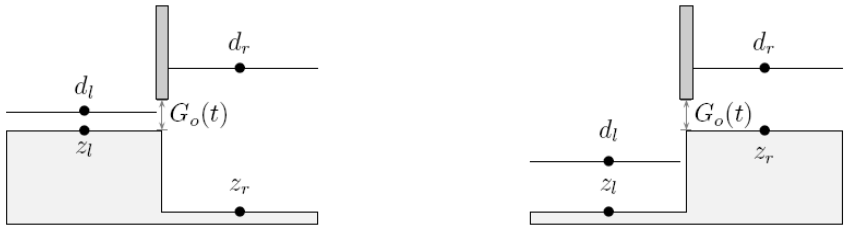


Figure 9.19 – Water levels for discharge under a gate in submerged conditions formulated as in (G2).

9.5 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 OilFlow2D, infiltration is treated as a loss. This process is mainly governed by two forces: gravity and capillarity action. The model provides three methods to compute the infiltration losses: Horton, Green:** Ampt and SCS-Curve Number (SCS-CN).

> **Note**
>
> When using the infiltration component without rainfall, only the Horton or the Green and Amp methods can be used since they take into account the existing surface water to determine the infiltration rates regardless if there is no precipitation. The SCS-CN method calculates infiltration as a function of the given rainfall and does not consider the ponded water.

9.5.1 Horton Infiltration Model

Horton's infiltration model suggests an exponential equation for modeling the soil infiltration capacity f_p:

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

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 and summarized in. Table and Table show the parameters for some general types of soil, as presented in. Note that no *k* values are shown. A value of $k = 4.14\ hr^{-1}$ is recommended in the absence of any field data.

- **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
- **Clay loam, silty clay loams:** 1.27
- **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 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:

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

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:

$$R \leq f_p \Rightarrow f = R \qquad\qquad R > f_p, t > t_p \Rightarrow f = f_p$$

where t_p represents the ponding time.
Following , 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 :

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

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 at t_p:

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

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:

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

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 behavior is that the soil cannot infiltrate more than the rainfall rate, so a limit in the recalculated infiltration capacity must be imposed:

$$f_p \leq R$$

9.5.2 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 , as presented in.

- **Sand:** 0.437(0.374-0.500); 4.95(0.97-25.36); 7.8
- **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:

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

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 provides the cumulative infiltration:

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

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

Equations and assume that the soil is ponded from the beginning. Additional considerations should be taken into account in order to model an unsteady storm pattern. Three possibilities can occur in every time step: 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 :

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

The result from eq. 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. Moreover, the real infiltration *f* rate will

be equal to the potential one $f_p \leq i$:

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

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:

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

From equations and 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 :

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

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

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

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

9.5.3 SCS-CN 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. 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.

Let us define the concepts of runoff or effective precipitation RO, rainfall volume RV, initial water abstraction which infiltrates before runoff begins I_a and the potential maximum retention S. Hence, the potential runoff can be calculated as $RV - I_a$. The main hypothesis of SCS-CN method is assuming equal relations between the real quantities and the potential quantities, as follows:

$$\frac{F}{S} = \frac{RO}{RV - I_a}$$

On the other hand, the water mass balance on the catchment lead us to:

$$RV = RO + F + I_a$$

By combining and and taking into consideration that the runoff cannot begin until the initial abstraction has been met:

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

The potential maximum retention S is estimated (in mm) by means of the Curve Number:

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

The initial abstraction is assumed proportional to S:

$$I_a = \alpha S$$

where traditionally $\alpha = 0.2$ for every watersheds (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. This parameter can be changed in OilFlow2D, and its influence in water runoff was studied in Caviedes et al..

To determine appropriate Curve Numbers we recommend following the guidelines provided in.

It is important to remark that SCS-CN method was not designed to consider time. Following , 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.

The SCS-CN method can be extended in order to estimate the temporal distribution of the water losses. By combining again and but solving for F:

$$F = \frac{S\,(RV - I_a)}{RV - I_a + S}, \qquad RV \geq I_a$$

By differentiating , taking into account that I_a and S are constant magnitudes, the following expression for the infiltration rate is obtained :

$$f = \frac{dF}{dt} = \frac{S^2 R}{RV - I_a + S}$$

being R the rainfall rate, defined as follows:

$$R = \frac{dRV}{dt}$$

9.5.3.1 Antecedent Moisture Conditions

In the SCS-CN method you can consider the Antecedent Moisture Content (AMC), that represents the preceding relative moisture of the soil prior to the storm event and its influence in the water runoff. This parameter allows accounting for the CN variation for different storm events, and the initial soil moisture for a given event. Three possible assumptions can be considered: dry conditions (AMC I), average conditions (AMC II) or wet conditions (AMC III) as summarized in Table.

- **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 , the Curve Number for dry or wet conditions has been recalculated in terms of the standard conditions according to Eqs. and :

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

$$CN(III) = \frac{23CN(II)}{10 + 0.13CN(II)}$$

On the other hand, some newer references recommend to use an empirical data table to compute both values.

9.6 Wind Component

The wind stress is added to the momentum equations source term vector as follows:

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

where

$$S_{wx} = C_d \frac{\rho_a}{\rho_w} u|U| \qquad S_{wy} = C_d \frac{\rho_a}{\rho_w} v|U|$$

being $U = (u, v)$ the wind velocity vector, ρ_a and ρ_w the air and water densities respectively, and C_d is the coefficient of aerodynamic resistance at a height of 10 m above the water level.
The model considers C_d constant but typically it increases with the wind velocity. Garrat (1977) suggested the following formula to determine C_d

$$C_d = (0.75 + 0.067U)\, 10^{-3}$$

where U is given in m/s. For wind velocities varying from 1 to 25 m/s C_d would be between 0.0008 and 0.0024 approximately. Powell, 2008, suggested that the C_d range in shallow water is 0.00095 to 0.00157, with values that could reach 0.0045 for severe storm events. Note that typically the wind velocity is obtained in angle/magnitude (ϕ, U) format, and the meteorological convention is to provide the wind direction from which wind is blowing from in clockwise sense. In this convention a north wind would have an angle of 0 degrees while an east wind is 90 degrees and so forth. To compute the (u, v) wind velocity vector components based on (ϕ, U) you can apply these formulas:

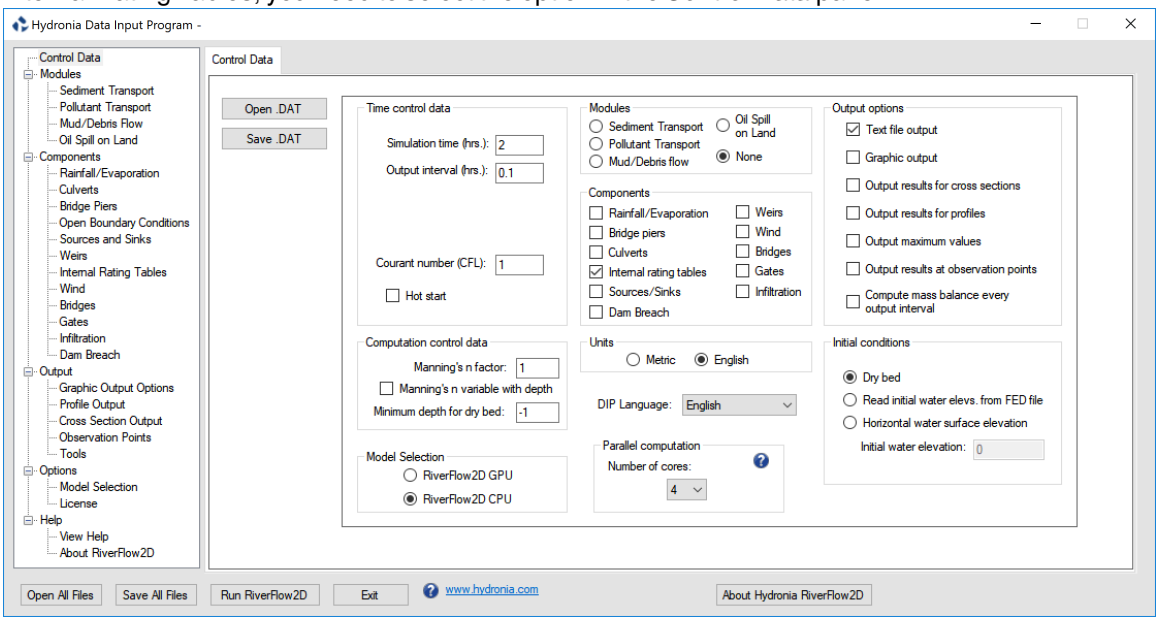
$$u = -Usin(\phi) v = -Ucos(\phi) \text{with} U = \sqrt{(u^2 + v^2)}$$

9.7 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.

> **Note**
>
> 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 DIP shown in Figure.



Internal Rating Table (IRT) plan data is entered in the QGIS Internal Rating Table Layer.

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

9.7.1 Internal Rating Table Calculations

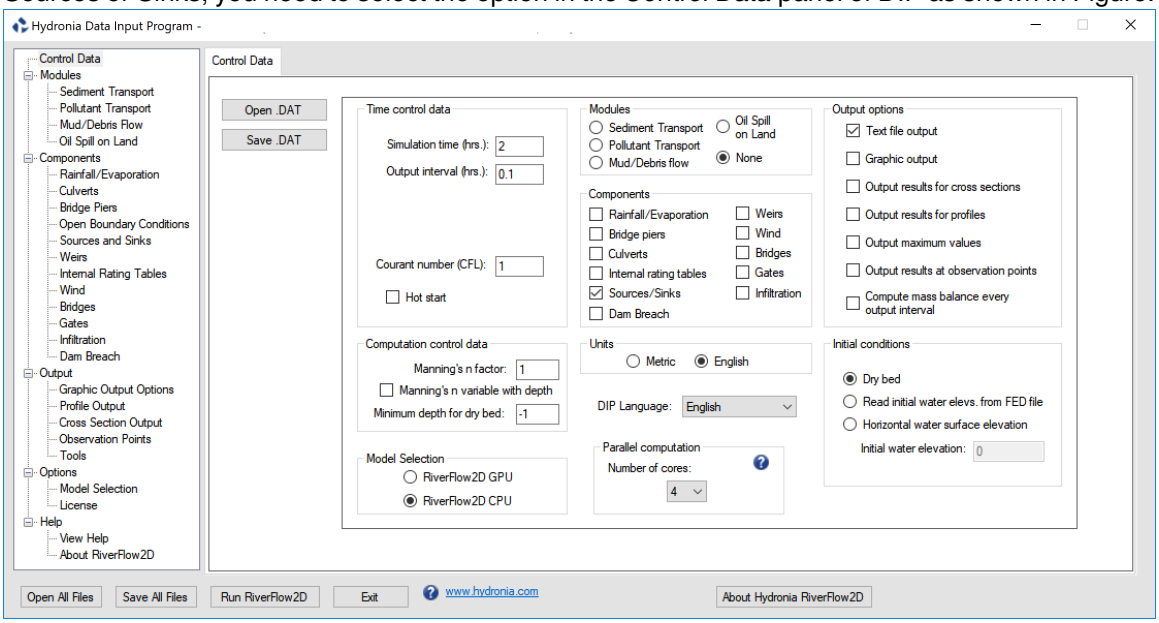
An internal rating table is implemented as a set of values of total discharge in terms of the water surface level $Q = Q(h + z)$. This table is defined along a polyline in the mesh. First, a common average water surface level is computed considering all the upstream cells along the polyline. Then, the 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.

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

9.7.2 Assumptions of Internal Rating Table Calculations

The rating table does not account for outlet control. ## 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 DIP as shown in Figure.

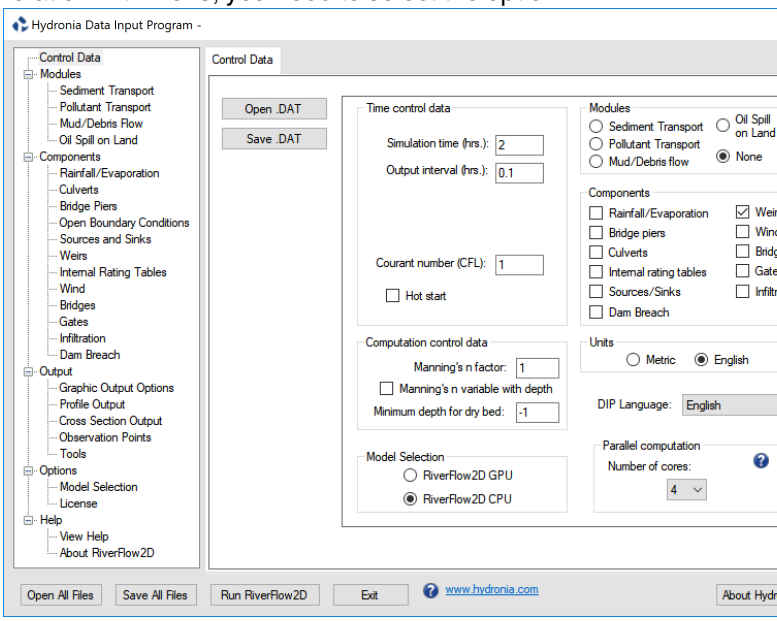


Sources and Sinks data is entered in OilFlow2D *Sources* layer.

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

9.8 Weirs

OilFlow2D *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 DIP as shown in Figure.



Weir plan data is entered in the QGIS *Weirs* layer. Since OilFlow2D requires that the weir passes through nodes, it is essential that mesh generation engine creates nodes along the weir polyline. To achieve this always remember to re-mesh after changing any weir aliment in the *Weirs* layer.

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

9.8.1 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:

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

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, $MAX\left(d_L, d_R\right) \leq d_w$, 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 $H_w + max\left(z_l, z_r\right) < min\left(d_l, d_r\right)$, the discharge is given by

$$q = C_d \frac{2}{3} \sqrt{2g} \left(d_r - d_l\right)^{3/2}$$

with C_d the non-dimensional weir discharge coefficient that takes values between 0.611 and 1.1. When $H_w + max\left(z_l, z_r\right) > min\left(d_l, d_r\right)$, the discharge is given by

$$q = C_d \frac{2}{3} \sqrt{2g} \left(d_r - H_w\right)^{3/2}$$

9.8.2 Assumptions of Weir Calculations

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

9.9 Dam Breach Modeling

OilFlow2D Dam Breach component provides a way to simulate a gradual breach of internal linear obstructions such as dams, levees, etc. Three methods are provided: Prescribed breach, breach formation by erosion over-topping and breach formation by piping.

The dam 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 pounding upstream.

9.9.1 Prescribed dam breach

In OilFlow2D, 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):

- 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.

Particular cases include $b(t) = 0$ that reduces the breach to a triangular weir, and $\alpha = 0$ represents a rectangular breach.

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

$$Q_b = C_d(2/3)\sqrt{2g}H^{3/2}$$

where $H = h + z - H_v$, $H_v = z_{crest} - H_b$, $B(H) = b(t) + 2\frac{H(t)}{tg\alpha}$, and C_d is the non-dimensional discharge coefficient that takes values between 0.611 and 1.1.

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

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

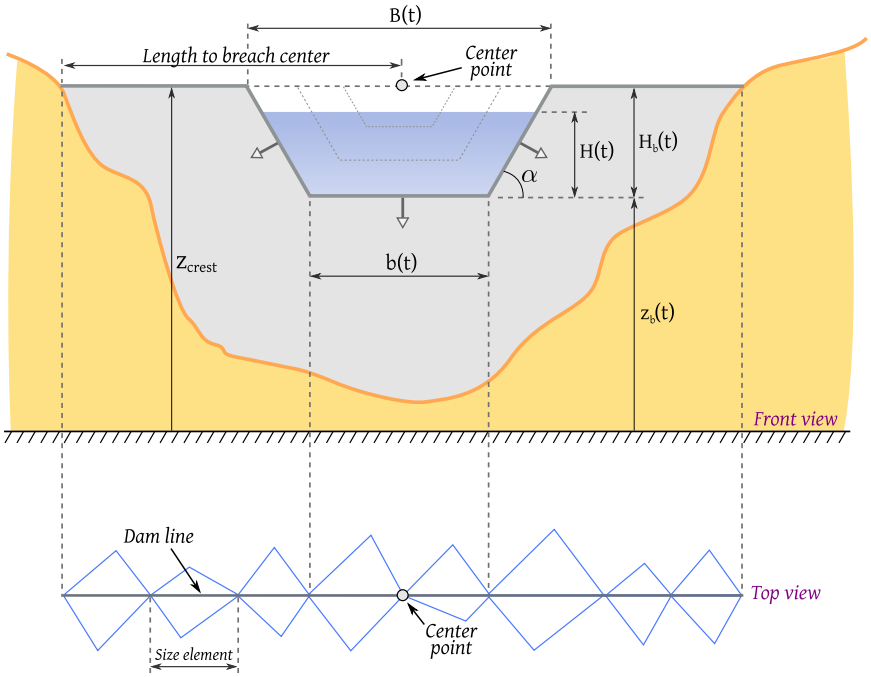


Figure 9.20 – Schematic dimensions for prescribed dam breach failure mode.

9.9.2 Dam breach failure by piping erosion

The dam breach evolution due to piping erosion assumes that the initial pipe channel cross-section is considered as an arch with a rectangular base at the bottom and a semicircle on top (See Figure). Orifice and open channel flow equations are used to compute the discharge for pressure and free surface flows, respectively. A shear stress-based formula is adopted to calculate the erosion rate. The arched pipe tunnel is assumed to enlarge along its width until the overlying soil can not maintain stability.

The pipe roof collapse is determined by comparing the overlying soil weight and the soil cohesion of the two sidewalls of the pipe. The failure planes are assumed to be vertical and, for the sake of simplicity, the collapse is assumed to move downstream instantaneously.

Cross-section of the expansion due to piping process before the dam collapse (left) and trapezoidal breach evolution after the dam collapse (right).

9.10 Flow discharge through the piping cross section

If the pipe is fully filled with water, the discharge through can be estimated using the following orifice equation:

$$Q_b = A\sqrt{\frac{2g\,(z_s - z_b)}{1 + fL/(4R)}}$$

where:

- $A = b^2 + \frac{1}{8}\pi b^2$ is the cross-sectional area of the pipe,
- b is the width of the base of the pipe,

- $z_{pb} = z_b + \frac{1}{2}b$ is the elevation of the pipe center line,
- z_b is the elevation of the pipe bottom,
- $z_s = h + z_{bed}$ is the elevation of the water surface level,
- L is the pipe length,
- $R = A/P$ is the pipe hydraulic radius
- $P = (3 + 0.5\pi)b$ is the pipe wet perimeter,
- $f = 8gn^2R^{-1/3}$ is the Darcy-Weisbach friction factor of the pipe surface,
- $n = \frac{1}{A_n}d_{50}^{1/6}$ is the Manning's roughness coefficient of the pipe surface, and
- $A_n = 12$ is a constant.

On the other hand, in case of partially filled pipe, roof collapse or overtopping, the discharge is computed by means of a free surface flow equation:

$$Q_b = k_{sm}\left(c_r b\,(z_s - z_b)^{1.5} + \frac{c_t(z_s - z_b)^{2.5}}{\tan\beta}\right)$$

being:

- k_{sm} is a dimensionless submergence correction for tail water effects,
- β is the breach side slope angle with respect to the horizontal,
- $c_r = 1.7\sqrt{m}/s$ is a discharge coefficient for the rectangular part of the breach section, and
- $c_t = 1.2\sqrt{m}/s$ is a discharge coefficient for the triangular part of the breach section.

9.11 Pipe erosion

Regarding the erosion process, a shear stress-based formula is adopted to calculate the erosion rate. As erosion takes place, the full pipe cross section is enlarged along its height and width due to the removal of materials until the collapse of the pipe roof occurs (see Figure). The vertical erosion of the pipe is computed using an excess detachment rate relation:

$$\frac{dz_b}{dt} = k_d(\tau_e - \tau_c)$$

where:

- k_d is the measured erosion coefficient at the breach,
- τ_c is the critical stress required to initiate detachment for the material,
- $\tau_e = \frac{\rho_w g n^2 Q_b^2}{A^2 R^{1/3}}$ is the bed shear stress in the pipe surface, and
- ρ_w is the water density.

The horizontal erosion rate is assumed to be equal to the vertical erosion rate and hence the evolution can be expressed as:

$$\frac{db}{dt} = 2k_d(\tau_e - \tau_c)$$

The collapse of the pipe roof is estimated by comparing the weight of the overlying soil and the cohesion of the soil on the two sidewalls of the pipe. The failure planes are assumed to be vertical and, for the sake of simplicity, the collapse is assumed to move downstream instantaneously. The arch finally becomes unstable due to the erosion of the pipe and the collapse of the soil mass above

the arch occurs. The failure of the roof occurs if the top of the eroding pipe ($z_b + 1.5b$) reaches the top of the dam (z_{crest}). On the other hand, the failure also occurs if the driving force F_d exceeds the soil resistant force F_r (Figure). Hence, the model compares these two forces along the vertical direction. Once the driving force (equal to the weight of the failure part) is larger than the resistant force, the roof above the pipe will collapse:

$$F_d = \rho_w g\big[p + G_s(1-p)\big]\left(A_a\, b - A_c \frac{L_2 + L_3}{2}\right) + \rho_w g G_s (1-p) A_b\, b$$

$$F_r = 2C\,(A_a + A_b)$$

where p is the porosity of the soil material, $G_s = \rho_d/\rho_w$ specific gravity of the soil, ρ_d soil material density and C cohesion of the dam fillings. The areas A_a, A_b and A_c are computed as follows:

$$A_a = \frac{L_2 + L_3}{2}\,(z_s - (z_b + b))$$

$$A_b = \frac{L_1 + L_2}{2}\,(z_{crest} - z_s)$$

$$A_c = \frac{1}{8}\pi b^2$$

being:

- L_1 the dam crest width,
- $L_2 = L_1 + \frac{z_{crest} - z_s}{\tan \alpha_{uw}} + \frac{z_{crest} - z_s}{\tan \alpha_{dw}}$,
- $L_3 = L_2 + \frac{z_s - (z_b + b)}{\tan \alpha_{uw}} + \frac{z_s - (z_b + b)}{\tan \alpha_{dw}}$,
- α_{uw} the upward dam slope angle, and
- α_{dw} the downward dam slope angle.

Schematic diagram of the piping situation.

The collapsed pipe roof is assumed to move downstream instantaneously. After the collapse, overtopping failure dominates and the breach flow discharge and the vertical erosion can be estimated using Eqs. and , respectively. The relationship between horizontal expansion and vertical undercutting is given by the change in the breach top width ΔB:

$$\Delta B = \frac{2\Delta z_b}{\sin \beta}$$

and the change in the breach bottom width Δb is given by:

$$\Delta b = 2\Delta z_b \left(\frac{1}{\sin \beta} - \frac{1}{\tan \beta}\right)$$

9.12 Overtopping erosion

The other type of erosion considered is the overtopping erosion, in which the breach already exists at the top of the dam and widens as the flow of water circulates through it (see Figure). The erosion process causes the width and height of the trapezoidal breach to evolve according to the same mathematical expressions as in the previous section, when the dam roof collapses:

- Breach depth evolution:

$$\frac{dz_b}{dt} = k_d(\tau_e - \tau_c)$$

- Breach width evolution:

$$\frac{db}{dt} = 2k_d(\tau_e - \tau_c)$$

- Change in the breach top width:

$$\Delta B = \frac{2\Delta z_b}{\sin\beta}$$

- Change in the breach bottom width:

$$\Delta b = 2\Delta z_b \left(\frac{1}{\sin\beta} - \frac{1}{\tan\beta}\right)$$

Trapezoidal breach evolution for the overtopping erosion case.

9.13 Dambreach flow as internal boundary condition

When using triangular meshes, there are three contributions to update the three conserved variables. At boundary cells at least one of the edges does not have a neighbor cell and boundary conditions are necessary to complete the information supplied by the numerical scheme. The presence of an internal hydraulic structure can be modeled by means of a mathematical condition defined along an interior line in the computational domain. This is called internal boundary condition (IBC), i.e. each pair of cells sharing an edge on that internal line are considered internal boundary cells. These cells are updated using both information from the numerical scheme and from the IBC. Figure shows an example of IBC defined along an internal boundary line where several pairs of cells (filled with blue) on both sides of the line share an edge. An external law is used to define the module of the discharge through them while the water depth is provided by the numerical scheme.

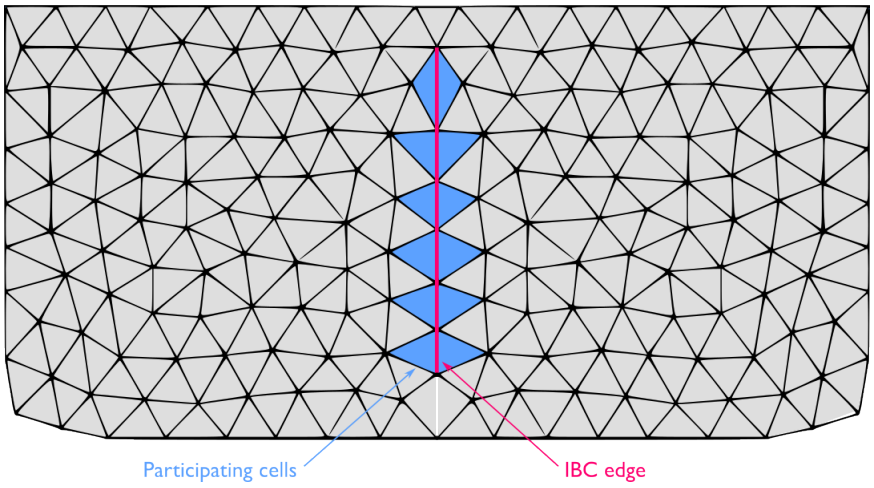


Figure 9.21 – Internal boundary cells.

Assuming the pipe roof collapse has not occurred and the flow direction from left to right, the upstream element is cell L and the downstream is cell R. Water surface elevation levels at the left side $z_{s,L}^{n+1} = (h+z)_L^{n+1}$, provided by the numerical scheme
are used to evaluate the discharge by means of the external discharge expression. First, the change of the pipe bottom elevation and width due to erosion are computed using and as:

$$z_b^{n+1} = z_b^n + \Delta t\, k_d\, (\tau_e - \tau_c)_b^n$$
$$b^{n+1} = b^n + \Delta t\, 2\, k_d\, (\tau_e - \tau_c)_b^n$$

where $(\tau_e - \tau_c)_b^n$ accounts for the erosive shear stress at the breach evaluated at the time level t^n. Then, the driving and the soil resistant forces, F_d and F_r respectively, are computed using and the pipe roof collapse condition is checked. Therefore, two cases must be taken into consideration:

1. If the roof collapse does not occurred, the pipe is considered filled with water and pressurized so that the enforced cell discharge in pipe during the next time level is computed using as:

$$Q_b^{n+1} = A\sqrt{\frac{2g(z_{s,L}^{n+1} - z_b^{n+1})}{1 + fL/(4R)}}$$

   where the integrated breach features A, f, L, R are evaluated at time t^{n+1}.

2. If the roof collapse condition is satisfied, the dam breach is assumed open and the enforced cell discharge in pipe during the next time level is computed using as:

$$Q_b^{n+1} = k_{sm}\left(c_r b^{n+1}(z_{s,L}^{n+1} - z_b^{n+1})^{1.5} + \frac{c_t(z_{s,L}^{n+1} - z_b^{n+1})^{2.5}}{\tan\beta}\right)$$

Note that once the roof collapse occurs, the pipe stability condition has not to be checked anymore and the dam breach is assumed open. Regardless the pipe is maintained or collapsed, the unit discharge at each cell pair composing the dam breach is assumed normal to the direction of the shared edge $\hat{\mathbf{n}}_\mathbf{b}$ and its module is updated as:

$$q_L^{n+1} = q_R^{n+1} = \frac{l}{W_b} Q_b^{n+1}$$

being W_b the total breach width and l the length of the shared edge for each internal boundary cell pair.

10

Hydronia Data Input Program (DIP)

Hydronia Data Input Program (DIP) user interface provides the tools to enter non-spatial data, and run OilFlow2D. All changes introduced on the DIP will be saved to the native OilFlow2D data files.

The DIP appears when you export the files to OilFlow2D from QGIS. You can also access the DIP double-clicking on the Hydronia Data Input Program 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:

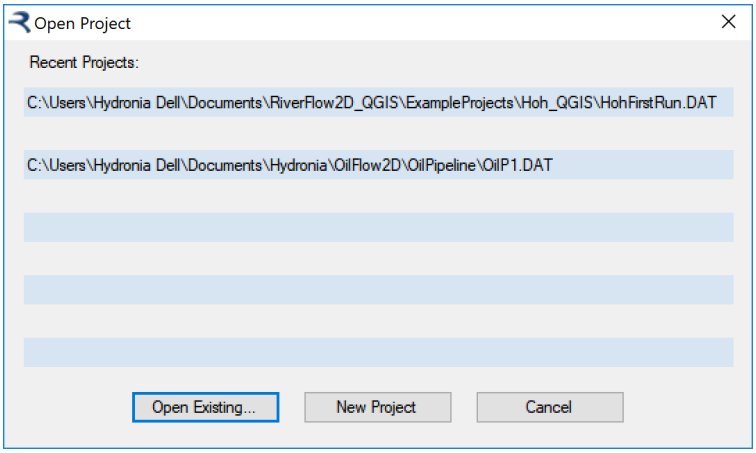


Figure 10.1 – DIP Open Project Dialog.

DIP provides a data input environment with panels that include all the non-spatial options to run OilFlow2D. The left column on the main window allows you select modules, components, output options, etc. When you click on one of the cells, the appropriate right side panel is activated. Each panel contains the data corresponding to each of OilFlow2D data files. For example, the *Control Data* Panel has all the data of the file.

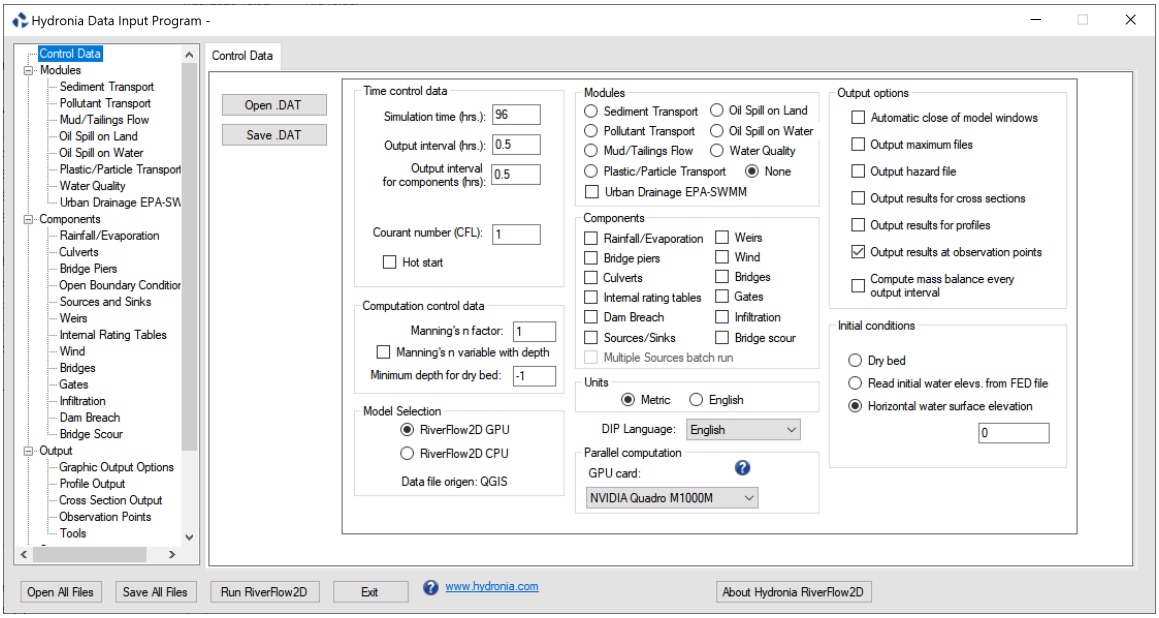


Figure 10.2 – Main DIP window.

DIP lets you select different model engines. Using the options the Model Selection frame you can select between OilFlow2D CPU or OilFlow2D GPU. Note that to run the GPU version you need the appropriate GPU hardware. Please contact Hydronia at mailto:support@hydronia.com to inquire about the currently supported NVIDIA GPU cards.

The following sections describe the panel dialogs of the DIP.

10.1 Control Data Panel (.DAT file)

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 OilFlow2D model. The program will launch with the *Control Data* panel visible.

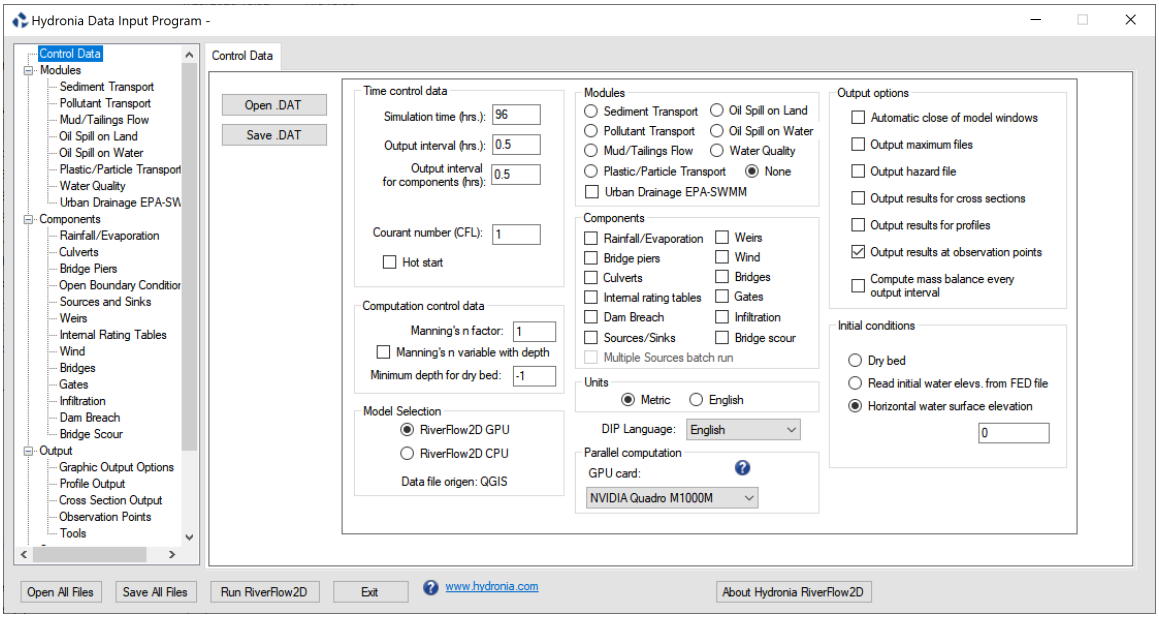


Figure 10.3 – *Control Data* Panel.

- **Open .DAT:** Opens an existing file.
- **Save .DAT:** Saves a file with the data shown on the 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 file*.
- **Run OilFlow2D:** Runs OilFlow2D .
- **Exit:** Closes DIP.
- **About DIP:** Shows a concise description of OilFlow2D.
- **www.hydronia.com:** Opens Hydronia home page.
- **DIP Language:** Drop down list that allows selecting the language of the Data Input Program user interface. Presently the options available are English and Spanish. Other language will be added in future releases.
- **Simulation time (h.):** Total simulation time in hours.
- **Output Interval (h.):** Time interval for output reporting.
- **Output Interval for components (h.):** Time interval for components output reporting. It applies to cross sections, profiles, observation points, culverts, weirs, dam breach, bridges, and gates.
- **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 to avoid model oscillations in the model results.

- **Hot start:** Use this option to restart the model from a previously simulation.
- **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**
> When exporting OilFlow2D files from QGIS, units are automatically set according to the selected Projection. Changing to units different from those of the projection should not be attempted since it will certainly lead to incorrect model results.

- **Manning's n factor:** Use the XNMAN factor to test the sensitivity of results to Manning's n and reduce the number of calibration runs. Using this option, will each cell Manning's n-value will be multiplied by XNMAN. 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.
- **Automatic close of model windows:** The model windows are automatically closed as soon as the program finalizes the execution.
- **Output maximum files:** Switch to allow reporting maximum values throughout the simulation to , , and maximum values output files.
- **Output hazard file:** The model will generate flood hazard levels based on the criteria used in different countries .
- **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 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 file.
- **Output results at observation points:** Switch to allow reporting time series of results at specified locations defined by coordinates in the Observation Points Panel.
- **Compute mass balance every output interval:** Switch to calculate detailed mass or volume balance. The report is written in the file. Keep this option selected to check general model mass balance, but it is recommended to turn it off for production runs, since it will speed up the model operation.

- **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 file.
- **Oil Spill on Water:** Option to activate the Oil Spill on Water modeling. The data can be edited in the Oil Spill on Water Panel. The data is written to the 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 file.- **Urban Drainage EPA-SWMM:** Switch to integrate surface water with storm drain EPA-SWMM model. The data is written to file.
- **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 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 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 file.
- **Bridge Scour:** Switch to calculate bridge scour at piers and abutments. The data can be edited in the Bridge Scour Panel. The data is written to 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 file.
- **Dam Breach:** Switch to activate the Dam Breach component. The data is written to file.
- **Gates:** Switch to model gates. The data can be edited in the *Gates* Panel. The data is written to file.
- **Infiltration:** Option to activate Infiltration loss calculations. The required data has to be entered in the Infiltration Panel. This data is written to 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 file.
- **Multiple sources batch run:** Switch to activate batch runs when more than one source is defined. The model will perform sequentially runs for each individual source in separate directories, that will have the name given to each source.
- **Sources/Sinks:** Switch to indicate existence of sources or sinks. The sources/sinks data can be edited in the Sources/Sinks Panel. The data is written to file.
- **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 file.
- **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 file:** Initial water surface elevations will be read from the 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.

10.2 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.

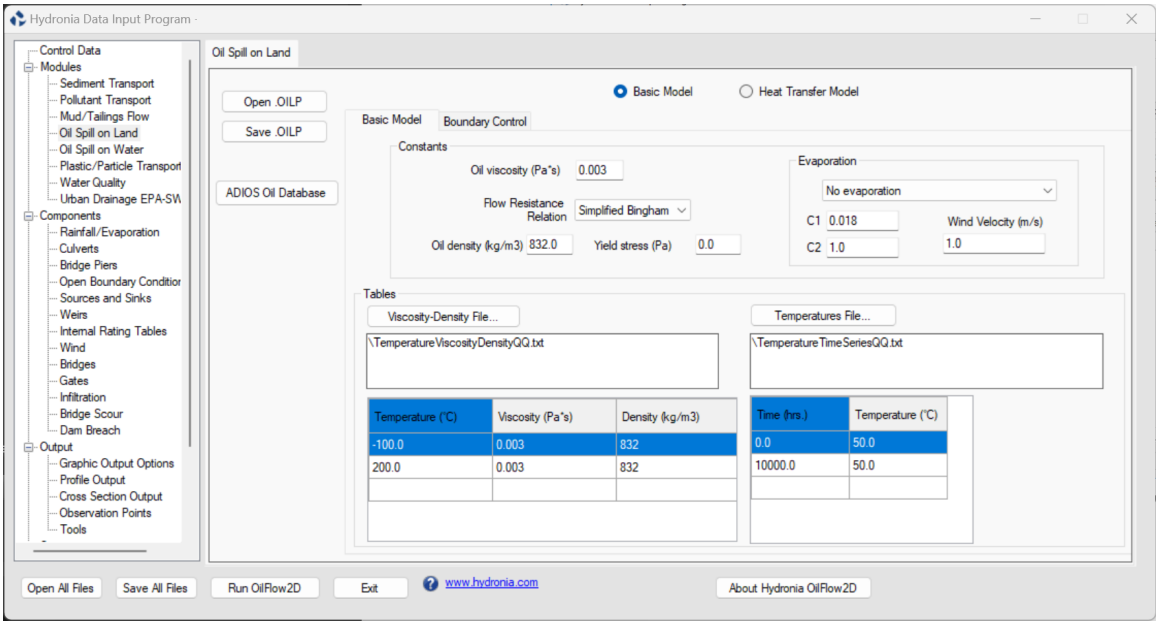


Figure 10.4 – Oil Spill on Land Basic Model Panel (OilFlow2D model).

Flow resistance relation

1. Full Bingham
2. Simplified Bingham
3. Quadratic

- **Oil Viscosity:** Fluid viscosity (Poise or lb/in^2).
- **Oil density:** Fluid density (kg/m^3 or lb/ft^3).
- **Temperature:** Ambient temperature (°C or °F).
- **Yield stress:** Yield stress (Pa or lb/in^2).

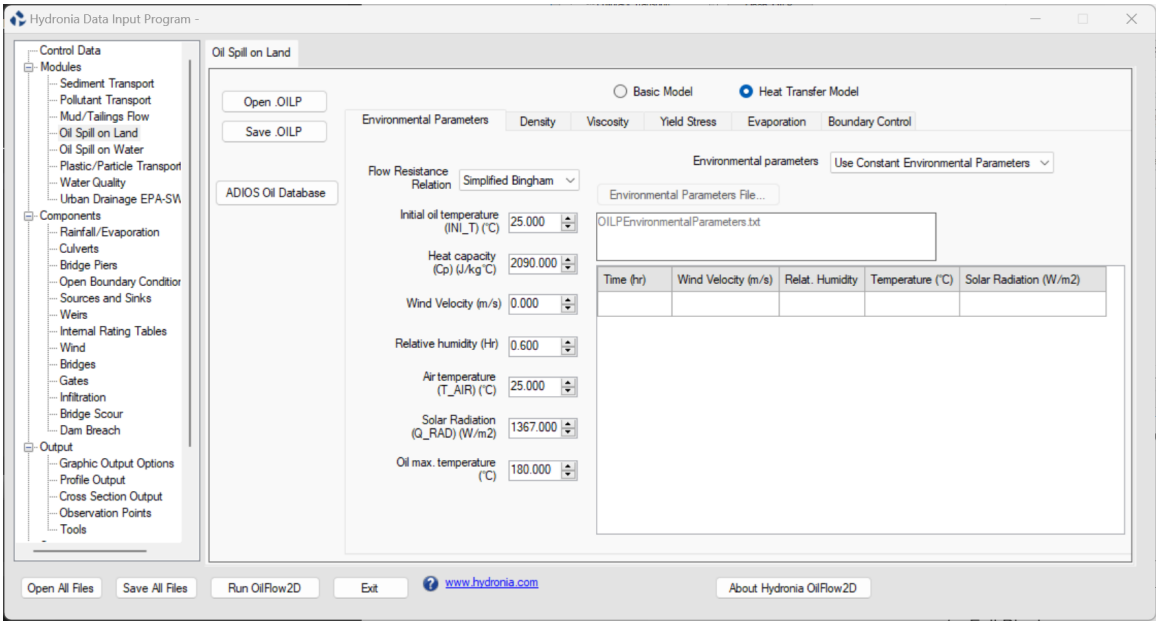


Figure 10.5 – Oil Spill on Land Heat Transfer Model Panel. Environmental Parameters Tab (OilFlow2D model).

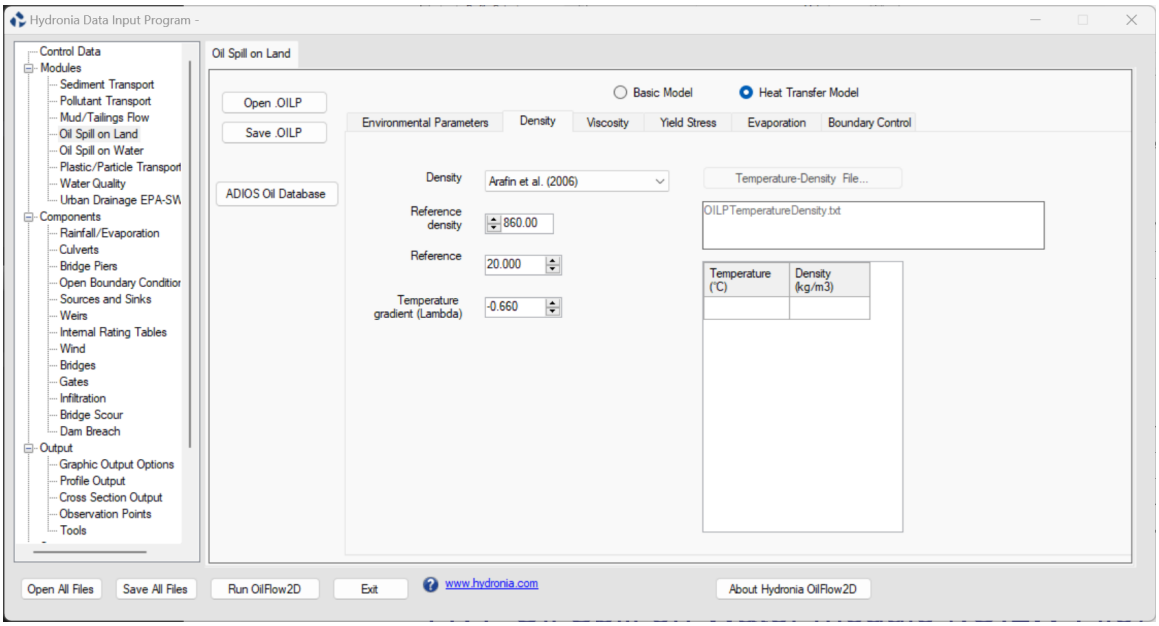


Figure 10.6 – Oil Spill on Land Heat Transfer Model Panel. Density Tab (OilFlow2D model).

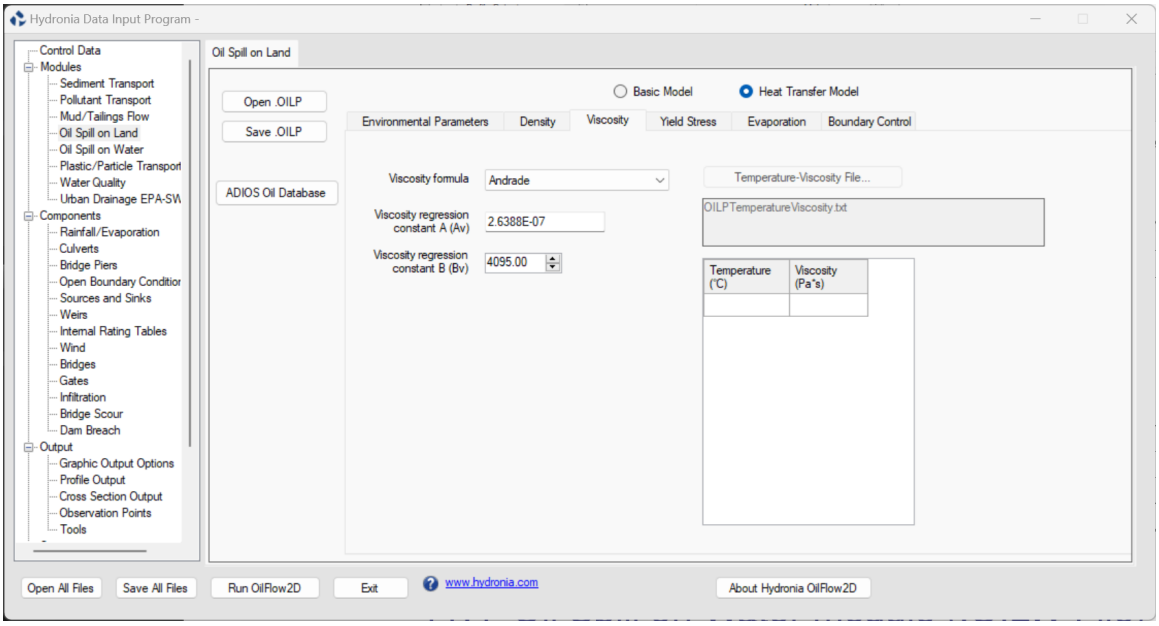


Figure 10.7 – Oil Spill on Land Heat Transfer Model Panel. Viscosity Tab (OilFlow2D model).

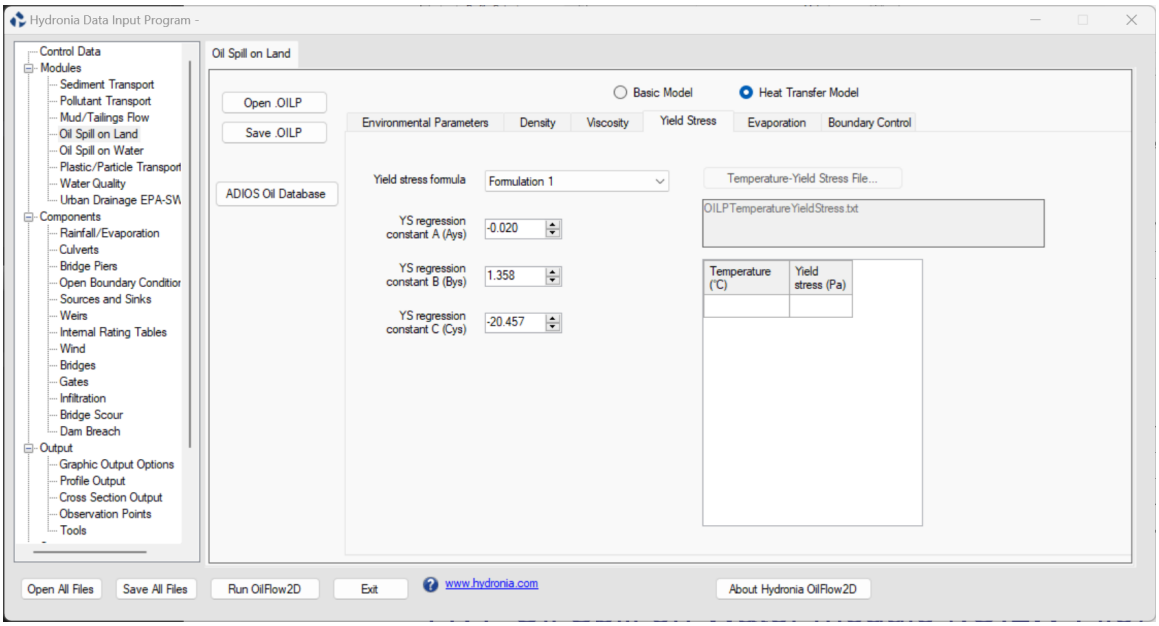


Figure 10.8 – Oil Spill on Land Heat Transfer Model Panel. Yield Stress Tab (OilFlow2D model).

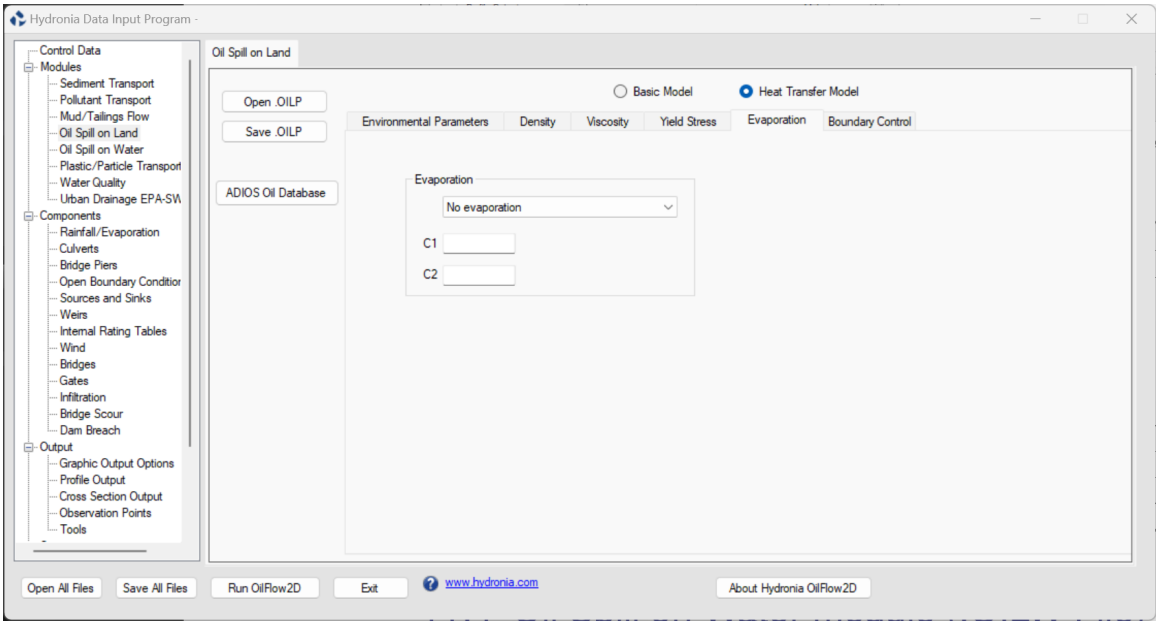


Figure 10.9 – Oil Spill on Land Heat Transfer Model Panel. Evaporation Tab (OilFlow2D model).

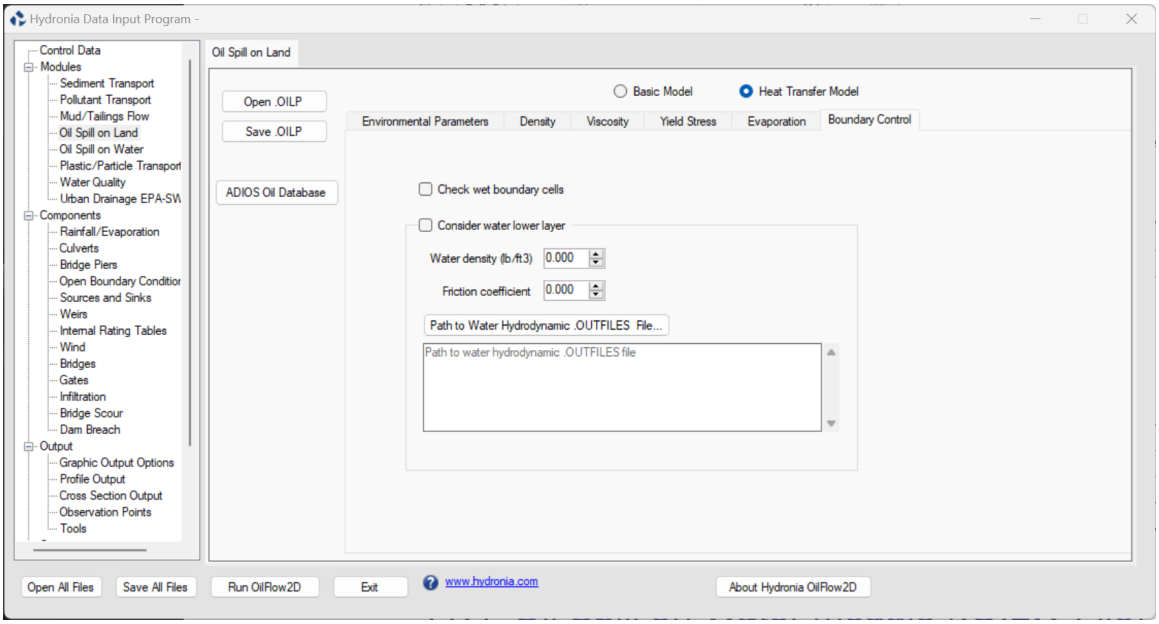


Figure 10.10 – Oil Spill on Land Heat Transfer Model Panel Boundary Control Tab (OilFlow2D model).

10.3 Oil Spill on Water module (.OILW File)

This panel allows entering data for OilFlow2D Oil Spill on Water module. To activate this panel, first select *Oil Spill on Water* on the *Components Frame* of the *Control Data* Panel.

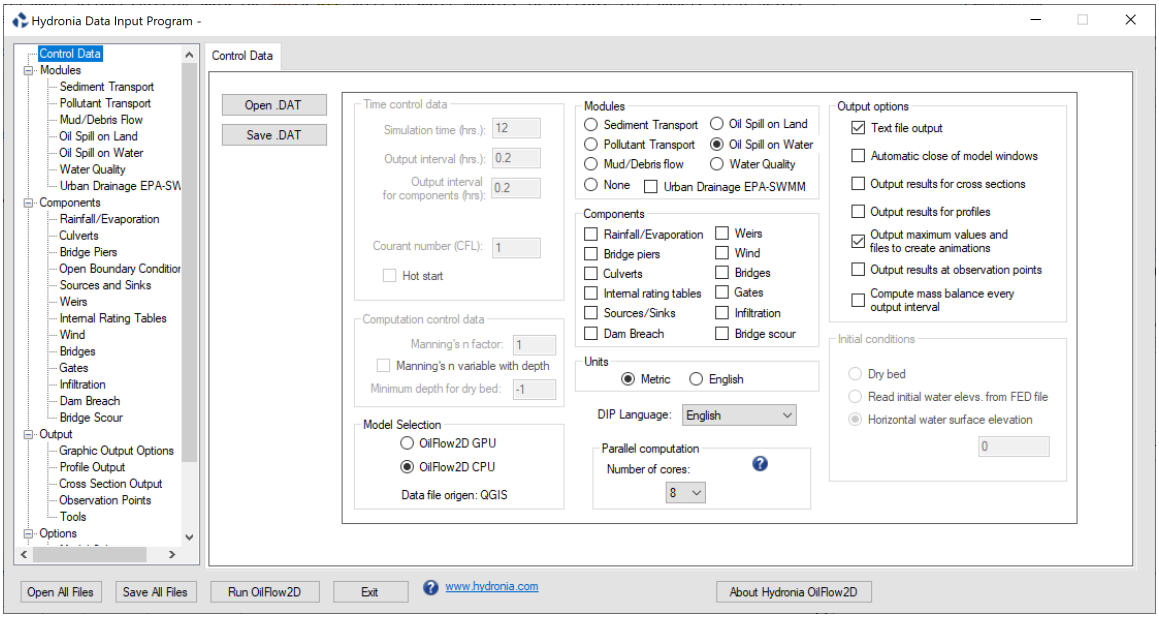


Figure 10.11 – Oil Spill on Water module selected in the Control Data Panel (OilFlow2D model).

The *Oil Spill on Water* panel includes tabs to enter data for the oil trajectory and oil weathering processes such as oil evaporation, emulsification, interaction with shorelines, dispersion, dissolution and sedimentation. Some of this processes are implemented in the present version and some will be added in forthcoming releases.

10.3.1 Oil Spill on Water Control Data Tab

Use the *Control Data* panel to enter general run control parameters as summarized in Table.

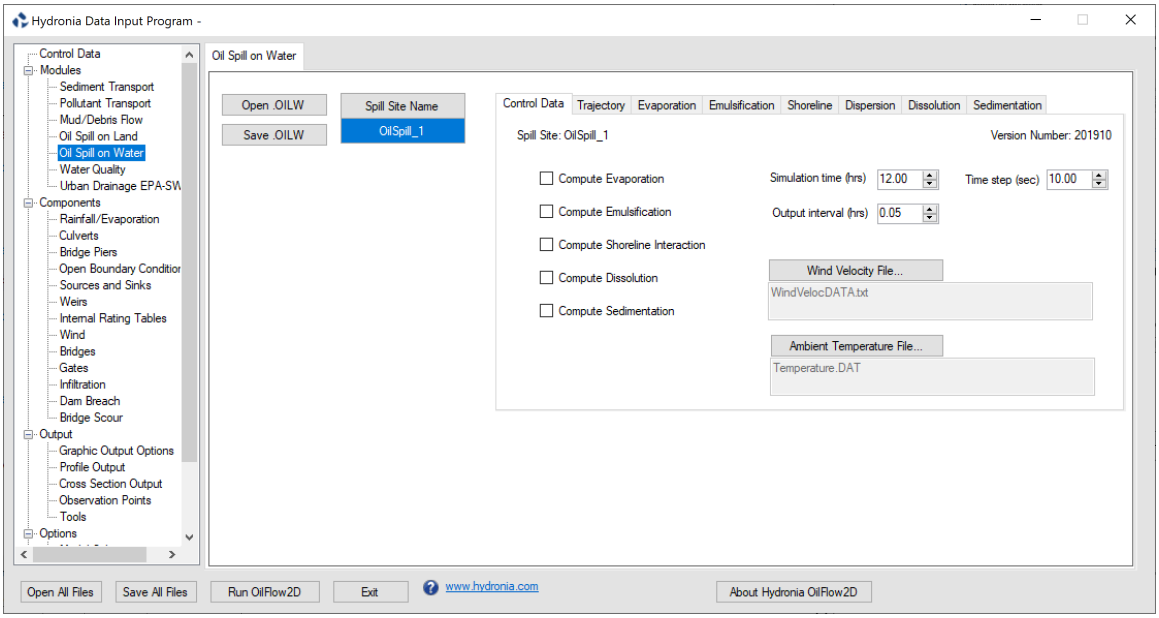


Figure 10.12 – Oil Spill on Water Panel. Control Data Tab (OilFlow2D model).

- **Process Switches:** Select whether to compute weathering processes.

1. Evaporation
2. Emulsification
3. Shoreline Interaction
4. Dissolution
5. Sedimentation

- **Simulation time (h.):** Total simulation time in hours.
- **Output Interval (h.):** Time interval for output reporting.
- **Time step (sec.):** Computational time step.
- **Wind Velocity File:** File containing wind velocity time series.
- **Ambient temperature file:** File containing temperature time series.

10.3.2 Oil Spill on Water Trajectory Tab

Use the *Trajectory* panel to enter parameters associated with the oil trajectory calculation as summarized in Table.

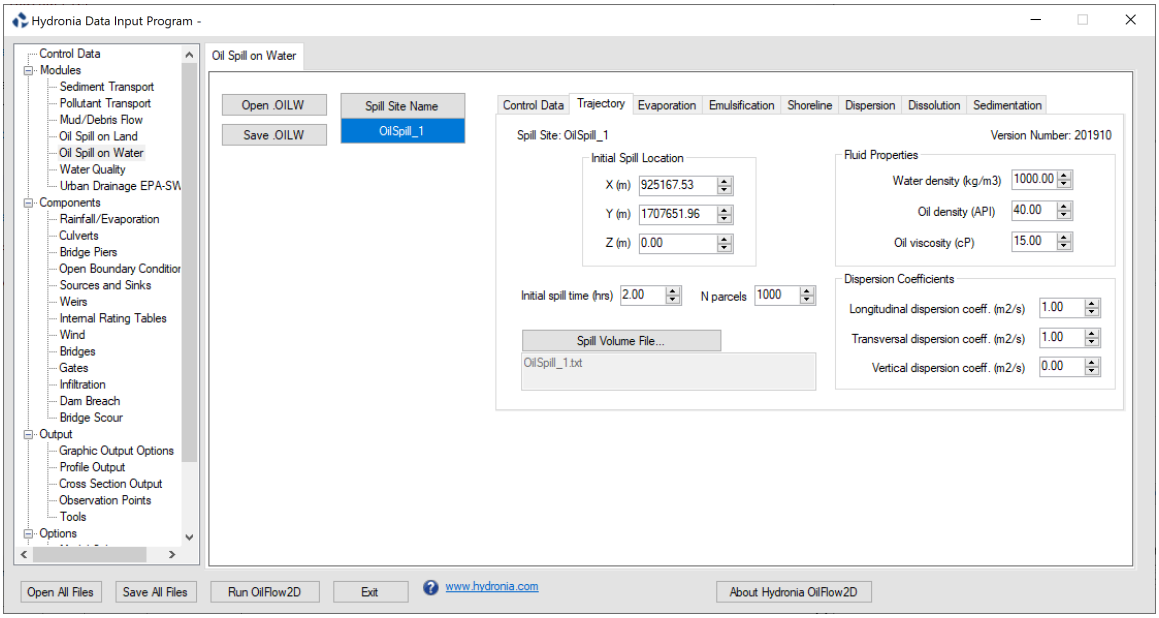


Figure 10.13 – Oil Spill on Water Panel. Trajectory Tab (OilFlow2D model).

- **X (m), Y (m), Z (m):** Coordinates of the location of the spill at time = 0.
- **Initial spill time (hrs.):** Time corresponding to the start of the spill with respect to the time in the velocity field calculations.
- **N parcels:** Number of parcels or particles used to represent the oil in the particle-tracking method.
- **Spill Volume File:** File containing the time series of oil volume entering the water. The volumes are given cumulatively for each time starting from 0 and ending with the total spill volume.
- **Water density:** Density of the receiving water body.
- **Oil density:** Oil density in API units.
- **Oil viscosity:** Dynamic viscosity of the spilled oil at time = 0.

- **Dispersion coefficients:** Longitudinal, transversal, and vertical dispersion coefficients (m^2/s or ft^2/s).

10.3.3 Oil Spill on Water Evaporation Tab

Use the *Evaporation* panel to enter the parameters required to compute oil evaporation as summarized in Table.

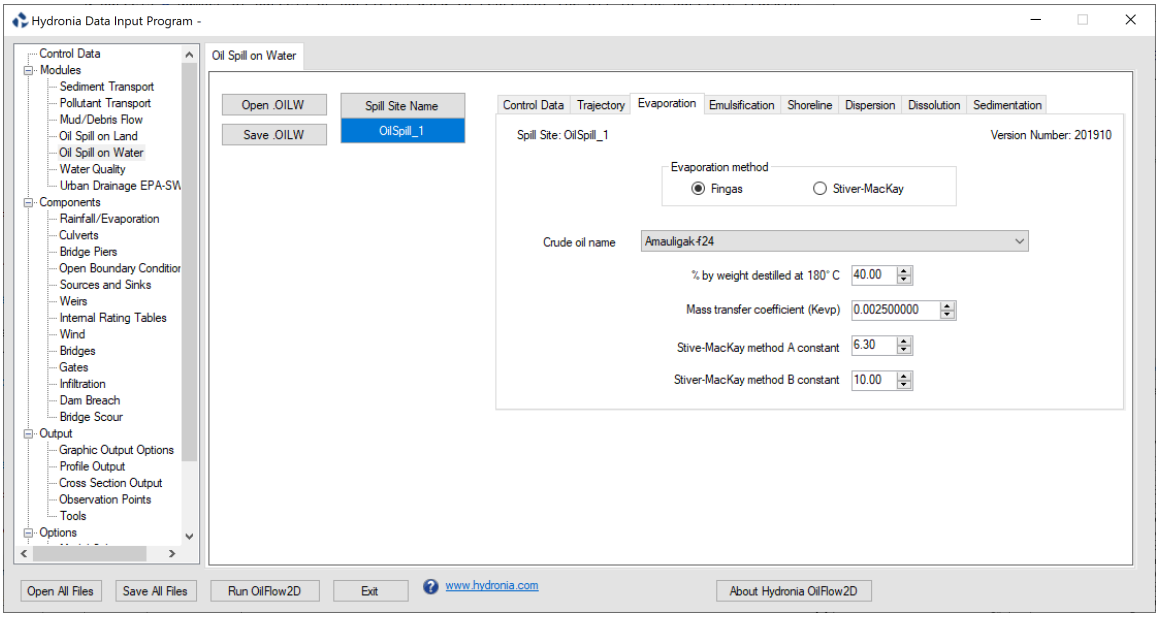


Figure 10.14 – Oil Spill on Water Panel. Evaporation Tab (OilFlow2D model).

- **Evaporation method:** To select either Fingas or Stiver-MacKay method.
- **Crude oil name:** Drop down list to access the Fingas method oil data base.
- **% by weight distilled at 180 °C:** Used when selecting generic oils that follow logarithmic square root evaporation law.
- **Mass transfer coefficient:** Required in the Stiver-MackKay method.
- **Stiver-MackKay method A constant:** Constant A used in Stiver-Mackay method.
- **Stiver-MackKay method B constant:** Constant B used in Stiver-Mackay method.

10.4 Pollutant Transport panel (.SOLUTES)

Use the *Pollutant Transport* panel to enter the parameters required to characterize the reaction rates between multiple solutes as summarized in Table.

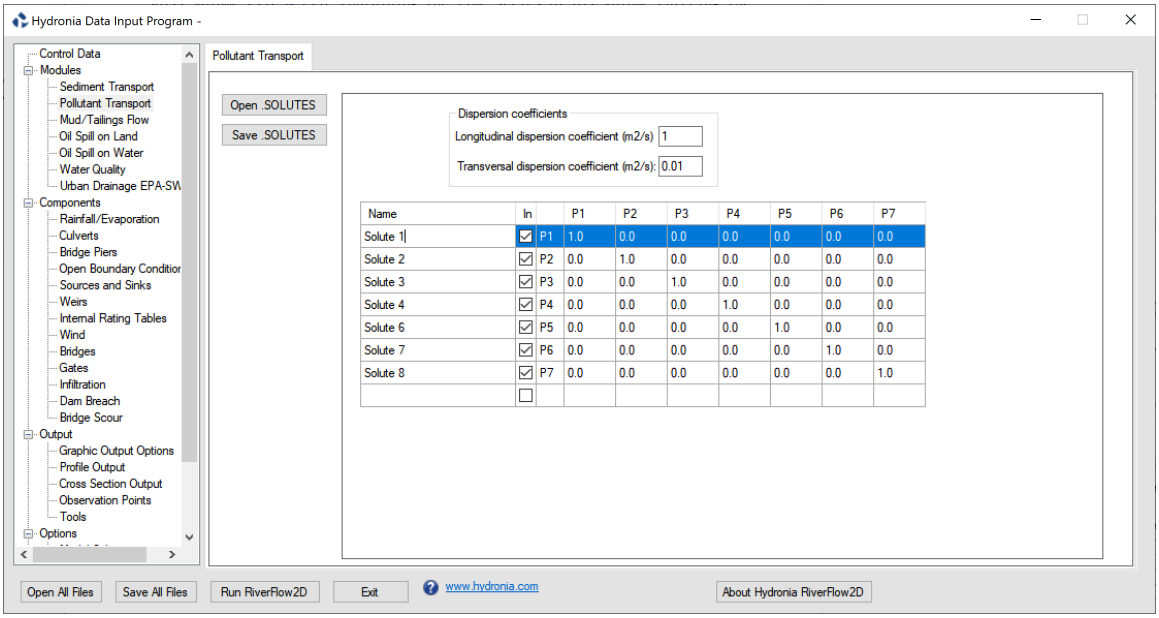


Figure 10.15 – Pollutant Transport panel.

- **Dispersion coefficients:** Longitudinal and transversal dispersion coefficients for the PL module (m^2/s or ft^2/s).
- **Name:** Solute name.
- **In:** Check box to include pollutant in the simulation.
- **Reaction coefficients:** Matrix to enter the linear reaction coefficients between the solutes. The matrix is symmetrical. The diagonal may be used to enter the linear decay coefficient for a given pollutant. You can add more solutes by clicking on the last row name. You can also eliminate a solute by selecting the name and using the Delete keyboard key. The columns tittles (P1, P2,…) correspond to each solute.

10.5 Graphic Output Options Tab (.PLT File)

This panel allows entering options to control OilFlow2D output. To activate this panel, first select *Graphic Output Options* from the *Output* group on the left panel DIP.

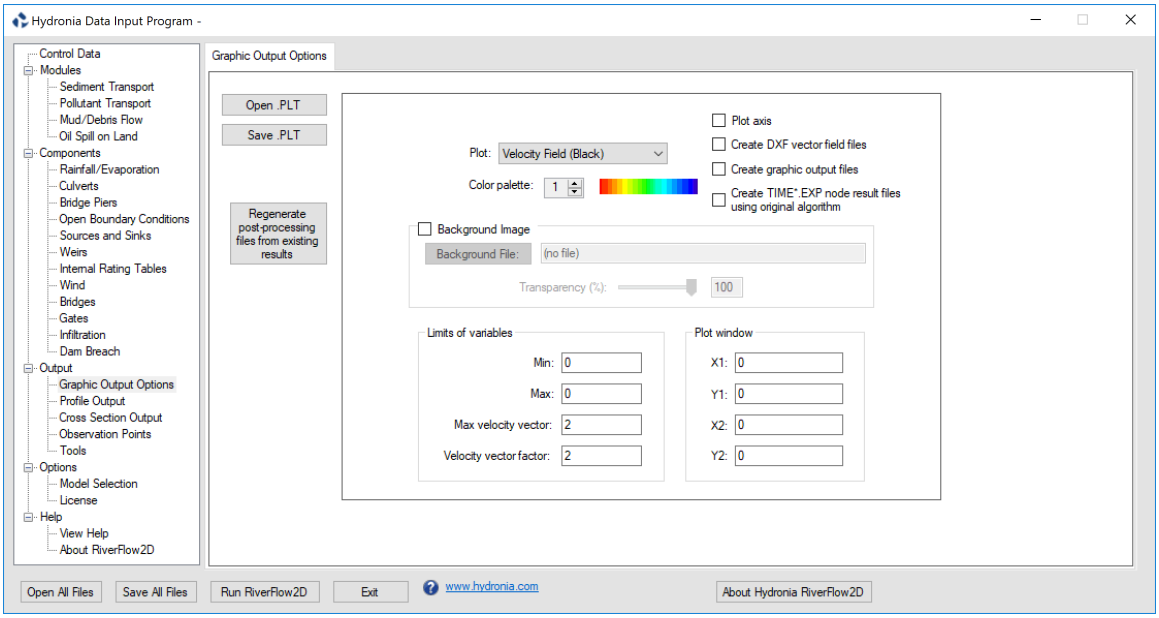


Figure 10.16 – Graphic Output Panel.

- **Open:** Opens an existing output data file.
- **Save:** Saves only the graphic output data to a file.
- **Regenerate post-processing files from existing results:** Use this button to create output files from existing simulations. The output files written during a previous simulation must be available.
- **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:** For future use.
- **Plot axis:** For future use.

- **Create DXF vector field files:** Generate velocity vector DXF (CAD) files. This option will also export the mesh in DXF format to the file: .
- **Create graphic output files:** For future use.
- **Create node result files using original algorithm.:** Using this option OilFlow2D when running will generate files using an algorithm to compute nodal values from cell values that was available in versions older than 2018.
- **Background image:** For future use.
- **Background file:** For future use.
- **Transparency:** For future use.
- **Min:** For future use.
- **Max:** For future use.
- **Max velocity vector:** For future use.
- **Velocity vector factor:** For future use.
- ****Plot Window Frame**:** For future use.
- **X1:** For future use.
- **Y1:** For future use.
- **X2:** For future use.
- **Y2:** For future use.

10.6 Profile Output Panel (.PROFILES File)

Use this panel to enter polyline coordinates where the model results are to be generated. The model will generate output and files. To activate this panel, first select the *Profile Output* in *Output* from the *Output* group on the left panel of DIP.
See output file section for output file content description.

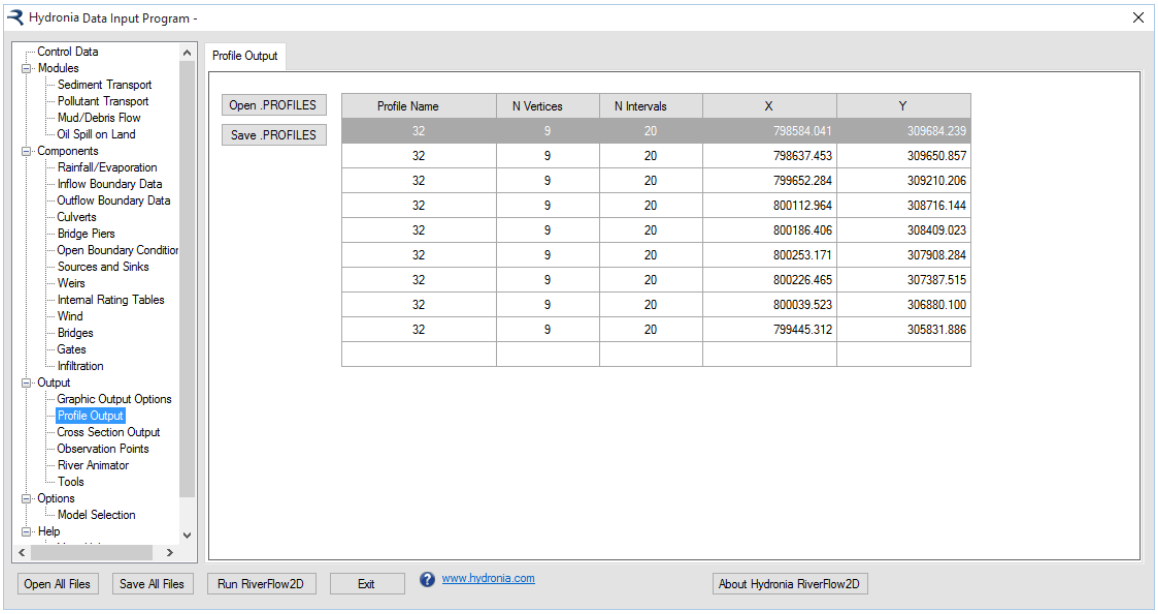


Figure 10.17 – Profile Output File.

- **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.
- **Open:** Opens an existing file.
- **Save:** Saves only the profile data to a file.

10.7 Cross Section Output Panel (.XSECS File)

Use this panel to enter coordinates for cross sections that intersect the triangular-cell mesh where you want to output model results. OilFlow2D will generate output and files. To activate this panel, first select the *Cross Section Output* from the *Output* group on the left panel of p p.

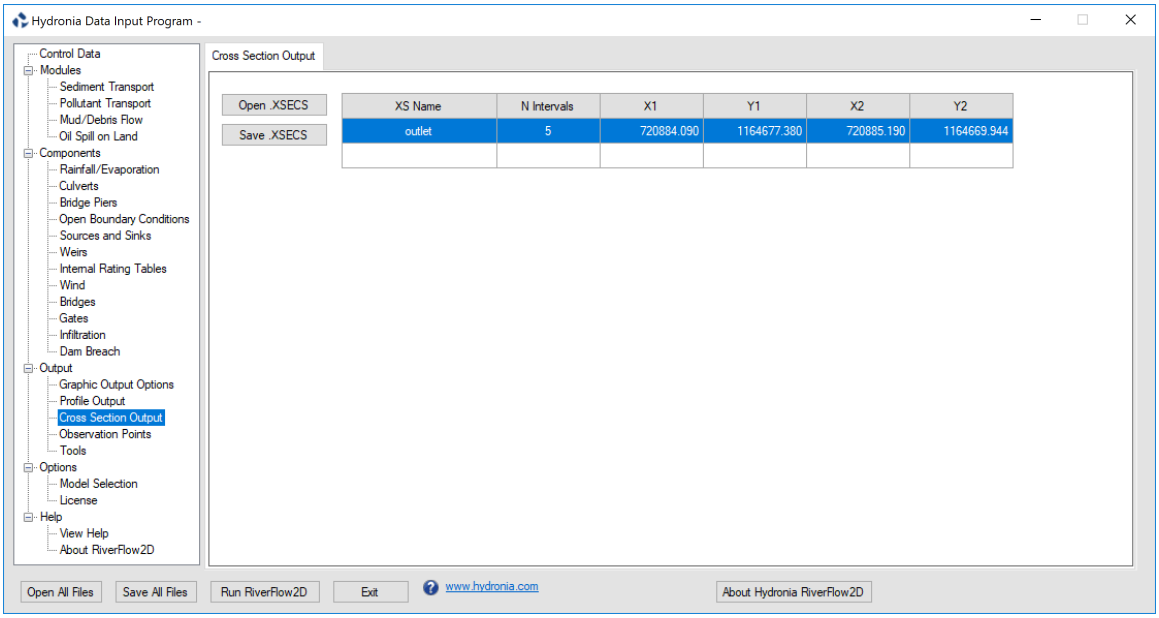


Figure 10.18 – Cross Section Output Panel.

- **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.
- **Open:** Opens an existing file.
- **Save:** Saves only the cross section data to a file.

10.8 Culverts Panel (.CULVERTS File)

This panel is used to display the content of the file and enter data for culverts. Figure 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 DIP.

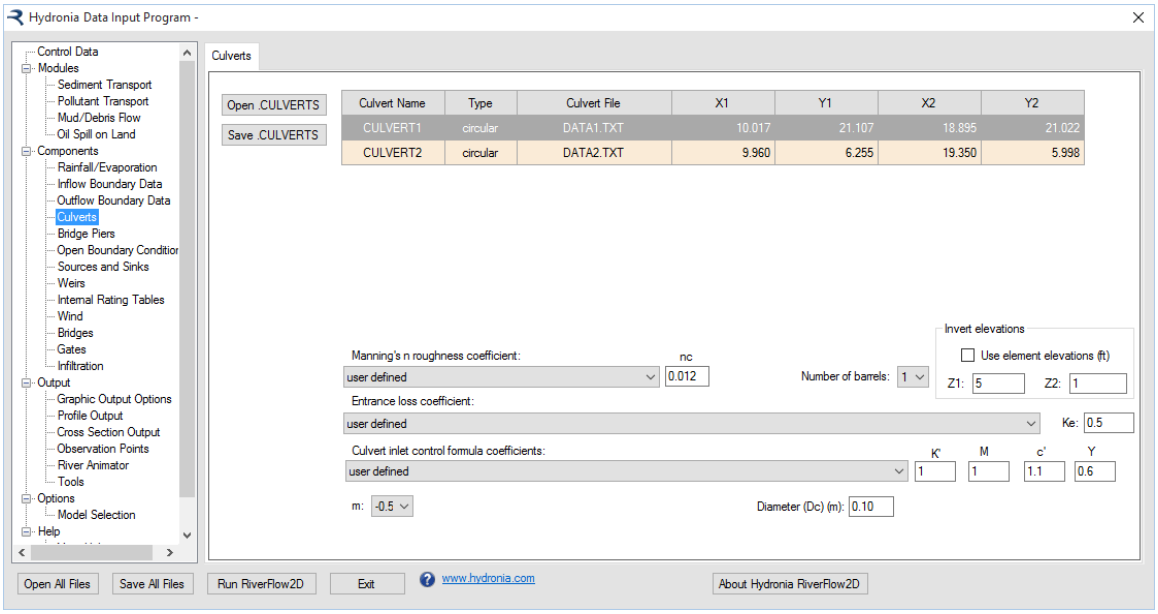


Figure 10.19 – Culverts Panel showing data in rating curve.

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

- **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 .
- **Entrance loss coefficient:** Culvert entrance loss coefficient given by Table .
- **Culvert inlet control formula coefficients:** Culvert inlet control formula coefficients given by Table .
- **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.
- **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 cell elevations:** When this check box is selected the model will extract the inlet and outlet invert elevations from the 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 cell elevations.
- **Open:** Opens an existing file.

- **Save:** Saves only the culvert data to a file.

10.9 Internal Rating Tables Panel (.IRT File)

This panel is used to display the content of the file and enter data for Internal rating Tables. 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 DIP.

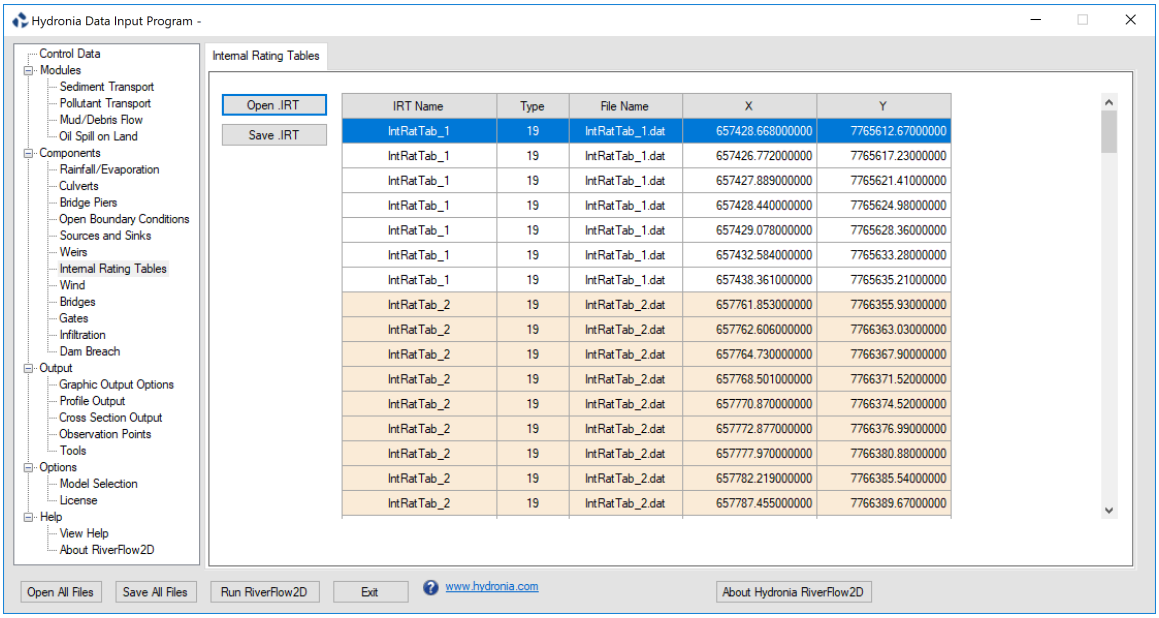


Figure 10.20 – Internal Rating Tables Panel.

- **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.
- **Open:** Opens an existing file.
- **Save:** Saves only the internal rating table data to a file.

10.10 Weirs Panel (.WEIRS File)

This panel is used to display the content of the 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 DIP.

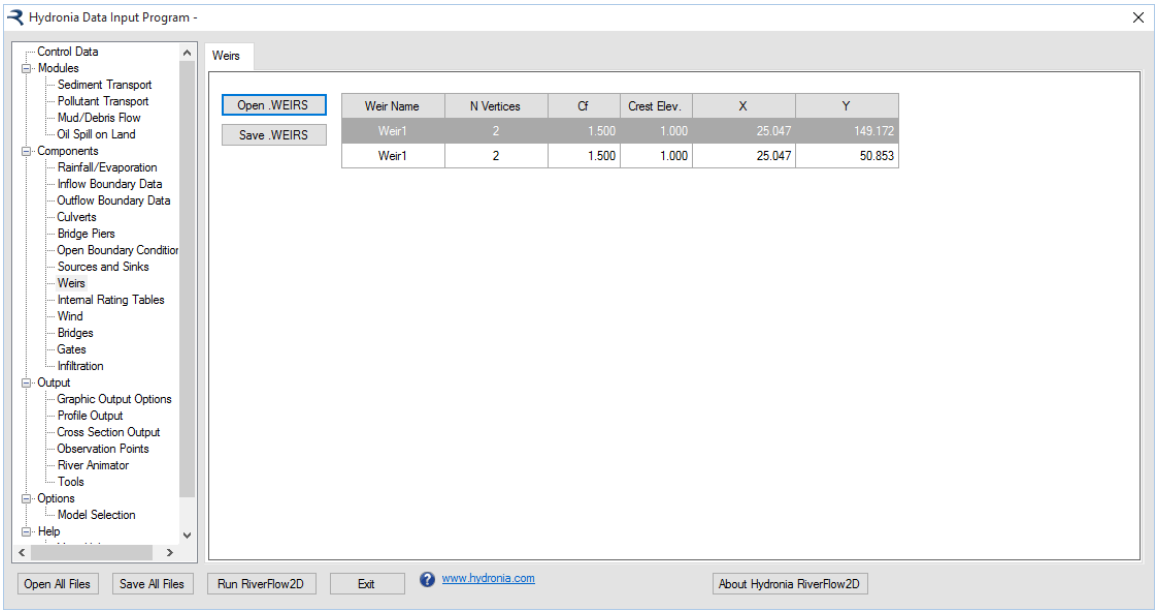


Figure 10.21 – Weirs Panel.

- **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).
- **Open:** Opens an existing file.
- **Save:** Saves only the weir data to a file.

10.11 Sources/Sinks Panel (.SOURCES File)

This panel is used to display the content of the 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 DIP.

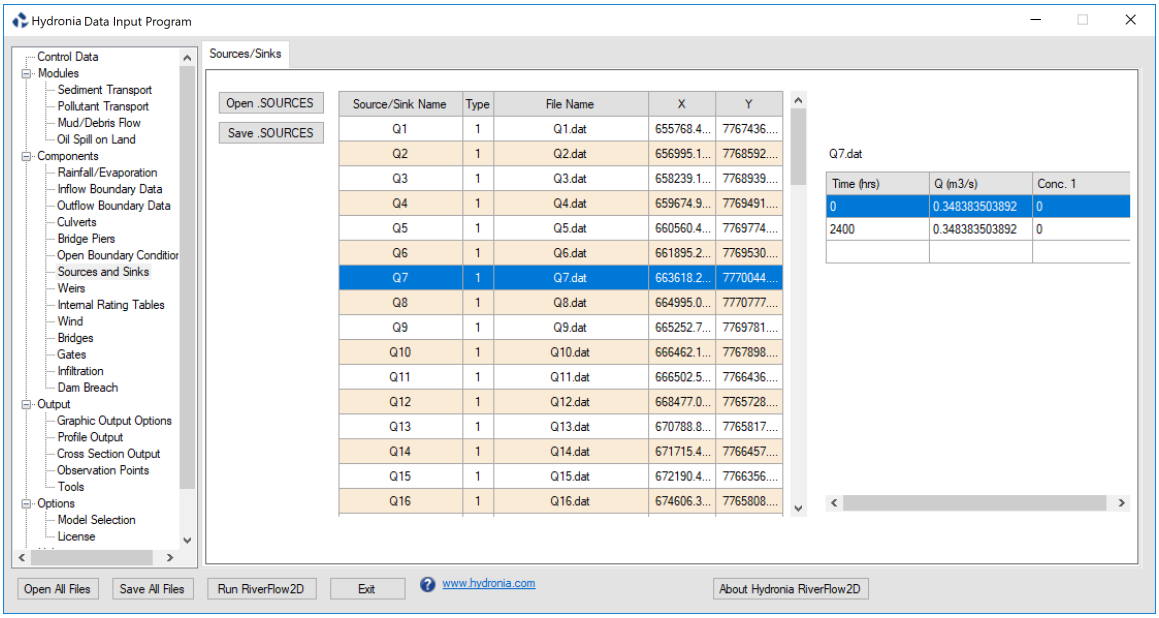


Figure 10.22 – Sources/Sinks Panel.

- **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 or rating table of each point source or sink.
- **Type:** Source/sink type. If equal to 1, the file should contain a hydrograph. If equal to 2, it contains a rating table with depths vs discharge values.
- **X, Y:** Coordinates of point.
- **Open:** Opens an existing file.
- **Save:** Saves only the sources and sinks data to a file.

10.12 Bridge Scour Panel (.SCOUR File)

This panel is used to display the content of the file. Use this Panel to edit bridge pier or abutment data and calculate scour on those structures. To activate this panel, first select *Bridge Scour* from the *Components* group on the left panel of DIP.

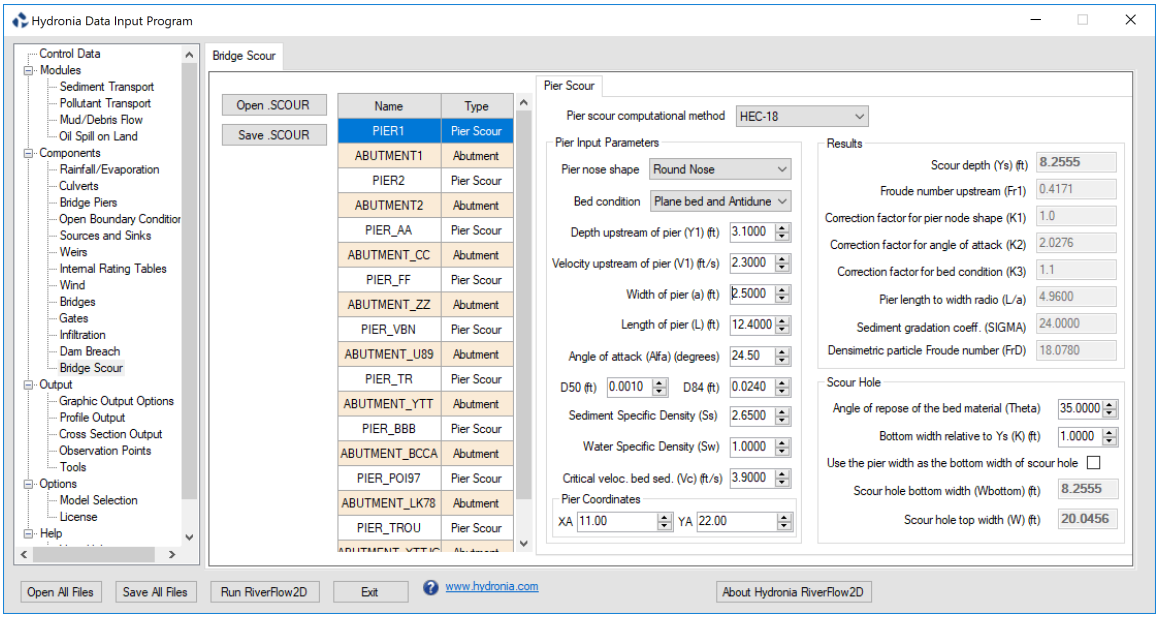


Figure 10.23 – Bridge Scour Panel.

- **Pier ID:** Pier name
- **Icomp::** Computational method
- **XA, YA:** Pier coordinates
- **Y1:** Flow depth directly upstream of the pier
- **V1:** Velocity upstream of the pier
- **Alfa:** Angle of attack
- **alfaRAD:** Angle of attack
- **ishape:** Pier shape
- **L:** Pier length
- **a:** Pier width
- **iBedCondition:** Bed condition
- **D50:** D50
- **D84:** D84
- **Sediment Specific Density:** Ss
- **Water Specific Density:** Sw
- **K1:** Correction factor for pier nose shape.
- **K2:** Correction factor for angle of attack of flow
- **K3:** Correction factor for bed condition
- **K:** bottom width relative to Ys.
- **theta:** angle of repose of the bed material
- **ys:** Scour depth
- **W:** scour hole top width
- **Wbottom:** scour hole bottom width
- **Fr1:** Froude Number upstream of pier
- **FrD:** Densimetric particle Froude Number
- **SIGMA:** Sediment gradation coefficient
- **Vc:** Critical velocity for initiation of erosion of the material

- **iAbutmentType:** Abutment Type
- **AlfaA:** Amplification factor for live-bed conditions
- **AlfaB:** Amplification factor for clear-water conditions
- **YmaxLB:** Maximum flow depth after scour for live-bed conditions
- **YmaxCW:** Maximum flow depth after scour for clear-water conditions
- **YcLB:** Depth including live-bed contraction scour
- **YsA:** Abutment scour depth
- **YcCW1:** Depth including clear-water contraction scour. Method 1
- **YcCW2:** Depth including clear-water contraction scour. Method 2
- **q1:** Upstream unit discharge
- **q2c:** Upstream unit discharge of the constricted opening
- **n Manning:** Mannings n
- **TauC:** Critical shear stress
- **GammaW:** Unit weight of water
- **BridgeXSEC_X1, BridgeXSEC_Y1, BridgeXSEC_X2, BridgeXSEC_Y2:** Coordinates of extreme points of Bridge Cross Section
- **UpstreamXSEC_X1, UpstreamXSEC_Y1, UpstreamXSEC_X2, UpstreamXSEC_Y2:** Coordinates of extreme points of Upstream Cross Section
- **Open:** Opens an existing file.
- **Save:** Saves only the bridge pier and abutment data to a file.

10.13 Bridge Piers Panel (.PIERS File)

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

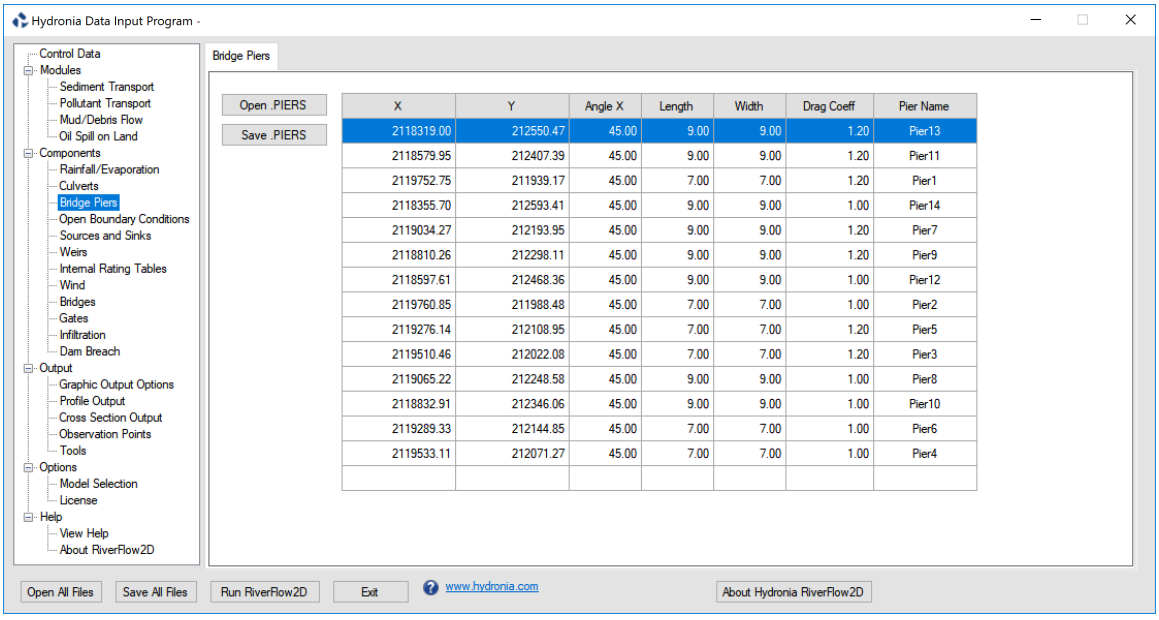


Figure 10.24 – Bridge Piers Panel.

- **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.
- **Open:** Opens an existing file.
- **Save:** Saves only the bridge piers data to a file.

> **Note**
>
> To simulate circular piers use the same width and length and set an adequate Drag Coefficient for round piers.

10.14 Observation Points Panel (.OBS File)

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

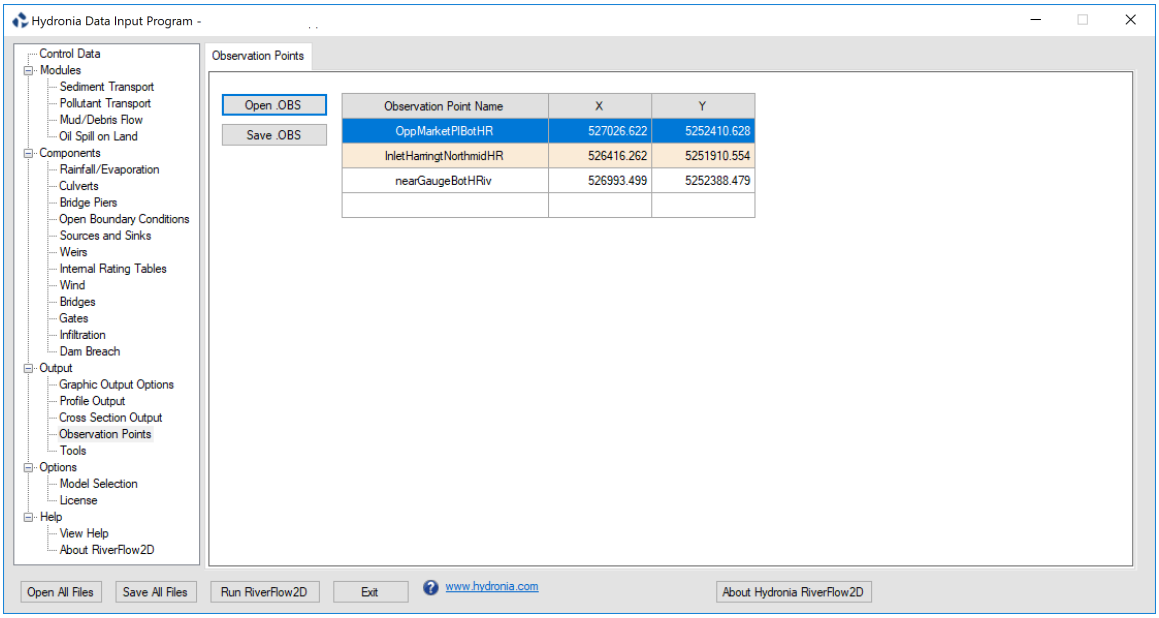


Figure 10.25 – Observation Points Panel.

- **Observation Point Name:** Name of observation point. Should not contain spaces and must have less than 26 characters.
- **X,Y:** Coordinates of point.
- **Open:** Opens an existing file.
- **Save:** Saves only the observation point data to a file.

10.15 Tools Panel

This section describes various utilities that are available through DIP. To activate this panel, first select the *Tools* from the *Output* group on the left panel.

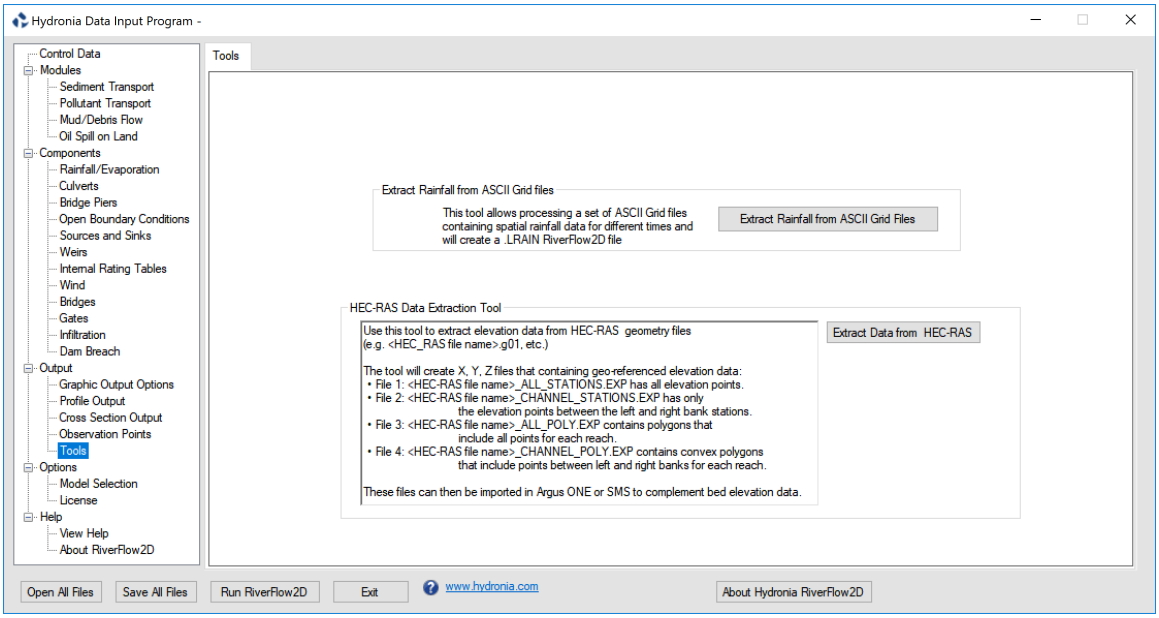


Figure 10.26 – Tools Panel.

10.15.1 Process Rainfall and Evaporation Data from ASCII Grid Files Tool

You can use this tool to process rainfall (e.g. NEXRAD) and evaporation ASCII Grid Files to prepare OilFlow2D data files that can be used for hydrologic simulations. The program will let you extract data from a set of point rainfall and evaporation ASCII grid files and to create a file in the format readable by OilFlow2D.

To use the tool, you need first to create file with a text editor (e.g Notepad) using the following format:

Line 1: Number of ASCII Rain Files

NRF

NRF lines containing:

Ti RAINFILEi.ASC

Where Ti is the time in hours and is the ASCII Grid file for the rainfall corresponding to time Ti.

Optionally if you have a set of evaporation files you add the following lines:

Number of ASCII Evaporation Files (must be equal to NRF)

NEV

NEV lines containing:

Ti EVAPFILEi.ASC

Where Ti is the time in hours and is the ASCII Grid file for the evaporation corresponding to time Ti.

It is assumed that in the rain and evaporation ASCII files values are given in mm or in. Since OilFlow2D uses intensities instead of mm or in, the values provided will be converted internally to

mm/hr or in/hr using the time interval determined from the times provided in the file described above. If the number of files (NRF) in the first line is positive, the rainfall/evaporation will be assumed to be given in points, and will be interpolated to each cell. If the number is negative, the program will consider the rain/evaporation given in squares centered at each grid point, and then the cell precipitation will be that of the grid where the cell centroid is located. This last method does not involve interpolation and is faster than the first method.
Once you have the file created and the files are located in the same folder, use the *Extract Rainfall from ASCII Grid Files Tool* button, select the file and click Open.
Wait for a few moments and enter the name of the file. The conversion process will take a few seconds or minutes depending on the number of files and their size.
To use the resulting file, you should copy it to the project folder, making sure to set the same file name as that of your project. For instance, if your project files are , , then name it as.

10.15.1.1 Example of a .RFC File with rainfall only

```
-4
0 rain_spas1275_001_20040917_0100_utc.asc
1 rain_spas1275_002_20040917_0200_utc.asc
2 rain_spas1275_003_20040917_0300_utc.asc
3 rain_spas1275_004_20040917_0400_utc.asc
```

10.15.1.2 Example of a .RFC File with rainfall and evaporation

```
-4
0 rain_spas1275_001_20040917_0100_utc.asc
1 rain_spas1275_002_20040917_0200_utc.asc
2 rain_spas1275_003_20040917_0300_utc.asc
3 rain_spas1275_004_20040917_0400_utc.asc
-4
0 evap_spas1275_001_20040917_0100_utc.asc
1 evap_spas1275_002_20040917_0200_utc.asc
2 evap_spas1275_003_20040917_0300_utc.asc
3 evap_spas1275_004_20040917_0400_utc.asc
```

10.15.2 HEC-RAS Data Extraction Tool

The purpose of this tool is to facilitate migrating existing HEC-RAS projects to OilFlow2D. 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 , , etc., and creates X Y Z files that can be readily imported in QGIS. 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.
& Contains all elevation points in all cross sections in the for all reaches and cross sections in the file.

& Contains polygons that include all elevation points in each reach.
& Contains only the elevation points between the left and right banks in all cross sections in the for all reaches in the file.
& Contains polygons that include only the elevation in the main channel for each reach.

11

Input Data File Reference

Data files for non-spatial information required to run the OilFlow2D. All OilFlow2D 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 OilFlow2D file formats. This section explains the input data files, and the parameters included in each file.
The OilFlow2D 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\OilFlow2D_QGIS\ExampleProjects.

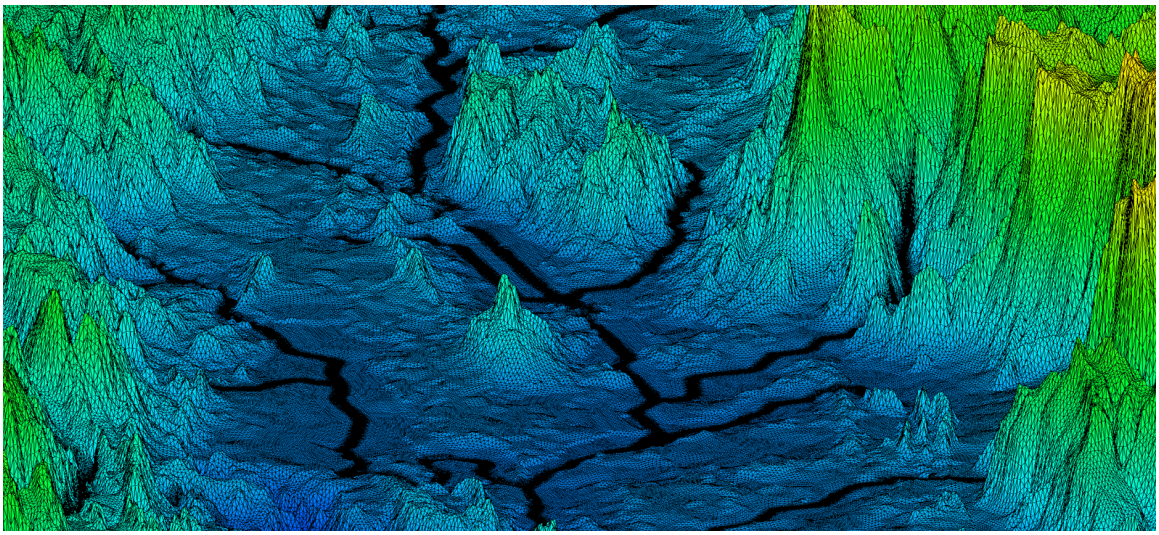


Figure 11.1 – Example of a OilFlow2D Mesh.

OilFlow2D 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: , , etc. The following table sum-

marizes the data files used by OilFlow2D model.

> **Note**
> Table DEPENDENCIES column indicates all required and optional files depending on the options selected. You may use this information to select the files that should be transferred to another computer that will perform the simulations, or to a Virtual Machine on a Cloud Service.

- **QGIS project file:** Required when using the OilFlow2D; This is the project file where QGIS stores all the spatial data used in the project, including the triangular cell mesh.
- **Elevation data:** any; Required; Scattered elevation data points.
- **Triangle-cell mesh data:** 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:** Internal file; List of external and island boundary nodes. This file is internally generated by OilFlow2D.
- **I/O boundary conditions:** Internal file; List of external boundary nodes, inflow and outflow conditions. This file is internally generated by OilFlow2D.
- **Boundary condition nodal file:** Required; List of external boundary conditions. For each boundary, it contains the list of nodes and the associated data file. Note that all files listed within are required to run the model, and should reside in the same folder. This file is now internally generated by OilFlow2D, based on the information in the file.
- **Run control data:** Required; General run control options, including time step, simulation time, metric or English units, graphical output options, initial conditions, components, etc.
- **Plot results options:** Optional; Graphical output options.
- **Observation points data:** Optional; Location of observation points where the model will report time series of results.
- **Cross section output:** 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:** Optional; Mesh profile cut where results are desired.
- **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:** Required when using the Pollutant Transport module.; Defines the initial concentrations over the mesh.
- **Bridges:** Required when using the Bridges component.; Bridge cross section geometry file is used to compute energy losses.

- **Culverts:** Required when using the Culvert component.; Culvert location and associated culvert data files.
- **Dam Breach:** Required when using the Dam Breach component.; Location and data for the dam breach.
- **Gates:** Required when using the Gates component.; Gate location and associated gate aperture data files.
- **Infiltration:** Required when using the Infiltration component.; Infiltration parameters data file.
- **Internal rating tables:** Required when using the Internal Rating Table component.; Data to impose discharge rating tables along internal boundaries.
- **Manning's n variable with depth:** Required when using variable Manning's n with depth.; Provides the parameters necessary to account for Manning's n roughness coefficient that vary with depth according to a user provided table. Created from polygons on the ManningsNz layer.
- **Bridge piers:** The Piers component is selected; Bridge pier data used to calculate pier drag forces.
- **Rainfall/Evaporation:** Required when using the Rainfall/Evaporation component.; Time series for rainfall and evaporation.
- **Sources and sinks:** Required when using the Sources component.; This file contains location of input discharge sources or output discharge sinks and associated time series of discharge data files.
- **Weirs:** Required when using the Weir component.; This file contains weirs polylines and associated weir data.
- **Wind:** Required when using the Wind component.; This file contains wind specific density and velocity data.
- **Bridge Scour:** Required to calculate bridge pier or abutment scour.; This file contains pier and abutment parameters needed to compute scour.
- **Oil Spills on Land:** Required when using the OilFlow2D model to simulate overland spills.; Provides the parameters necessary to model overland oil spills.
- **Oil Spills on water:** Required when using the OilFlow2D model to simulate oil spills over water.; Provides the parameters necessary to model oil spills on water.
- **Pollutant transport:** Required when using the Pollutant Transport module.; Data for passive or reactive pollutants. ## Run Control Data

11.0.1 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 ISWMM

Line 4: Wet-dry bed method switch.

IWETDRY

Line 5: Output control switches.

IEXTREMES IXSEC IPROFILE NOGRAPH 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: Mass Balance Reporting Switch.

IMASSBAL

Line 11: Unit system definition switch.

NUNITS

Line 12: Minimum flow depth for dry areas.

HMIN

Line 13: Initial water surface elevation.

INITIAL_WSE

Line 14: Pollutant transport model switch.

IPOLLUTANT

Line 15: Wind stress switch.

IWIND

Line 16: Oil Spill model switch.

IMDOIL

Line 17: Number of cores or GPU ID.

IDGPU

Line 18: Graphical User Interface that created the files.

IGUI

Line 19: Additional components.

ISCOUR IMULTSOURCES IHAZARD HARRIVAL FUTURE5 FUTURE6 FUTURE7 FUTURE8 FUTURE9 FUTURE10

11.0.1.1 Example of .DAT file

```
201905
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
2
1  0  0  0.05  0  0  0  0  0  0  0
```

- **CFL:** R; $(0, 1]$; -; Applies to OilFlow2D and OilFlow2D GPU models. Courant number. Default value is set to 1.0. CFL may need to be set to lower values if results show signs of unexpected oscillations.
- **DUMMY:** R; -; -; Dummy parameter for future use. Ignored in OilFlow2D.
- **HMIN:** R; -1 or > 0; m/ft; In OilFlow2D 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.
- **HARRIVAL:** R; ≥ 0; m/ft; The model will report the inundation or frontal wave arrival time to each cell when the depth at the cell reaches HARRIVAL for the first time during the simulation.
- **IADDISP:** I; 0,1; -; Switch to activate the pollutant transport model.

1. Turn off pollutant transport computations.
2. Apply pollutant transport.

- **IBRIDGES:** I; 0,1; -; Switch to activate the Bridges component.

1. Turn off Bridges component.
2. Apply Bridges component.

Requires file. See details on the *Bridges* Section of this manual.
- **ICULVERTS:** I; 0,1; -; Switch indicating if one-dimensional culverts will be used.

1. No culverts will be used.
2. Use culverts.

Requires file. See details on the *Culverts* Section of this manual.
- **IDAMS:** I; 0,1; -; Switch to activate the Dam Breach component.

1. Turn off Dam Breach component.
2. Apply Dam Breach component.

Requires file. See details on the *Dam Breach* Section of this manual.
- **IDGPU:** I; ≥ 0; -; OilFlow2D: This parameter indicates how many processors or cores will be used in the parallel computation. The maximum number will depend on the processor capabilities. OilFlow2D 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.

1. Do not report maximum values.
2. Report maximum values.

- **IGATES:** I; 0,1; -; Switch to activate the Gates component.

1. Turn off Gates component.
2. Apply Gates component.

Requires 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 OilFlow2D files.

1. Aquaveo SMS
2. QGIS

- **IHAZARD:** I; 0,1; -; Switch to create flood hazard files.

1. Does not create hazard files.
2. The model will create the hazard files.

- **IHOTSTART:** I; 0,1; -; Switch to start run from scratch or continue a previous simulation.

1. Start simulation from initial time.
2. Start simulation from previous run.

- **IINTRC:** I; 0,1; -; Switch for internal rating tables.

1. Do not use internal rating table component.
2. 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.

1. Prescribed horizontal water surface elevation
2. Initial dry bed on whole mesh.
3. Initial water surface elevations read from 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.

1. Manning's n is constant for all depths.

2. Manning's n may vary with depths as defined in the file.

- **IMASSBAL:** I; 0,1; -; Mass balance report switch. Used to define when to calculate mass balance and create the file.

1. Mass balance is not calculated every time step, and is not created.
2. Mass balance is calculated every time step, and is created

. See comment 9.
- **IMDOIL:** I; 0,3; -; Switch to select the oil model.

1. Do not run the oil model.
2. Run the oil spill on land flow model. Requires file. See details on the Oil Spills on Land section of this manual.
3. Run the oil spill on water model. Requires file. See details on the Oil Spills on Water section of this manual.

- **IMS:** I; 1,2; -; Model switch used to select the hydrodynamic model engine.

1. OilFlow2D.
2. OilFlow2D GPU.

- **IMULTSOURCES:** I; 0,1; -; Switch used to select multiple-sources batch processing. When set to 1, the model will create a sub-directory (named as the source ID) for each source, and perform independent runs in each sub-directory.

1. All sources will be considered acting simultaneously.
2. The model will perform as many runs as sources are defined.

- **IOBS:** I; 0,1; -; Switch to report time series of results at specified locations defined by coordinates.

1. Do not report on observation points.
2. Report on observation points.

Requires file. See details on the *Observation Points* section of this manual.
- **IPIERS:** I; 0,1; -; Switch to allow accounting for pier drag force.

1. Do not use pier drag force option.
2. Use pier drag force option.

Requires 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, 2; -; Switch to select pollutant model.

1. Do not run pollutant transport models.
2. Run pollutant transport advection-dispersion-reaction model. Requires file. See details on the *Pollutant Transport* section of this manual.

- **IPROFILE:** I; 0,1; -; Switch to control profile output.

1. No profile results output.
2. Results will be output along a prescribed profile.

Requires file. See comment 4.
- **IRAIN:** I; 0-4; -; Switch for rainfall and evaporation input.

1. No rainfall modeling.
2. Not used.
3. Rainfall/evaporation.
4. Infiltration.
5. Rainfall/evaporation and Infiltration.

- **ISCOUR:** I; 0,1; -; Switch for scour computations.

1. Deactivate scour computation around piers and abutments.
2. Compute scour around bridge piers or abutments. Requires file.

- **ISOURCES:** I; 0,1; -; Switch for sources and sinks.

1. No sources or sinks are present.
2. Sources or sinks are present. Requires file.
3. Sources or sinks are present, but each source will be solved as a separate scenarios in different sub-directories named according to each source ID. Requires file.

See details on the *Sources* section of this manual.
- **ISWMM:** I; 0,1; -; Switch for linking with EPA-SWMM model.

1. Deactivate link with EPA-SWMM model.
2. Compute surface water flow and interaction with storm drains with EPA-SWMM model. Requires file and a compatible SWMM model file.

- **IWEIRS:** I; 0,1; -; Weir computation on internal boundary switch.

1. Do not use weir computation on internal boundaries.
2. 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.

1. Do not consider wind stress.
2. Consider wind stress. Requires file.

See details on the *Wind Stress* section of this manual.
- **IXSEC:** I; 0,1; -; Cross section output switch.

1. No cross section result output.
2. Cross section results will be output to file. Requires file. See comment 5.

- **NOGRAPH:** I; 0,4; -; Variable to control automatic closing of model run monitoring window.

1. Window remain open until user clicks close button.

2. The model windows will automatically close as soon as the run finalizes.

- **NUNITS:** I; 0,1; -; Variable to indicate unit system:

1. Metric units.
2. English units.

- **RELEASE:** I; -; -; Release number ID used internally for reference. Should not be modified.
- **TLIMT:** R; > 0; h.; Total simulation time.
- **TOUT:** R; $\leq TLIMT$; h.; Output time interval for reporting results.
- **XNMAN:** R; [0.1-2]; -; Manning's n coefficient multiplier. See comment 6.

11.0.1.2 Comments for the .DAT file

1. Setting the CFL (Courant Friederich-Lewy) or Courant number is critical for adequate stability and ensure mass conservation. OilFlow2D 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:

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

where Δt = DT is the time-step, Δx is a measure of the minimum triangular 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, OilFlow2D 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 data file for each node in the mesh.

4. Use the IPROFILE option to allow OilFlow2D to generate results along a polyline. The polyline and other required data should be given in the Profiles file , which is defined later in this document.

5. Use this option to allow OilFlow2D to generate results along prescribed cross sections. The cross sections and other required data should be given in Cross Section file 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 cell 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. The user can select whether the model will calculate bass balance or not. This has implications particularly in the GPU model since mass balance calculations are done in the CPU, with the resulting performance overhead and runtime increase. Yu may want to turn it on to review how the model is conserving volume or mass. Once that is checked, it is recommended to turn it off for maximum performance.

11.1 Mesh Data

11.1.1 Mesh Data File: .FED

This file contains the data that defines the triangular-cell mesh, and includes node coordinates, connectivity for each triangular cell, node elevations, Manning's n coefficients and other parameters. This file is created OilFlow2D. OilFlow2D assures that the file will be created error-free and consistent with the boundary conditions and other mesh parameters. Editing this file outside OilFlow2D may introduce unexpected errors.
Line 1: Number of cells 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 cell parameters.
IE NODE(IE,1) NODE(IE,2) NODE(IE,3) MANNINGN(IE) ELZB(IE) ELINITWSE(IE) ELMINERODELEV(IE)

11.1.1.1 Example of a .FED file

```
   1965   1048    5    5
1  243401.515   94305.994   51.071  0.000   -9999.000   0   0
2  243424.157   94325.674   49.833  0.000   -9999.000   0   0
3  243446.800   94345.354   49.136  0.000   -9999.000  12   0.025
4  243469.443   94365.034   48.879  0.000   -9999.000   0   0
5  243503.168   94394.347   51.662  0.000   -9999.000  12   0.025

...

1044  243830.638   93310.994   48.603  0.000   -9999.000   6   QIN.DAT
1045  243492.493   93320.046   49.987  0.000   -9999.000   6   QIN.DAT
1046  243693.660   93297.785   47.390  0.000   -9999.000   0   0
```

```
1047  243964.332   93388.332   50.843  0.000   -9999.000   0   0
1048  243861.431   93893.192   50.863  0.000   -9999.000   0   0
1   456   987   188   0.035   51.395  0.000   -9999.000  0.000
2   478   183   809   0.035   49.778  0.000   -9999.000  0.000
3   336    37   869   0.035   53.992  0.000   -9999.000  0.000
4   601   393    97   0.035   53.486  0.000   -9999.000  0.000
5   456   509   987   0.035   51.690  0.000   -9999.000  0.000
...

1961  1024   972    23   0.035   47.480  0.000   -9999.000  0.000
1962   930  1028   377   0.035   48.126  0.000   -9999.000  0.000
1963  1028   960   377   0.035   48.385  0.000   -9999.000  0.000
1964  1043  1017   426   0.035   51.994  0.000   -9999.000  0.000
1965   850    78    77   0.035   49.715  0.000   -9999.000  0.000
```

This mesh has 1965 cells, 1048 nodes.

- **BCTYPE:** I; -; -; Code to indicate type of open boundary. See further details about boundary conditions on the file description below.
- **BCFILENAME:** S; < 26; -; Boundary condition file name. Should not contain spaces and must have less than 26 characters. See further details on the file description below.
- **DUMMY:** I; -; -; Always equal to 2.
- **ELINITWSE(IE):** R; -; m or ft; Initial water surface elevation for cell EL. Used in OilFlow2D and OilFlow2D GPU.
- **ELMINERODELEV (IE):** R; ≥ 0; -; Minimum erosion elevation allowed at each cell. Used in OilFlow2D and OilFlow2D GPU.
- **ELZB (IE):** R; -; m or ft; Initial bed elevation for cell EL. Used in OilFlow2D and OilFlow2D GPU.
- **INITWSE(IN):** R; -; m or ft; Initial water surface elevation for node IN.
- **IE:** I; > 0; -; Cell 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 cell IE.
- **MINERODELEV (IN):** R; ≥ 0; m or ft; Minimum erosion elevation allowed at each node.
- **NELEM:** I; 1-5; -; Number of triangular cells.
- **NNODES:** I; > 0; -; Number of nodes.
- **NODE(IE,1), NODE(IE,2), NODE(IE,3):** I; > 0; -; Node numbers for cell 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.

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

These files contain boundary condition data used only internally by the model. Both files are internally generated by OilFlow2D. The format of the file is as follows
Line 1: Number of nodes on external boundary.

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

11.1.2.1 Example of .IFL file

1165
365 1 WSE97out.TXT
367 1 WSE97out.TXT
431 1 WSE97out.TXT

This 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 .
The format of the 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)

11.1.2.2 Example of a .OBCP file

2
12
UNIF1.DATP
24
2916
...
3299

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

This 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 , 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 , and there are 17 nodes on the boundary.

- **BCTYPE:** I; -; -; Code to indicate type of open boundary. See Table and comment 1.
- **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.

lp9.9cm

- **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.
- **6:** Imposes water discharge. An associated boundary condition file must be provided. See comment 2.
- **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.
- **13-16:** For future use.
- **17:** Imposes Water Surface Elevation. This condition is similar to BCTYPE 1, but it forces perpendicular velocity to the input line. An associated boundary condition file must be provided. See comments 2 and 4.
- **19:** Imposes single-valued stage-discharge rating table along an internal polyline. An associated boundary condition file must be provided. See comment 8.

11.1.2.3 Comments for the .IFL and .OBCP files

1. OilFlow2D 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 OilFlow2D 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 helps determining which conditions to use for most applications.

    - **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 inaccuracies due to violation of the theoretical boundary condition requirements of the shallow water equations.

2. When imposing a single variable (water surface elevation, or discharge Q), 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 Q, 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 OilFlow2D 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 OilFlow2D version.

7. On free outflow condition boundaries, the model calculates velocities and water surface elevations applying the full equations from the internal cells. 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. 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 cells adjacent to the boundary line, which may result in adverse slopes or slopes that do not capture the general trend the reach.

10. 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 cells adjacent to the boundary line. The condition then can be visualized as if the given discharge enters over the inflow cells. For each time, the model evenly divides the discharge between all the inflow cells. For example if there are Ne inflow cells and the imposed discharge is Qin, each cell 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 cells have lower bed elevations than the surrounding cells. 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.

11.1.3 Mesh Boundary Data File: .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**

internally generated by OilFlow2D.
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)

11.1.3.1 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.

- **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.

11.1.3.2 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.

11.2 Bridges

OilFlow2D 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 cells, 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.

11.2.1 Bridges Data File: .BRIDGES

This component requires the data file that is internally generated by the model based on the geometrical representation entered in the OilFlow2D. The 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 cell numbers along bridge alignment.
CELL_A(1) CELL_B(1)

…
CELL_A(NC) CELL_B(NC)

11.2.1.1 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
```

- **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.

11.2.2 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)$\leq$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.

11.2.2.1 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.

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

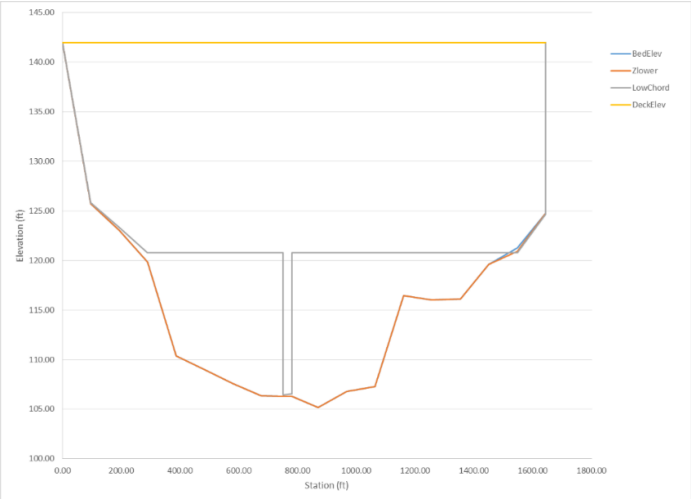


Figure 11.2 – Front view of a bridge cross section.

- **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.

11.3 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: Culvert file version number.
CULFILEVER
Line 2: Number of culverts.
NCULVERTS
FOR EACH CULVERT (NCULVERTS):
IF (CULFILEVER = 202208)
CulvertID
CulvertType
IF (CulvertType is 0, 1, 2, -3, -4, -5)
CulvertFile
X1 Y1 X2 Y2
ELSE IF (CulvertType is 11, 12, -14, -15)
CulvertFile
NcellsUPS cellID_L_1 cellID_L_2 … cellID_L_NcellsUPS
NcellsDNS cellID_R_1 cellID_R_2 … cellID_R_NcellsDNS
ENDIF (CulvertType)
ELSE
CulvertID
CulvertType
CulvertFile
X1 Y1 X2 Y2
ENDIF (CULFILEVER)
END (NCULVERTS)

11.3.1 Example of a .CULVERTS file

```
202208
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
```

- **CULFILEVER:** I; -; -; Culvert file version number. Current version is 202208.
- **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, 11, 12,-3,-4,-5,-14,-15; -; Culvert type. See comments 1 and 2.
- **NCULVERTS:** I; > 0; -; Number of culverts.
- **NcellsUPS:** I; > 0; -; Number of upstream exchange cells.
- **NcellsDNS:** I; > 0; -; Number of downstream exchange cells.
- **X1 Y1 X2 Y2:** R; -; m or ft; Vertex coordinates defining each culvert line.

11.3.2 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.

11.3.2.1 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
```

- **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 cekk.
- **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 cell.
- **DEPTH:** R; > 0; m or ft; Water depth.
- **Q:** R; > 0; m^3/s or ft^3/s; Water discharge.

11.3.3 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

11.3.4 Example of the culvert characteristic data file

```
202208
 1
 0.5
 0.012
 1
 1
 1.1
 0.6
```

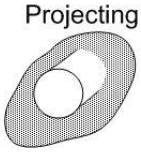
```
-0.5
 0.10
 5.0
 1.0
```

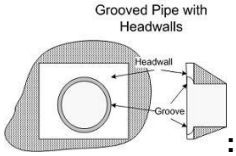
This example culvert characteristics data file indicates that the culvert one barrel (Nb =1), Ke=0.4, nc=0.012, Kp=1, cp =1, M =1.1, Y=0.6, m=-0.5, and Dc=0.10, INVERT_Z1=5.0 and INVERT_Z2 = 1.0.

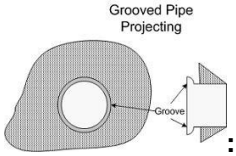
- **Nb:** I; -; -; Number of identical barrels. The computed discharge for a culvert is multiplied by Nb to obtain the total culvert discharge.
- **Ke:** R; 0-1; -; Entrance Loss Coefficient given in Table .
- **nc:** R; 0.01-0.1; -; Culvert Manning's n Coefficient given in Table .
- **K':** R; 0.1-2.0; -; Inlet Control Coefficient given in Table .
- **M:** R; 0.6-2.0; -; Inlet Control Coefficient given in Table .
- **c':** R; 0.6-2.0; -; Inlet Control Coefficient given in Table .
- **Y:** R; 0.5-1.0; -; Inlet Control Coefficient given in Table .
- **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 cell.
- **Good joints, smooth walls:** 0.012
- **Projecting from fill, square-cut end:** 0.015
- **Poor joints, rough walls:** 0.017
- **2-2/3 inch $\times$ 1/2 inch corrugations:** 0.025
- **6 inch $\times$ 1 inch corrugations:** 0.024
- **5 inch $\times$ 1 inch corrugations:** 0.026
- **3 inch $\times$ 1 inch corrugations:** 0.028
- **6 inch $\times$ 2 inch corrugations:** 0.034
- **9 inch $\times$ 2 1/2 inch corrugations:** 0.035

- **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
- **Without grate:** 0.5
- **With grate:** 0.7
- **Corrugated metal pipe:** 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
- 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
- **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

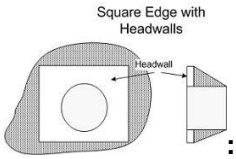
- **Concrete:** Circular; Beveled ring; 45° bevels; 0.1379; 2.5000; 0.9651; 0.7400
- **Concrete:** Circular; Beveled ring; 33.7° bevels; 0.1379; 2.5000; 0.7817; 0.8300
- **Concrete:** Rectangular; Wingwalls; 30° to 75° flares; square edge; 0.1475; 1.0000; 1.2385; 0.8100
- **Concrete:** Rectangular; Wingwalls; 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; 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 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
- **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
- 

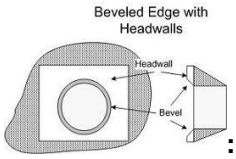
: End of the culvert barrel projects out of the embankment.
- 

: Grooved pipe for concrete culverts decreases energy losses through the culvert entrance.
- 

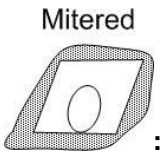
: This option is for concrete pipe culverts.

- 

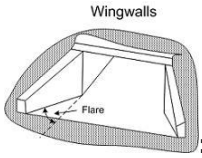
: Square edge with headwall is an entrance condition where the culvert entrance is flush with the headwall.

- 

: '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.

11.3.5 Comments for the .CULVERTS and culvert characteristics files

1. The type of culvert and its flow condition is defined through the CulvertType parameter as follows:

   CulvertType = 0: [000] Discharge calculated by rating curve (Q vs inlet depth). Only inlet and outlet cells are used for volume exchange.

   CulvertType = 1: [001] Rectangular/box section culvert. Only inlet and outlet cells are used for volume exchange.

   CulvertType = 2: [002] Circular section culvert. Only inlet and outlet cells are used for volume exchange.

   CulvertType = 11: [011] Rectangular/box section culvert. Inlet and outlet cells plus neighboring cells are used for volume exchange.

   CulvertType = 12: [012] Circular section culvert. Inlet and outlet cells plus neighboring cells are used for volume exchange.

   CulvertType = -3: [100] Discharge calculated by rating curve (Q vs inlet depth). Only inlet and outlet cells are used for volume exchange. Only flow from (X1,Y1) to (X2,Y2) is allowed.

   CulvertType = -4: [101] Rectangular/box section culvert. Only inlet and outlet cells are used for volume exchange. Only flow from (X1,Y1) to (X2,Y2) is allowed.

CulvertType = -5: [102] Circular section culvert. Only inlet and outlet cells are used for volume exchange. Only flow from (X1,Y1) to (X2,Y2) is allowed.

CulvertType = -14: [111] Rectangular/box section culvert. Inlet and outlet cells plus neighboring cells are used for volume exchange. Only flow from (X1,Y1) to (X2,Y2) is allowed.

CulvertType = -15: [112] Circular section culvert. Inlet and outlet cells plus neighboring cells are used for volume exchange. Only flow from (X1,Y1) to (X2,Y2) is allowed.

2. For CulvertType 0, culvert discharge is computed using a given rating table on the CulvertFile file.

3. For CulvertType 1, 2, 11, 12, -4, -5, -14, and -15 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).

11.4 Dam Breach Data File: .DAMBREACH

This component requires the data file that is generated by the QGIS plugin. The file has the following format:
Line 1: Dam breach file version number.
DBFVERSION
Line 2: Number of dams.
NUMBEROFDAMS
Then for each dam it follows NUMBEROFDAMS group of lines with the following data:
Dam name.
DAM_ID
Failure mode.
DAM_FAILMODE
Dam breach center coordinates.
X0 Y0
Dam breach definition parameters
ZC Angle CD t_initial zb0 d50 tau_c k_sm k_d Gs Porosity C damCrestWidth UpstreamSlope DownstreamSlope
Dam breach file
DAMBREACHFILE
Number of cells pairs along the dam alignment.
NC
NC lines containing pairs of cell numbers along dam alignment.
CELL_A(1) CELL_B(1)
…
CELL_A(NC) CELL_B(NC)

11.4.1 Example of a .DAMBREACH file

202208

```
1
DAMBREACH1
1
5300.0  600.0
216.3  45.0  0.601  0.0   0.0   0.0   0.0   0.0   0.0   0.0   0.0   0.0   0.0   0.0
0.0
DambreachFile1.dat
7
332  334
69  335
67  349
65  358
50  360
41  363
4  378
```

- **DBFVERSION:** I; > 0; –; File version number, e.g. 202208.
- **NUMBEROFDAMS:** I; > 0; –; Number of dams.
- **DAM_ID:** S; < 26; –; Bridge ID.
- **DAM_FAILMODE:** I; –; 1, 2, 3; Failure mode:
- Prescribed failure.
- Overtopping Erosion.
- Piping Erosion.
- **X0, Y0:** R; –; [m or ft]; Dam-breach center coordinates. These coordinates are calculated by the model using the distance from one of the dam polyline end points given in the QGIS and DIP dialogs.
- **ZC:** R; –; [m or ft]; Initial dam crest elevation.
- **Angle:** R; [5, 90]; –; Breach side slope angle with respect to the horizontal.
- **CD:** R; –; –; Non-dimensional breach discharge coefficient.
- **T_initial:** R; –; h.; Breach start time
- **Zb0:** R; –; [m or ft]; Initial elevation of breach bottom
- **D50:** R; –; [m or ft]; Mean dam material diameter.
- **Tau_c:** R; –; [Pa or lb/in^2]; Critical shear stress.
- **K_sm:** R; –; –; Submergence correction for tailwater effects.
- **Kd:** R; –; [m^3/(N s) or ft^{2}s/lb]; Erosion coefficient.
- **Gs:** R; –; –; Dam material specific gravity.

- **Porosity:** R; (0,1); -; Dam material porosity given in fractions of 1, e.g. 0.4.
- **C:** R; –; [Pa or lb/in^2]; Dam material cohesion.
- **DamCrestWidth:** R; –; [m or ft]; Dam crest width.
- **UpstreamSlope:** R; [0-1]; –; Dam upstream slope.
- **DownstreamSlope:** R; [0-1]; –; Dam downstream slope.
- **DAMBREACHFILE:** S; < 26; –; Used only for the prescribed failure mode (1) but a dummy text should be given always for failure modes 2 and 3. The file contains the time series of the breach width and height opening. File name should contain no black spaces. See details in sections and .
- **NC:** I; > 0; –; Number of cell pairs along the dam alignment.
- **CELL_A(i) CELL_B(i):** I; –; –; Cell pair along dam alignment.

11.4.2 Breach time evolution data file for prescribed failure mode

For the prescribed failure model 1, the breach temporal evolution file is necessary to define the width and height of the breach opening for each time. The format is described as follows:
Line 1: Number of times.
NT
NT lines with these entries:
TIME(1) WIDTH(1) HEIGHT(1)
...
TIME(NT) WIDTH(NT) HEIGHT(NT)

11.4.2.1 Example of the breach time evolution data file (Prescribed Failure Mode only)

```
3
0  1  1
0.25      20        25
1         20        25
```

11.4.3 Comments for the .DAMBREACH file

These are the breach definition parameters required for each failure mode:

- **Prescribed:** ZC, Angle, and CD.
- **Overtopping erosion:** ZC, Angle, CD, t_initial, zb0, d50, tau_c, k_sm, k_d.
- **Piping erosion:** ZC, Angle, CD, t_initial, zb0, d50, tau_c, k_sm, k_d, Gs, Porosity, C, damCrestWidth, UpstreamSlope, DownstreamSlope.

Note that in the line containing the *Dam breach definition parameters* in the file always have 15 values, even when not all of them are used for the Prescribed and Overtopping modes.

11.5 GATES Data Files: .GATES

This component requires the data file that is internally generated by the model based on the geometrical representation entered in the OilFlow2D QGIS plugin. The 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)

11.5.1 Example of a .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
```

- **Cd:** R; > 0; -; Non-dimensional 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.

11.5.2 Gate Aperture Time Series File

Line 1: Number of points in time series of gate aperture data.
NPOINTS
NPOINTS lines containing:
Time and aperture.
TIME H(I)

11.5.3 Example of a Gates Aperture Data File

```
3
0  0.0
2  0.5
4  1.0
```

- **NPOINTS:** I; > 1; -; Number of data points in the gate aperture time series.
- **TIME:** R; > 0; h.; Time.
- **H(I):** R; -; m or ft; Gate aperture for the corresponding time.

11.6 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-cell 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 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)

11.6.1 Example of a .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_B.DAT
799482.440  309453.678
799135.525  309118.164
798914.020  309269.634
798787.701  309467.583
```

This file indicates that there are 2 internal rating table polylines, the ID of the first one is IRT_A, which has 4 vertices, BCTYPE 19 and file name.

- **IRT_NPL:** I; > 0; -; Number of IRT polylines.
- **IRT_NV:** I; ≥ 2; -; Number of points defining each IRT polyline.
- **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; Vertex coordinates 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.

11.6.2 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.

11.7 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 polygons where rainfall time series are defined.
NP
NP 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))

11.7.0.1 Example of a .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 and the polygon is defined by four vertices.

- **NPZONE(i):** I; ≥ 1; -; Number of vertices defining polygon i.
- **NP:** I; -; -; Number of polygons.
- **RAINEVFILE:** S; ≤ 26; -; Rainfall intensity. See comment 1.
- **X(i) Y(i):** R; > 0; m or ft; Vertex coordinates of i polygon.

11.7.1 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.

11.7.1.1 Hyetograph and Evaporation data file

Line 1: Number of points in time series of rainfall and evaporation.
NPRE
NPRE lines containing:
Time Rainfall intensity, Evaporation intensity.
TIME RAININT EVAPINT

11.7.2 Example of a Hyetograph and Evaporation data file

```
8
0.0  0.0  0.01
1.0  1.0  0.02
3.0  4.0  0.02
6.0  12.0  0.00
6.2  7.0  0.00
7.0  3.0  0.0
7.1  0.0  0.0
9.0  0.0  0.0
```

- **EVAPINT:** R; ≥ 0; mm/h or in/h; Evaporation intensity. See comment 1.
- **NPRE:** I; -; -; Number of times in rainfall and evaporation time series.
- **RAININT:** R; ≥ 0; mm/h or in/h; Rainfall intensity. See comment 1.
- **TIME:** R; > 0; hours; Time interval

11.7.3 Comments for the Hyetograph and Evaporation data file

1. To calculate the rainfall/evaporation over the mesh, the model will use rainfall and evaporation intensities given for each time interval. For instance in the example above, for all times between 1 and 3 hours, the rainfall intensity will be equal to 1 mm/h and evaporation intensity equal to 0.02 mm/h. For times between 3 and 6 hours the rainfall intensity will be equal to 1 mm/h and evaporation intensity equal to 0.02 mm/h, and so on for other times.
2. If the user has a file in the project folder, the program will apply the data contained in that file to all cells whose centroid falls outside the polygons given in the *RainEvap* layer, and not covered by any other polygon.

11.8 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))

11.8.1 Example of a .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 and the polygon is defined by four vertices.

- **&:** 7
- **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.

11.8.2 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.

11.8.2.1 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 f_c 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 AMC**

11.8.3 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.

- **AMC:** I; > 0; 1, 2, 3; Antecedent Moisture Content (AMC). Represents the preceding relative moisture of the soil prior to the storm event. Allows accounting for variation of CN for different storm events, or initial soil moisture for a given event using Eqs. and. See possible AMC values in Table .
- **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; -; Number of data parameters depending on the infiltration model selected. Should be set as follows: 3 for Horton of Green and Ampt, and for SCS-CN methods.
- **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.

).

- **Less than 13 mm:** Less than 36 mm

- **2:** 13 mm to 28 mm; 36 mm to 53 mm
- **3:** More than 28 mm; More than 53 mm

11.9 Manning's n Variable with Depth Data File: .MANNN

This file is created by the OilFlow2D QGIS plugin based on the data you enter in the *ManningsNz* layer. 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))

11.9.0.1 Example of a .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 and the polygon is defined by four vertices.

- **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.

11.9.1 Comments for the .MANNN file

1. The spatial distribution of Manning's n variable with depth is given as a number of polygons that cover the mesh area. Polygon borders may touch each other or leave a small gap; the model assigns Manning's n data to a cell based on which polygon contains the cell centroid, so exact boundary matching is not required. Zones not covered by any polygon (complementary area) are assigned data from the DefaultManningsn.DAT file (see below).
2. When the .MANNN file is in use (IMANN=2 in the .DAT run control file), values defined in the regular Manning N layer (the .MannN2 file) are ignored. The Manning N layer itself does not need to be deleted from the project.

11.9.1.1 Manning's n variable with depth data file

Line 1: Number of points in Manning's n file.
NP
NP lines containing:
DEPTH(i) MANNINGS_N(i)

11.9.2 Example of a Manning's variable with depth data file

```
3
0.  0.1
0.3  0.1
1.0  0.03
```

- **DEPTH(i):** R; ≥ 0; m or ft; Flow depth. See comment 1.
- **MANNINGS_N(i):** R; ≥ 0; -; Manning's n corresponding to DEPTH(i). See comment 1.
- **NP:** I; -; -; Number values in file.

11.9.3 Comments for the Manning'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.

11.9.3.1 Default Manning's n data file: `DefaultManningsn.DAT`

When the .MANNN file does not cover the entire mesh, the complementary area is assigned data from a DefaultManningsn.DAT file placed in the project folder. The model loads this file automatically. If it is not present, the model applies a default value of 0.035 to the complementary area.

The file format is identical to the inner Manning's n variable with depth data file documented above:

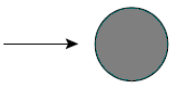
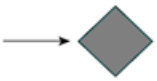
```
Line 1: Number of points (NP).
NP lines: DEPTH(i) MANNINGS_N(i)
```

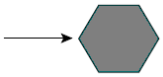
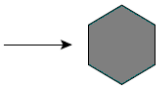
11.9.4 Bridge Piers Drag Forces File: .PIERS

This option requires the data file that is internally generated by the model based on the geometrical representation entered in the OilFlow2D QGIS plugin. The 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

11.9.4.1 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
...
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
```

- **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.
- **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.
- **Round cylinder:** 
- **Square cylinder:** 
- **Square cylinder:** 

- **Square cylinder:** R/B; C_D
- **with:** 0; 2.2
- **rounded corners:** 0.02; 2.0
- **& 0.17:** 1.2
- **& 0.33:** 1.0
- **Hexagonal cylinder:** 
- **Hexagonal cylinder:** 
- **& L/B:** C_D
- **& 1:** 1.0
- **& 2:** 0.7
- **& 4:** 0.68
- **& 6:** 0.64
- **& L/B:** C_D
- **& 1:** 2.2
- **& 2:** 1.8
- **& 4:** 1.3
- **& 6:** 0.9

11.9.4.2 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:

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

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.

To account for the drag force that the pier exerts on the flow, OilFlow2D converts it to the distributed shear stress on the cell where the pier centroid coordinate is located. The resulting pier shear stress expressions in x and y directions are as follows:

$$\tau_{px} = \frac{1}{2} C_D \rho U \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right)$$

$$\tau_{py} = \frac{1}{2} C_D \rho V \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right)$$

where A_e is the cell area.

11.10 Bridge Pier and Scour Data File: .SCOUR

This file stores data required to compute scour around bridge piers and abutments.
Line 1: Number of piers and abutments.
NP
NP groups of lines containing the following data:
Imode
PierID
Icomp
XA, YA
Y1
V1
Fr1
alfa
ishape
L
a
iBedCondition
D50
D84
Vcritical
SedimentSpecificDensity
WaterSpecificDensity
FrD
K1
K2
K3
K
theta
ys
W
Wbottom
iAbutmentType
AlfaA
AlfaB
YmaxLB
YmaxCW
YcLB
YcCW1

YcCW2
YsA
q1
q2c
n Manning
Tauc
BridgeXSEC_X1, BridgeXSEC_Y1, BridgeXSEC_X2, BridgeXSEC_Y2
UpstreamXSEC_X1, UpstreamXSEC_Y1, UpstreamXSEC_X2, UpstreamXSEC_Y2

11.10.1 Example of a .SCOUR file

```
2
DrainA
2
Drain.TXT
799019.633  309402.572
DischargeIn
1
Discharge.TXT
799222.740  309048.493
```

- **Pier ID:** S; & -; Pier name
- **Icomp:** I; (1, 2, 3, 4); & Computational method
- **XA, YA:** R; -; & Pier coordinates
- **Y1:** R; > 0; m, ft; Flow depth directly upstream of the pier
- **V1:** R; > 0; m/s, ft/s; Velocity upstream of the pier
- **Alfa:** R; [0, 180]; Degrees; Angle of attack
- **alfaRAD:** R; [0, Pi]; Radians; Angle of attack
- **ishape:** I; &; Pier shape
- **L:** R; > 0; m, ft; Pier length
- **a:** R; > 0; m, ft; Pier width
- **iBedCondition:** I; &; Bed condition
- **D50:** R; > 0; m, ft; D50
- **D84:** R; > 0; m, ft; D84
- **Sediment Specific Density:** R; (0,3); & Ss
- **Water Specific Density:** R; (0,1.2]; & Sw
- **K1:** R; &; Correction factor for pier nose shape.
- **K2:** R; &; Correction factor for angle of attack of flow
- **K3:** R; &; Correction factor for bed condition
- **K:** R; (0,3); (0,3); bottom width relative to Ys.
- **theta:** R; 20-48°; Degrees; angle of repose of the bed material
- **ys:** R; ≥ 0; m, ft; Scour depth
- **W:** R; ≥ 0; m, ft; scour hole top width

- **Wbottom:** R; ≥ 0; m, ft; scour hole bottom width
- **Fr1:** R; > 0; & Froude Number upstream of pier
- **FrD:** R; > 0; & Densimetric particle Froude Number
- **SIGMA:** R; > 0; & Sediment gradation coefficient
- **Vc:** R; > 0; m/s/, ft/s; Critical velocity for initiation of erosion of the material
- **iAbutmentType:** I; [1-2]; -; Abutment Type
- **AlfaA:** R; [1-2]; -; Amplification factor for live-bed conditions
- **AlfaB:** R; [1-2]; -; Amplification factor for clear-water conditions
- **YmaxLB:** R; ≥ 0; m or ft; Maximum flow depth after scour for live-bed conditions
- **YmaxCW:** R; ≥ 0; m or ft; Maximum flow depth after scour for clear-water conditions
- **YcLB:** R; ≥ 0; m or ft; Depth including live-bed contraction scour
- **YsA:** R; &; Abutment scour depth
- **YcCW1:** R; ≥ 0; m or ft; Depth including clear-water contraction scour. Method 1
- **YcCW2:** R; ≥ 0; m or ft; Depth including clear-water contraction scour. Method 2
- **q1:** R; ≥ 0; m^2/s or ft^2/s; Upstream unit discharge
- **q2c:** R; ≥ 0; m^2/s or ft^2/s; Upstream unit discharge of the constricted opening
- **n Manning:** R; ≥ 0.01; -; Manning's n
- **TauC:** R; ≥ 0; Pa or ln/ft^2; Critical shear stress
- **GammaW:** R; ≥ 0; N/m^3 or lb/ft^3; Unit weight of water
- **BridgeXSEC_X1, BridgeXSEC_Y1, BridgeXSEC_X2, BridgeXSEC_Y2:** R; -; m of ft; Extreme point coordinates of Bridge Cross Section
- **UpstreamXSEC_X1, UpstreamXSEC_Y1, UpstreamXSEC_X2, UpstreamXSEC_Y2:** R; -; m or ft; Extreme point coordinates of Upstream Cross Section

11.10.2 Comments for the .SCOUR 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.
2. To model inflows use positive discharge values, and to model outflows use negative values.

11.11 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 or rating table, and the coordinates of the point as follows:
SOURCEID
SOURCETYPE
ISFILENAME
X_S(I) Y_S(I)
...

11.11.1 Example of a .SOURCES file

```
2
DrainA
2
Drain.TXT
799019.633  309402.572
DischargeIn
1
Discharge.TXT
799222.740  309048.493
```

This file indicates that there are 2 sources/sinks. The first one is named DrainA located at coordinate: X = 799019.633 and Y = 309402.572 and is SOURCETYPE 2, indicating that the data file contains a rating table of depth vs discharge for the drain. The second source is DischargeIN and is type 1 where a hydrograph (time vs discharge) is given in.

- **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.
- **SOURCETYPE:** I; $1, 2$; -; Type of data for the source or sink. If equal to 1, the file should contain a hydrograph. If equal to 2, it contains a rating table with depths vs discharge values.
- **X_S Y_S:** R; -; m or ft; Coordinates of source/sink.

11.11.2 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.
2. To model inflows use positive discharge values, and to model outflows use negative values.

11.12 Multiple Sources file

This file helps facilitating the input of many inflow sources and is typically used when simulating multiple spills from a pipeline. The file can be read in QGIS using the *Import Multi-sources file* on the Tools OilFlow2D. You can prepare the file in any text editor using the following format.
Line 1: Number of sources.
NSOURCEP
NSOURCESP lines containing source/sink point identification text, point X, Y coordinates, and name of the file containing the discharge time series or rating table for each point as follows:
SOURCEID X_S(I) Y_S(I) ISFILENAME
...

11.12.1 Example of a multiple source file

```
9
Source1  6232789.844  1941100.871  SOURCE_1.txt
Source2  6231510.593  1939867.858  SOURCE_2.txt
Source3  6230662.896  1938943.098  SOURCE_3.txt
Source4  6230154.278  1936954.865  SOURCE_4.txt
Source5  6229214.106  1935136.170  SOURCE_5.txt
Source6  6227179.634  1933764.443  SOURCE_6.txt
Source7  6224158.752  1931853.273  SOURCE_7.txt
Source8  6221877.678  1930758.974  SOURCE_8.txt
Source9  6219519.540  1928847.803  SOURCE_9.txt
```

This file indicates that there are 9 sources/sinks. The first one is named Source1 located at coordinate: X = 6232789.844 and Y = 1941100.871 and is the corresponding data file that contains the source discharge vs time series.

- **NSOURCESP:** I; > 0; -; Number of source or sink points.
- **ISFILENAME:** S; -; -; Name of file containing the time series of each source. Must not contain blank spaces.
- **SOURCEID:** S; < 26; -; Name of point source or sink. Must not contain blank spaces.
- **X_S Y_S:** R; -; m or ft; Coordinates of source/sink.

Once you have generated the file, you can use the *Import Multi-sources file* tool to populate the *Sources* layer:

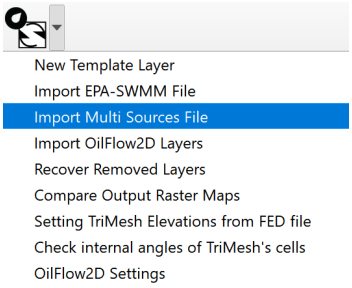


Figure 11.3 – Multi-sources file read.

Then in the Multi-sources dialog, enter the file name, and select to create a new *Sources* layer, or to add the sources to an existing *Sources* layer.

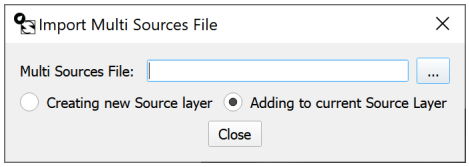


Figure 11.4 – Multi-sources dialog.

11.13 Weirs Data Files: .WEIRS and .WEIRP

These data files allow 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 model 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)

11.13.1 Example of a .WEIRS file

```
4
WEIR_1
10  0.60  155.000
6217603.64  1925043.47  155.000
6217585.08  1925060.22  155.000
6217566.52  1925076.97  155.000
6217547.97  1925093.72  155.000
6217529.41  1925110.47  155.000
6217510.85  1925127.22  155.000
6217492.29  1925143.97  155.000
6217473.73  1925160.72  155.000
6217455.17  1925177.47  155.000
6217428.80  1925201.27  155.000
WEIR_2
8  0.60  155.000
6217496.72  1924525.22  200.000
6217475.99  1924539.18  200.174
6217455.25  1924553.15  200.348
6217434.52  1924567.11  200.522
6217413.78  1924581.08  200.695
6217377.46  1924605.54  201.000
6217353.53  1924612.77  201.229
6217329.60  1924620.01  201.457
```

This file indicates that there are 2 weirs. The first one is named WEIR_A and is defined by a polyline with 4 vertices. Weir non-dimensional discharge coefficient is equal to 0.60.
The file is generated by the model based on the file and has a similar structure. They differ in that the , instead of the list of vertices, has the list of triangular cell pairs at each side of the weir.
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
NC CD WRELEVCELL
CELL_R(1) CELL_L(1) WRCREST(1)
CELL_R(2) CELL_L(2) WRCREST(2)
...
CELL_R(NC) CELL_L(NC) WRCREST(NC)

11.13.2 Example of a .WEIRP file

```
      4
WEIR_1
0.60  -999
9
8409          8851    155.000
8636          8677    155.000
8618          8705    155.000
8613          8703    155.000
8647          8602    155.000
8571          8841    155.000
8727          8809    155.000
8826          8824    155.000
8828          8731    155.000
WEIR_2
0.60  -999
7
3233          3212    200.087
3230          3202    200.261
3221          3193    200.435
3123          3189    200.608
3416          3112    200.847
3262          3762    201.114
3053          2980    201.343
```

- **CD:** R; > 0; -; Non-dimensional weir discharge coefficient. See comment 1.
- **CELL_R(I) CELL_L(I):** I; -; -& Cells at each side of the polyline.
- **NWEIRS:** I; > 0; -; Number of weir polylines.

- **NC:** I; ≥ 2; -; Number of cell pairs along each weir polyline.
- **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.
- **WRELEVCELL:** R; -; m of ft; Weir crest elevation for all the weir. If WRCRESTELEVCELL = -999 a weir elevation is provided for each weir polyline vertex.
- **WRCRESTELEV:** R; -; m of ft; Weir crest elevation for all the weir. If WRCRESTELEV = -9999 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; Vertex coordinates defining each weir polyline. See comment 2.

11.13.3 Comments for the .WEIRS File

1. Weir discharge is computed between pairs of nodes along the polyline based on the following formula:

$$Q = C_d \frac{2}{3} \sqrt{2g} L H^{3/2}$$

   where L is the distance between nodes, H is the total head upstream of the polyline segment and C_d is the non dimensional discharge coefficient that takes values between 0.611 and 1.1. The model checks for submergence and it occurs C_d will be corrected according to the correction factor defined by (FHWA, 2001).

2. Weir polylines should be defined avoiding abrupt direction changes (e.g. $\geq$ 90 degree turns), because such angles may create errors in the algorithm that identifies the nodes that lie over the polyline.

11.14 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))

11.14.1 Example of a 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 and the polygon is defined by four vertices.

- **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 = 0.001. See comments for guidance to compute CD.
- **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.

11.14.2 Comments for the 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.
3. The following formula was proposed by Garrat (1971) to compute CD: CD = $(0.75+0.067W)10^{-3}$, where W is the wind velocity in m/s.

11.14.3 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

11.14.4 Example of a Wind Velocity and Data File

```
3
0.  0.0  0.0
24  4.0  -3.0
48  4.0  -3.0
```

- **NPOINTS:** I; > 1; -; Number of data points in the wind velocity time series.
- **TIME:** R; > 0; h; Time.
- **UX(I) UY(I):** R; -; m/s or ft/s; Wind velocity components in x and y directions.

11.15 Oil Containment Booms Data File: .BOOMS

This data file contains the data to implement oil containment booms in the OilFlow2D model. They are represented entering polylines in the *SpillBooms* layer. The user can select the boom type, and skirt height.
Line 1: Number of boom polylines.
NBOOMS
NBOOMS group of lines including weir ID, number of vertices defining each boom polyline, the boom type, skirt height, and fraction loss, followed by the coordinates each vertex as shown:
BOOM_ID
BOOM_TYPE
FUTURE_USE TRAPPING_FRACTION TUG_VEL_U TUG_VEL_V 0.0 0.0 0.0 0.0 0.0 0.0
NV
X_B(1) Y_B(1)
X_B(2) Y_B(2)
...
X_B(NV) Y_B(NV)

11.15.1 Example of a .BOOMS file

```
2
BOOM_1
1
1.0   0.9   0.0    0.0    0.0    0.0    0.0    0.0    0.0    0.0
5
4
799429.362  308905.287  200.
799833.895  308354.857  201.
```

```
799986.424  307738.111  202.
799847.158  307141.259  203.
BOOM_1
2
1.0  0.5   0.0    0.0    0.0    0.0    0.0    0.0    0.0    0.0
5
4
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 indicates that there are 2 booms. The first one is named BOOM_1, it is Type 1) and is defined by a polyline with 4 vertices.

- **BOOM_TYPE:** I; $[1, 4]$ DEFAULT 1; -; Boom type defined as follows
- Curtain
- Fence
- Sorbent
- Bubble barrier
- **TRAPPING_FRACTION:** R; > 0 DEFAULT 0.0; -; Oil fraction retained by boom with respect of the oil captured.
- **NBOOMS:** I; > 0; -; Number of boom polylines.
- **NV:** I; ≥ 2; -; Number of points defining each boom polyline.
- **BOOM_ID:** S; < 26; -; Boom's name. Should have less than 26 characters and must not contain blank spaces.
- **SKIRT_HEIGHT:** R; - DEFAULT 0& m or ft; Boom skirt height
- **TUG_VEL_U& R:** - DEFAULT 0; m/s or ft/s; Tug velocity component in x direction.
- **TUG_VEL_V& R:** - DEFAULT 0; m/s or ft/s; Tug velocity component in x direction.
- **X_B(I) Y_B(I):** R; -; m or ft; Vertex coordinates defining each boom polyline.

11.16 Oil on Land Model File: .OILP

This section applies to the OilFlow2D model. The file provides the parameters necessary to model flow of viscous fluids including oil over complex terrain using the OilFlow2D model and considers the heat transfer between the flowing oil with the surroundings and the dependence of oil properties with temperature.
Line 1: OILP file version number.
OL_FILEVERSION

Line 2: Flow resistance relation.

OL_FRR

Line 3: Yield stress.

OL_YS

Line 4: Fluid viscosity.

OL_VIS

Line 5: Internal friction angle. Not used in this release.

OL_THETA

Line 6: Oil density.

OL_DENS

Line 7: Temperature time series file.

OL_TEMPTSERIES

Line 8: Temperature - Viscosity - Density table file.

OL_TEMPVISCDENS

Line 9: Evaporation option.

OL_EVAPOPTION

Line 10: Evaporation calculation coefficients.

OL_C1 OL_C2

Line 11: Wind velocity.

OL_WINDVEL

Line 12: Heat transfer option switch.

OL_HTOPTION

Line 13: Density formulation.

OL_DENSFOR

Line 14: Density formulation parameters.

RHO0 T0 Lambda OL_DENSPAR4 … **OL_DENSPAR10**

Line 15: Density formulation file.

OL_DENSFILENAME

Line 16: Viscosity formulation.

OL_VISCFOR

Line 17: Viscosity formulation parameters.

Av Bv OL_VISCPAR3 … **OL_VISCPAR10**

Line 18:: Viscosity formulation file.

OL_VISCFILENAME

Line 19: Yield stress formulation.

OL_YSFOR

Line 20: Yield stress formulation parameters.

Ays Bys Cys OL_YS … **OL_YS**

Line 21: Yield stress formulation file.

OL_YSFILENAME

Line 22: Heat transfer parameters.

INI_T_OPTION INI_T Hr T_AIR Q_RAD Cp OL_MAXTEMP OL_HT8 … **OL_HT10**

Line 23: Environmental Parameters file.

OL_ENVFILENAME

Line 24: Environmental Parameter Options.
OL_ENVOPTIONS
Line 25: Switch for wet boundary cells.
OL_WETBOUNDARY
Line 26: Switch for water lower layer.
OL_WATERLAYER
Line 27: Water density and friction coefficient.
OL_WATERRHO OL_CD
Line 28: Path to the water hydrodynamic .textout files.
OL_WATERPATH
Line 29: Total number of .textout files.
OL_NFTEXOUR
Line 30: Time interval of textout files.
OL_TEXTOUTDT
Line 31: Oil retention switch and retention depth.
OL_RETOPT OL_HRETENTION

11.16.1 Example of a .OILP file

```
202405
3
1.
0.00899
1
2200.
Temptseries.TXT
Tempviscdensetable.TXT
1
0.018 10
2.25
1
1
0.1 0.2 0.3 0.4 0. 0. 0. 0. 0. 0.
densityfile.txt
1
0.11 0.12 0.13 0.14 0. 0. 0. 0. 0. 0.
Viscosityfile.txt
1
-0.0196 1.3575 -20.454 0. 0. 0. 0. 0. 0. 0.
YSfile.txt
0 30.0 0.3 25.0 300. 1900. 180. 0. 0. 0.
EnvironmentalParameters.txt
2
```

```
0
0
1000.  10.
Path
10
2.0
0  0.0
```

- **Av, Bv:** R; $-$; -; Regression constants for the Andrade viscosity formula $\mu = e^{[Av+(Bv/T)]}$ applied when OL_VISCFOR = 1.
- **Ays, Bys, Cys:** R; $-$; -; Regression constants in Yield Stress formula $Ys = 10^{(AysT^2+BysT-Cys)}$ when OL_YSFOR = 1.
- **Lambda (Λ_ρ):** R; > 0; -; Temperature gradient in the Arafin et al. formula $\rho(T) = \rho_0+\Lambda_\rho(T-T_0)$ when OL_DENSFOR = 1.
- **Cp:** R; > 0; J/kg°C or BTU/lb°F; Heat Capacity or Specific Heat. If Q_RAD is set to -1, the model will use the Cp read from the Environmental parameters file indicated in line 22.
- **Hr:** R; $[0, 1]$; -; Relative humidity given as a fraction of 1.
- **INI_T:** R; > 0; °C or °F; Constant initial temperature (if INI_T_OPTION=1).
- **INI_T_OPTION:** I; $0, 1$; -; Switch to choose between constant temperature distribution (0) or spatially distributed temperature (1) read from .tinitial file.
- **OL_C1, OL_C2:** R; > 0; -; Evaporation calculation coefficients used to calculate viscosity and density according to equations and. Use OL_C1 = 0.018 and OL_C2 = 10.
- **OL_DENS:** R; > 0; kg/m^3 or lb/ft^3; Fluid density that will be used in the basic model if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
- **OL_DENSFILENAME:** S; -; -; Density formulation file. Will be used in future updates for the interpolation option, and contains a table of temperature vs density.
- **OL_DENSFOR:** I; 1,2,3; -; Density formulation used in the Heat Transfer model.
- Arafin et al. 2006. formulation: $\rho(T) = \rho_0 + \Lambda_p(T - T_0)$, with ρ_0 in kg/m^3 or lb/ft^3, T$_0$ in °C or °F, and Λ_p in kg/m^3°C or lb/ft^3°F.
- Arafin density formulation for density. The model does not incorporate density variable in the equations.
- Density is set by interpolation from the given table in OL_DENSFILENAME. In this case the model does not incorporate the variable density in the equations, since it is given by the table values for a given temperature.
- **OL_DENSPAR4..10:** R; -; -; Density formulation parameters that will be added in future updates for alternative formulas.

- **OL_ENVFILENAME:** S; -; -; Environmental parameters data file. See the detailed description below. The model will use the data in this file, always that there OL_ENVFILENAME is present in the project folder.
- **OL_EVAPOPTION:** R; 0, 1, 2; -; Evaporation calculation option:
- No evaporation is calculated,
- Evaporation rate is calculated and properties estimated according to given density viscosity table,
- Evaporation rate is calculated and properties calculated by the model using formula.
- **OL_FRR:** I; 3; -; Flow resistance relation. (see Table)
- **OL_HTOPTION:** I; 0, 1; -; Basic model vs Heat transfer model switch:
- Basic model. No heat transfer is calculated,
- Heat transfer model.
- **OL_HT8..10:** R; -; -; Heat transfer parameters that will be added in future updates.
- **OL_MAXTEMP:** R; > 0; °C or °F; Maximum oil temperature allowed.
- **OL_RETOPT:** I; 0,1; -; Surface retention switch. Oil retention by the soil will be disabled when this switch is set to 0, and if set to 1, the oil will be retained by the terrain up to the depth indicated by OL_RETOPT.
- **OL_HRETENTION:** R; $>= 0$; -; Surface retention depth applicable to all cells. It is the $h_{\text{retention}}$ variable in equation. The model can consider spatially variable oil retention if the .deteninitial file exits in the scenario folder. Note that OL_HRETENTION should be in the order of millimeters since the model assumes that velocity is 0 for all oil depth below its value. See the .deteninitial file format below.
- **OL_TEMPTSERIES:** S; ≤ 26; -; Temperature time series file used in the basic model.
- **OL_TEMPVISCDENS:** S; ≤ 26; -; Temperature-viscosity-density table file. The basic model will use the temperature for a given time to interpolate the viscosity and density.
- **OL_THETA:** R; > 0; Degrees; Not used in this release.
- **OL_VIS:** I; > 0; Pa-s or lb-s/in^2; Fluid viscosity that the basic model will use if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
- **OL_VISCFILENAME:** S; -; -; Viscosity formulation file. Will be used in future updates for the interpolation option, and contains a table of temperature vs viscosity.
- **OL_VISCFOR:** I; 1,2; -; Viscosity formulation used in the Heat Transfer model.
- Andrade (1934) formulation $\mu = e^{[Av+(Bv/T)]}$, with Av in Pa-s or lb-s/in^2, Bv in Pa-s/°C or lb-s/in^2°F, and T is the oil temperature in °K

- Determined by interpolation from table provided in file OL_VISCFILENAME.
- **OL_VISCPAR3..10:** R; -; -; Viscosity formulation parameters that will be added in future updates for alternative formulas.
- **OL_WINDVEL:** R; ≥ 0; m/s or ft/s; Wind velocity assumed constant in time and space used to compute evaporation rate in basic model that does not considers heat transfer. Note that according to formula , if the wind velocity is zero, the evaporation rate is zero as well.
- **OL_YS:** R; > 0; Pa or lb/in^2; Yield stress used in the basic model.
- **OL_YSFILENAME:** S; -; -; Yield stress formulation file. Contains a table of temperature vs Yield stress.
- **OL_YSFOR:** I; 1,2; -; Yield stress in Pa or lb/in^2 formulation.
- Formulation 1: $Ys = 10^{AysT^2+BysT-Cys}$, with Ays in Pa/°C^2 or lb/in^2°F^2, Bys in Pa/°C or lb/in^2°F, and Cys in Pa or lb/in^2,
- Determined by interpolation from table provided in file OL_YSFILENAME.
- **OL_ENVOPTIONS:** I; 0,1,2; -; Set the option for using Environmental Parameters.
- The model will not compute heat transfer,
- Use constant environmental parameters as entered on line 21 of the .OILP file,
- Use the parameters defined on the OL_ENVFILENAME file.
- **OL_YSPAR4..10:** R; -; -; Yield stress formulation parameters that will be added in future updates for alternative formulas.
- **Q_RAD:** R; > 0; W/m^2 or BTU/ft^2 s; Solar radiation. Constant Q_RAD value used if OL_ENVOPTIONS os set to 1 (Constant).
- **RHO0 (ρ_0):** R; > 0; kg/m^3 or lb/ft^3; Reference density in the Arafin et al. formula $\rho(T) = \rho_0 + \Lambda_\rho(T - T_0)$ when OL_DENSFOR = 1.
- **T0 (T_0):** R; $-$; °C or °F; Reference Temperature in the Arafin et al. formula $\rho(T) = \rho_0 + \Lambda_\rho \cdot (T - T_0)$ when OL_DENSFOR = 1.
- **T_AIR:** R; $-$; °C or °F; Air temperature. If Q_RAD is set to -1, the model will use the T_AIR read from the Environmental parameters file indicated in line 22.
- **OL_WETBOUNDARY:** I; $0, 1$; -; Switch for wet boundary cells.
- No wet cell options.
- The model will write a file named wetBcells.out containing the list boundary cell numbers that are wet during the run.
- **OL_WATERLAYER:** I; $0, 1$; -; Switch for water lower layer.
- The model will ignore oil flow over water.

- The model will consider oil flow over underlying water. This requires a previous water hydrodynamic run that determines the water flow (see OL_WATERPATH below)
- **OL_WATERRHO:** R; > 0; g/m^3 or lb/ft^3; Water density.
- **OL_CD:** R; $0.001 - 0.005$; -; Non-dimensional drag coefficient between oil and water.
- **OL_WATERPATH:** S; $-$; -; Path to hydrodynamics cell_time_*.textout files. These files contain the results of an existing water hydrodynamic run.
- **OL_NFTEXOUR:** I; $-$; -; Number hydrodynamic cell_time_*.textout files.
- **OL_TEXTOUTD:** R; $-$; h; Time interval between hydrodynamic cell_time_.textout files.
- **2:** Full Bingham; $\tau_b = \tau_0$ with $f_1(\tau_b, \tau_0) = 0$; Viscosity, yield stress, density.
- **3:** Simplified Bingham; $\tau_b = 1.5\tau_y + 3\tau_\mu$; Viscosity, yield stress, density.
- **7:** Quadratic; $\tau_b = \tau_t + \tau_y + \frac{\kappa}{8}\tau_\mu$; Manning's n, viscosity, yield stress, density.

11.16.1.1 Temperature Time Series file

The temperature time series files is only used in the basic model and contains the ambient temperature for each time along the simulation. The first line has the number of lines in the time series, then a list of time in hours vs temperature in degrees Celsius or ^F depending on the unit system used.
The following is an example of a 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.

11.16.2 Temperature-Viscosity-Density Table file

This file is only used in the basic model and includes the viscosity and density for different temperatures. The basic model will interpolate from the table values to assign to the oil the viscosity and density for the current temperature.
The following is an example of a Temperature-Viscosity-Density 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 1.38×10^{-6} lb-s/in^2 and the density 50 lb/ft^3.

11.16.3 Temperature vs Density Time Series file

The temperature time series file is only used in the heat transfer model and contains a table of temperature and corresponding oil density for each time along the simulation. The first line has the number of lines in the time series, then a list of temperature and density. This file is optional and will only be necessary when selecting the interpolation for density option.
The following is an example of a temperature vs density time series file:

```
8
10  1010
15  1004
20  998
22  995
25  980
30  975
35  970
40  960
```

11.16.4 Temperature vs Viscosity Time Series file

The temperature time series file is only used in the heat transfer model and contains a table of temperature and corresponding oil dynamic viscosity for each time along the simulation. The first line has the number of lines in the time series, then a list of temperature and dynamic viscosity. This file is optional and will only be necessary when selecting the interpolation for viscosity option.
The following is an example of a temperature vs viscosity time series file:

```
8
10  0.10
15  0.09
20  0.07
22  0.04
25  0.02
30  0.009
35  0.0084
40  0.007
```

11.16.5 Temperature vs Yield Stress Time Series file

The temperature time series file is only used in the heat transfer model and contains a table of temperature and corresponding oil yield stress for each time along the simulation. The first line has the number of lines in the time series, then a list of temperature and yield stress. This file is optional and will only be necessary when selecting the interpolation for yield stress option.
The following is an example of a environmental parameter file:

```
9
0.00  2000.0
```

```
35.00  2000.0
37.50  1000.0
40.00  200.0
42.50  80.0
45.00  20.0
47.50  1.0
50.00  0.020
100.00  0.020
```

11.16.6 Environmental Parameters Time Series file

This file contains environmental parameters that affect the oil heat transfer. The first line contains the number of lines with data. Then each subsequent line includes the time in hours, wind speed (m/s or ft/s), relative humidity (0-1), air temperature (°C or°F), and solar radiation (W/m^2 or BTU/ft^2). The model will use the data in this file if the solar radiation entered in line 21 is equal to -1.
The following is an example of a environmental parameter file:

```
12
1    2.20    0.90    21.90    0.0
2    1.71    0.90    21.71    0.0
3    1.79    0.93    21.33    0.0
4    2.23    0.94    21.13    0.0
5    2.58    0.97    20.77    0.0
6    4.04    0.99    20.13    0.0
7    3.41    1.00    19.54    1.8
8    1.93    0.99    19.62    13.5
9    1.18    0.96    20.08    74.8
10   1.65    0.87    21.64    166.0
11   1.49    0.81    22.97    246.7
12   1.34    0.71    33.64    368.8
```

11.16.7 Oil Retention Depth Data File: .DETENINITIAL

The purpose of this file is to indicate the oil retention depth (HRet) for each cell so that the model considers spatially varied retention depth.

> **Note**
>
> NOTE: It is important to note that the oil retention depth represents the minimal depth for oil flow. Consequently, the velocity of cells with an oil depth smaller than HRet will be zero. For this reason, the values for HRet should be on the order of millimeters.

The model can create this file based on polygons entered in the OilFlow2D QGIS plugin based on the data in the *Oil Retention* layer. The user can enter polygons and assign a corresponding retention depth file name to each polygon.

The files will contain the list of retention depth, one per line, for all cells.
HRet(1)
HRet(2)
…
HRet(NELEM)

11.17 Oil on Water File: .OILW

This file only applies to the OilFlow2D model. This file provides the parameters necessary to model oil spill on water using the OilFlow2D model.
Line 1: **OILW_VersionNumber**
Line 2: **NSpillSites**
Line 3: **[ControData]**
Line 4: **Evaporation Switch**
Line 5: **Emulsification Switch**
Line 6: **Shoreline Switch**
Line 7: **Dispersion Switch**
Line 8: **Dissolution Switch**
Line 9: **Sedimentation Switch**
Line 10: **Simulation Time**
Line 11: **Output Interval**
Line 12: **Time Step**
Line 13: **Wind Velocity File**
Line 14: **Ambient Temperature File**
Line 15: **Velocity Field Path**
Line 16: **Velocity Field .OUTFILES**
Line 17: **Grid Cell Factor (GCF)**
Line 18: **Boom Component Switch**
Line 19: **Wind Velocity Coefficient**
For SpillSite 1 to NSpillSites
Spill Site ID
[Trajectory]
Water Density
X Y Z
Number of Parcels
Oil Density
Oil Viscosity
Initial Spill Time
Disp_L Disp_T Disp_V
Spill Accumulated Volume File
Moving Spill Switch
Spill Release File
0
0

0
0
0
0
0
0
[Evaporation]
Evaporation Method // 1: Fingas; 2: Stiver-MacKay
Oil ID
PD Fraction by weight distilled at 180oC.
Kevp // Mass transfer coefficient //Stiver-MacKay
A //Stiver-MacKay constant A
B //Stiver-MacKay constant B
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Emulsification]
Kemul
Yf
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Dissolution]
Kdiss
S // Water solubility
So // Solubility of fresh oil
Alfa // Decay constant
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Dispersion]
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use

0 // For future use
0 // For future use
0 // For future use
[Sedimentation]
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Shoreline]
BottomTrapping // Bottom Trapping option
ShoreTrapping // Shore Trapping option
Vegetation trapping switch
Dispersant Application Switch
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
End SpillSite

- **Version Number:** I; -; -; Integer number indicating the model version.
- **NSpillSites:** I; ≥ 1; -; Number of spill sites.
- ****[CONTROLDATA]:** &; & Control data group with common to all spill sites.
- **Boom switch:** I; 0,1; -; 0: Booms will not be considered. 1: Booms activated.
- **Evaporation switch:** I; 0,1; -; 0: No evaporation will be calculated. 1: Evaporation will be calculated.
- **Emulsification switch:** I; 0,1; -; 0: No emulsification will be calculated. 1: Emulsification will be calculated.
- **Shoreline switch:** I; 0,1; -; 0: Oil interaction with shores will not be considered. 1: Oil interaction with shores will be considered.
- **Dispersion switch:** I; 0,1; -; 0: No oil dispersion will be calculated. 1: Oil dispersion will be calculated.
- **Dissolution switch:** I; 0,1; -; 0: No dissolution will be considered. 1: Dissolution will be considered.

- **Sedimentation switch:** I; 0,1; -; 0: No oil sedimentation will be calculated. 1: Oil sedimentation will be calculated.
- **Simulation time:** R; > 0; h.; Total simulation time in hours.
- **Output interval:** R; > 0; h.; Report output interval in hours.
- **Time step:** R; > 0; s.; Time step that will be used in the numerical solution of the particle-tracking algorithm.
- **Grid Cell Factor (GCF):** R; > 0; -; This factor is used to construct a virtual grid over the mesh that is used to optimize the particle location algorithm. The grid size is equal to GCF * Minimum Cell Size. The program will determine all the mesh cells that lie within each grid element. In this way, the search algorithm will first locate the particle within the grid element and then, to determine in which cell the particle is, it will search only among the cells that are in that grid element, hence reducing the search time. Larger the GCFs means that mode cells will be in each grid element, therefore making the search more inefficient. The default value is 20, which will construct a grid elements 20 times larger than the minimum cell size.
- **Wind velocity file:** S; -; -; Wind velocity file name. See the wind velocity file format section.
- **Temperature file:** S; -; -; Ambient temperature file name. See the ambient temperature file format section.
- **Velocity field path:** S; -; -; Path where the results of the velocity field files are located.
- **Spill site ID:** S; ≤ 26; -; Name of the spill site.
- ****[TRAJECTORY]&:** &; Trajectory data group for each spill site. ****
- **Water density:** R; ≥ 0;
- **X Y Z:** R; -; ft. - m.; Spill site coordinates.
- **Nparcels:** I; > 0; -; Number of parcels or particles representing the spill.
- **Oil density:** R; > 0; API; Initial oil density of the oil at the time of the spill.
- **Oil viscosity:** R; > 0; cP - lb-s/ft^2; Initial oil viscosity at the time of the spill.
- **Initial Spill Time:** R; > 0; h.; Initial oil viscosity at the time of the spill.
- **Disp_L:** R; > 0; ft^2/s - m^2/s; Longitudinal dispersion coefficient.
- **Disp_T:** R; > 0; ft^2/s - m^2/s; Transversal dispersion coefficient.
- **Disp_V:** R; > 0; ft^2/s - m^2/s; Vertical dispersion coefficient.
- **Spill Accumulated Volume File:** S &-; c-; Spill volume file. See format in section .
- **SpillReleaseSwitch:** I; 0,1; -; Spill path switch. 0 indicates fixed point spill, 1 indicates that the spill path will be given by the SpillReleaseFile volume file.
- **SpillReleaseFile:** S &-; c-; Spill release path file. See format in section .

- ****[EVAPORATION]&:** &; Evaporation data group for each spill site. **
- **Evaporation Method:** I; 1,2; -; 1: Fingas; 2: Stiver-MacKay.
- **OilID:** I; -; -; Oil ID for Fingas method selected from drop-down list.
- **PD:** R; [0,1]; -; Fraction by weight distilled at 180^{o}C.
- **Kevap:** R; ≥ 0; -; Mass transfer coefficient.
- **A:** R; ≥ 0; -; Stiver-MacKay constant A.
- **B:** R; ≥ 0; -; Stiver-MacKay constant B.
- ****[EMULSIFICATION]&:** &; Emulsification data group for each spill site. **
- **Kemul:** R; ≥ 0; -
- **Yf:** R; ≥ 0; -
- ****[DISSOLUTION]&:** &; Dissolution data group for each spill site.
- **Kdiss:** R; ≥ 0;
- **S:** R; ≥ 0; & Water solubility.
- **So:** R; ≥ 0; & Oil solubility.
- **Alpha:** R; ≥ 0; & Decay constant.
- ****[SHORELINE]&:** &; Shore and bottom interaction data group for each spill site.
- **BottomTrapping:** I; 0,1; -; 0: Particles will not be trapped when they hit the bottom (default). 1: The oil particles will be permanently attached to the bottom as soon as they touch it.
- **Vegetation trapping switch:** I; 0,1; -; 0: Particles will not be trapped by vegetation. 1: The oil particles will be attached to the cells belonging to vegetation polygons.
- **Dispersant Application Switch:** I; 0,1; -; 0: No dispersants will be simulated, 1 indicates that dispersants will be simulated. ****

11.17.1 Example of a .OILW file

```
201910 // version
1 // NSpillSite
[ControlData]
0 // Evaporation Switch
0 // Emulsification Switch
0 // Shoreline Switch
0 // Dispersion Switch
0 // Dissolution Switch
0 // Sedimentation Switch
5 // Simulation Time
```

```
0.1 // Output Interval
10.0 // Time Step
Wind.DAT
Temperature.DAT
C:/Users/Hydronia Dell/Documents/Magdalena
base.OUTFILES // Velocity field output files
100 // Grid Cell Factor
0 // Boom component deactivated
0 // Velocity coefficient
OilSpill_1
[Trajectory]
1000.00 // Water Density
925018.358  1707701.919  0.000  // X Y Z
1000 // Nparcels
40.00 // Oil Density
15.000 // Oil Viscosity
1.00 // Initial SpillTime
1.000  1.000  0.001 // Disp_L  Disp_T  Disp_V
OilSpill_1.txt
0 // Moving Spill Switch
0 // Spill Release File
0
0
0
0
0
0
0
0
[Evaporation]
1 EvaporationMethod // 1: Fingas; 2: Stiver-MacKay
1 // Oils
40.00 PD // percentage (by weight) distilled at 180oC.
0.0025 // Kevp (mass transfer coefficient) //Stiver-MacKay
6.30 // A Stiver-MacKay
10.0 // B Stiver-MacKay
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Emulsification]
0.000002 // Kemul
0.7 // Yf
```

```
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Dissolution]
1.00 // Kdiss
0.00 // S Water solubility
0.0184 // So Solubility of fresh oil
0.423  // Alfa  Decay constant
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Dispersion]
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Sedimentation]
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
0 // For future use
[Shoreline]
0 // No bottom Trapping
1 // Do shore trapping
0 // Vegetation trapping switch
0 // Dispersant Application Switch
0 // For future use
0 // For future use
```

```
0 // For future use
0 // For future use
0 // For future use
0 // For future use
```

11.17.2 Accumulated Volume Input File

This file specifies the volumes released in the spill for each time. Regardless of the project unit system the volumes should be provided in barrels (bbl). The format is as follows
Line 1: Number of points in the accumulated spill volume time series.
NPOINTS
NPOINTS lines containing:
Time and Accumulated Volume.
TIME ACCVOL

11.17.3 Example of a Accumulated Volume File

```
3
0. 0.
2. 3000
10. 20000
```

- **NPOINTS:** I; $\geq$ 1; -; Number of data points in time series.
- **TIME:** R; $\geq$ 0; h; Time.
- **ACCVOL:** R; -; bbl; Accumulated spill volume until this time. Note that it is required that the accumulated volume given for one time needs to be always greater or equal to the accumulated volume given for the previous time.

11.17.4 Spill Release File

This file specifies the path of the spill release. The spill will be released along the path defined in this file. The format is as follows
Line 1: Number of points in the spill release time series.
NPOINTS
NPOINTS lines containing:
Time and point coordinate
TIME Xp Yp

11.17.5 Example of Spill Release File

```
5
0.0   923730.325     1710175.128
0.5   923408.268     1710245.895
1.0   923192.219     1710246.443
```

```
1.5    923022.774      1710195.172
2.0    922888.358      1710131.769
```

- **NPOINTS:** I; $\geq$ 1; -; Number of data points in time series.
- **TIME:** R; $\geq$ 0; h; Time.
- **Xp:** R; -; ft - m; X coordinate of this spill point.
- **Yp:** R; -; ft - m; Y coordinate of this spill point.

11.18 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 data is included in the data file that has the following format:
Line 1: Maximum number of pollutants (solutes)
NP_MAX
Line 2: Number of pollutants used in the run
NP_USED
Line 3: List of solutes used (all in one line)
I(1) I(2) … I(NP_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_NP_MAX
Line 6 to NP_MAX+6: First order reaction rate coefficient matrix K(I,J)
K(1,1) K(1,2) … K(1,NP_MAX)
K(2,1) K(2,2) … K(2,NP_MAX)
…
K(NP_MAX,1) K(NP_MAX,2) … K(NP_MAX,NP_MAX)

11.18.1 Example of a .SOLUTES file

```
5
4
1  2  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
```

& There are five pollutants
- **4:** Four pollutants will be used in this run 1 2 4 5& The active pollutants will be 1, 2, 4 and 5
- **0.10.02:** Longitudinal dispersion coefficient is 0.1 and transversal = 0.02. - **NITRATE:** 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 - **0.20.00.00.00.0:** First order reaction constant for pollutant No. 1 is 0.2 - **0.00.10.00.00.0:** First order reaction constant for pollutant No. 2 is 0.1 - **0.00.01.10.00.0:** First order reaction constant for pollutant No. 3 is 1.1 - **0.00.00.00.20.0:** First order reaction constant for pollutant No. 4 is 0.2 - **0.00.00.00.00.5:** First order reaction constant for pollutant No. 5 is 0.2

- **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.
- **NP_MAX:** I; > 0; -; Maximum number of pollutants.
- **NP_USED:** I; > 0; -; Number of pollutants used in the present run.

11.18.2 Pollutant Transport Module Initial Concentration Data File: .CINITIAL

The initial concentration data file is created when the dataset with the initial concentrations is exported from the OilFlow2D QGIS plugin based on the data entered in the *Initial Concentrations* layer. The user can enter polygons and assign a corresponding initial concentration file name to each polygon. The format of these files is as follows:
Line 1: Space separated initial concentrations for each pollutant.
C(1) C(2)…C(NP_MAX)
Where C(i), is the initial concentration for pollutant i. With this information the model will determine the initial concentrations for each cell based on the polygons entered. All the cells that are not contained within the polygons on the *InitialConcentrations* layer, will be assigned the zero initial concentrations. Therefore the file will have NP_MAX columns and NELEM lines.

> **Note**
> The pollutant concentration units are arbitrary. The user can use volume concentration Cv (fraction of 1), mg/l, ppt, ppm, or any other suitable units, provided that the inflow boundary conditions are consistent.

11.19 Urban Drainage Module Data File: .LSWMM

This data file allows to dynamically run the OilFlow2D model with the EPA-SWMM model to simulate surface/storm-drain network interaction. The user can define multiple exchange locations corresponding the manholes defined in the EPA-SWMM project file , and assign different diameters and discharge coefficients to each exchange node.
Line 1: File Version Format.
202507
Line 2: Number of manholes or exchange nodes.
NMAN
NMAN group of lines including manhole ID, manhole coordinate, diameter and discharge coefficient as follows:
MANHOLE_ID(I)
X(I) Y(I)
Cd(I)
DIAM(I)
NCELLS Cell(1) Cell(2) … Cell(NCELLS)
Q_OPT
AUX_PARAM
…

11.19.1 Example of a .LSWMM file

```
202507
2
N1
264930.824  664804.843
0.61
0.800
4  235  7665  7869  8798
0
0
N2
264896.000  664747.000
0.61
1.300
5  2236  3634  6832  3745  2561
1
HvsQTable.txt
```

This file corresponds to the format 202507 and indicates that there are 2 manholes. The first one is N1, has coordinates 264930.824 664804.843, Cd = 0.61 and diameter = 0.8. Three are 4 cells contributing to manhole N1 (235 7665 7869 8798), and 5 cells contributing to manhole N2 (2236

3634 6832 3745 2561). For node N2 a rating table will be used to calculate the inflow discharge given by in the HvsQTable.txt file.

- **AUX_PARAM:** S, R; -; -
- AUX_PARAM line is ignored.
- AUX_PARAM line contains the rating table (depth vs discharge) file name. See format below.
- AUX_PARAM line contains a text for future use.
- AUX_PARAM line contains two parameters, cells elevation H_ROOF and coeffient KT_roof.
- **Q_OPT:** I; $0, 1, 2, 3$; -
- Default calculation based on circular orifice formula as a function of the average depth among all wet cells in the list.
- Exchange discharge is set based on the rating table in (depth vs Q). The file name should be given in the AUX_PARAM line.
- For future use.
- Calculates exchange discharge based on the rooftop tool described in section.
- **Cd(I):** R; > 0; -; Discharge coefficient for exchange node I.
- **CELL(I):** R; -; -; Cell numbers that contribute flow to this manhole.
- **DIAM(I):** R; -; m of ft; Diameter of exchange node I. See Comment below.
- **MANHOLE_ID(I):** S; < 26; -; Name of exchange node I. Should have less than 26 characters and must not contain blank spaces.
- **NCELLS:** I; ≥ 0; -; Number of cells associated with this manhole. If NCELLS = 0, the manhole cell is the only exchange cell with the SWMM model.
- **NMAN:** I; > 0; -; Number of manholes or exchange nodes.
- **X(I) Y(I):** R; -; m of ft; Coordinates of exchange node I.

11.19.2 Depth-Discharge Data Files

This format applies to the depth vs. discharge table used for CALC_OPT = 1.
Line 1: Number points in data series.
NDATA
NDATA lines containing depth and discharge.
h(I) Q(I)
Where h(I) is water depth and Q(I) is the corresponding discharge.

11.19.2.1 Comments for the File

Water discharge from the mesh into the SWMM inflow node is computed with one of two options:

1. If CALC_OPT = 0, Q is calculated assuming a circular intake of diameter DIAM and discharge coefficient Cd.

2. If CALC_OPT = 1, Q is calculated interpolating from the h vs Q rating table given in AUX_PARAM file.

   item If CALC_OPT = 3, Q is calculated based on the rooftop tool described in section.

11.20 Output control data

11.20.1 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 name and format of the observation point files are described in this section. Based on the number of points entered in the *Observation points* layer, the file will 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)
...

11.20.1.1 Example of a .OBS file

```
3
PointA
798798.380  309627.950
PointB
799146.926  309430.876
PointC
799721.8608  309041.615
```

This file has three points. The first point is named PointA and has coordinates: X=798798.380 Y=309627.950.

- **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; Point coordinates.

11.20.2 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

11.20.2.1 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
```

- **COLORSCHEME:** I; 1-6; -; Variable to select plot option. See comment 1.
- **IAXES:** I; 0,1; -; Switch to control weather to plot axes.

1. Do not plot X and Y axes.
2. Plot X and Y axes.

- **IDXF:** I; 0,1; -; Switch to control velocity field output in DXF CAD format.

1. Do not output DXF mesh and velocity field.
2. Create mesh and velocity field DXF files for each output time.

- **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.

1. Plot velocity field using vectors in black.
2. Plot velocity field using vectors in black over depths.
3. Plot velocity field over bed elevations.
4. Plot water elevations.
5. Plot velocity field using vectors in color.
6. Plot depths.
7. Plot bed elevations.
8. Plot velocity field over water elevations.
9. Plot bed elevation changes.
10. Plot suspended sediment or pollutant concentrations.

- **IGRAPHFILES:** I; 0,1; -; Variable to control whether to output graphic files.

1. Do not output graphic files.
2. Output graphic files.

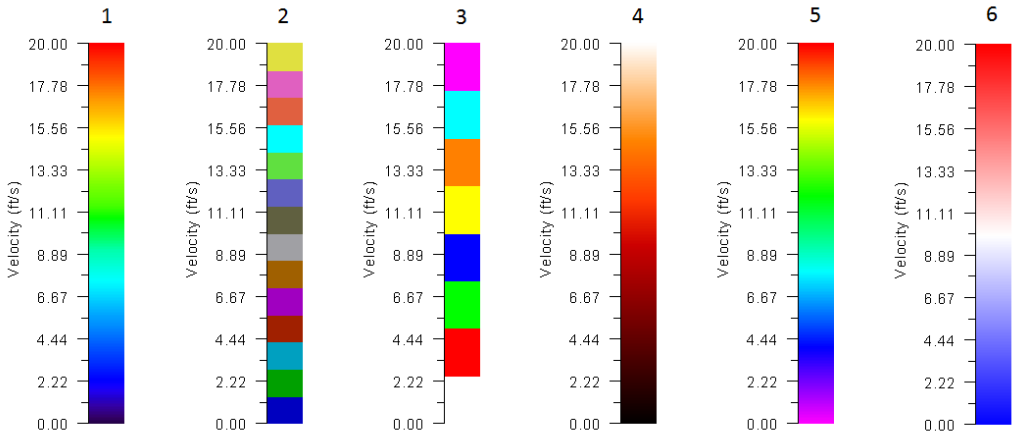
- **IMAGEFILE:** S; -; -; Name of aerial image file including path and extension. Supported formats include , and. Other graphic file formats will be included in forthcoming releases.
- **IMAGEWF:** S; -; -; Name of aerial image world file including path and extension. World file format should follow ESRI specifications. See comment 2.
- **IVSF:** I; 0,1; -; Switch to control velocity field output in a file that allows creating shapefiles.

1. Do not output SF velocity field.
2. 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.
- **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.

11.20.2.2 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:



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.

   & ,
   & , ,
   & , ,

- **,:** ,

11.20.3 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)
...

11.20.3.1 Example of a .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.

- **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.

11.20.4 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)

11.20.4.1 Example of a .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 file indicates there are 3 cross sections. The first one has ID = CrossSectionA and will be divided in 40 segments.

- **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.

11.20.4.2 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.

11.21 Elevation data

11.21.1 X Y Z data with header

These files contain scattered data in the format suitable to import it in a text editor or spreadsheet program. For example the *BedElevations* data layer. It usually has 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 per point (header)
NUMBER_OF_DATA_POINTS
NUMBER_OF_DATA_POINTS lines with X, Y and parameters data.
X(POINT) Y(POINT) P1(POINT) P2(POINT) … PN(POINT)

11.21.1.1 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 file, there are 11086 elevation data points, one parameter per point (the elevation for each point).

- **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.

11.21.1.2 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. OilFlow2D does not support geographical coordinates in Latitude/Longitude format.
2. Elevation values should be given in the same units as the corresponding coordinates.

11.22 Boundary conditions data files

11.22.1 One Variable Boundary Condition Files

This format applies to the following data files:

- Time vs. Water Surface Elevation (BCTYPE = 1, 17)
- Time vs. Discharge (BCTYPE = 6)

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, or Q, depending on the boundary condition **BCTYPE**.

11.22.1.1 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.
```

- **NDATA:** I; > 0; -; Number of points in data series.
- **TIME:** R; > 0; h; Time in hours. The time interval is arbitrary.
- **VARIABLE:** R; -; -; Represents Water Surface Elevation, or Discharge, depending on the boundary condition.

11.22.2 Two Variables Boundary Condition Files

This format applies to the following data files:

- Time vs. Discharge Q and Water Surface Elevation (BCTYPE = 5)

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:

- **5:** Q; WSE

11.22.2.1 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.
```

- **NDATA:** I; > 0; -; Number of points in data series.
- **TIME:** R; > 0; h; 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.

11.22.3 Multiple-Variable Boundary Condition Files

This format applies to the following data file:
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:

- **Q:** Qs

11.22.3.1 Example of the Multiple-Variable Boundary Condition File

```
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.
```

- **NDATA:** I; > 0; -; Number of points in data series.
- **TIME:** R; > 0; h; Time in hours. The time interval is arbitrary.
- **VARIABLE1:** R; -; -; Represents Water Discharge.
- **VARIABLE2..N:** R; -; -; Represents Sediment Discharge for the given fraction.

11.22.4 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.

11.22.4.1 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
```

- **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.

11.22.5 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).

11.22.5.1 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
```

- **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.

12

Output File Reference

OilFlow2D generates results in many output ASCII text files. These files can be easily accessed with any text editor and they can be imported into QGIS o other GIS software for visualization and analysis. OilFlow2D always creates output ASCII files in both English and metric units depending on the units provided in the data files.

12.1 Output File Overview

The following tables summarizes the output files generated by OilFlow2D:

- **List of output times:** Reports output times for result files.
- **Run control parameters, components used, etc.:** 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 file is in metric units and file in English units.
- **Native model output files:** Reports the model results at cells for each reporting interval.
- **Run control parameters, components used, etc.:** 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 file is in metric units and file in English units.
- **Triangular-cell mesh information:** These files provide comprehensive information about the triangular-cell mesh. is in metric units and file in English units.
- **Run progress results:** This file report for each output interval the computer time, average time step, inflow and outflow water at open boundaries and volume and mass conservation errors.

- **Maximum values tabular output:** For each output interval maximum nodal velocity modules, depths, and are written to file is in metric units and in English units.
- **Time series at observation points:** These files report time series of results for the cell where the point is located. The results include time series of velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, and Froude number. File name format is as follows: for metric units and for English units, where is the observation point name.
- **Mass balance:** Report total inflow and outflow discharges and volumes for each output interval.
- **Hot start:** Generated in the OilFlow2D model to restart a simulation from previously computed results. The file contains the restart time in seconds, the source-output time code, and the restart output index.
- **Native model output files:** Report the model results at cells for each reporting interval.
- **Cross section output:** For all output intervals, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number, water discharge. is in metric and in English units.
- **Cross section hydrographs:** Report a hydrograph table for each cross section. is the water hydrograph and the sediment flux hydrograph for each cross section. See Comment 1.
- **Profile output:** 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 is in metric and in English units.
- **Culverts:** Output discharge at every culvert for each report interval. File name format is as follows: where culvertID is the user provided name.
- **Internal Rating Tables:** Output discharge at every IRT for each report interval.File name format is as follows: , where IrtID is the text provided by the user to identify the Internal Rating Table.
- **Weirs:** Report results for weirs. is in metric units and WEIRE in English units.
- **General results:** When the Create Graphic Output Files check box is selected in the Graphic Output Panel, OilFlow2D model will output files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of OilFlow2D results. ParaView (https://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 o programming using ParaView's batch processing capabilities.
- **Spatial distribution of results for each report interval:** For each output interval cell velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number, etc., are written to file. These files are used to prepare Results vs Time maps. The file names is .

- **Spatial distribution of pollutant concentrations for each report interval:** For each output interval cell concentrations are written to file. These files are used to prepare Pollutant Concentration vs Time maps. The file name is .- **Oil particle coordinates, and properties for each report interval:** For each output interval particle coordinates, volume, density, viscosity, and oil state are written to file. These files are used by prepare oil property vs Time maps. The file name is .

- **Maximum Values at Cells 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 for generating maps. The files named as follows:. See Comment 1.

- **Time-to-Depth at Cells File:** This file reports the time at which certain depths are reached during the simulation, inundation time, etc. and allow seamless transfer to QGIS. The files are named as follows:. See Comment 1.

- **Hazard Intensity Values at Cells File:** 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. and allow seamless transfer to QGIS for map preparation. The files are named as follows:. See Comment 1.

12.1.0.1 Comments for Output Files

In the OilFlow2D model these files are generated during the final step after the model completes the run, and when post processing results using the Plot OilFlow2D results on the *Data Input Program Graphic Output Options* panel.

12.1.1 Essential files required to generate maps, graphics and animations

As it is clear from the list of files given above, OilFlow2D creates a significant number files containing model results, and some of them may be huge for large project. However, only a subset of these files are required to create graphs in OilFlow2D. Knowing which output files are required is often of practical importance when there is a need to reduce the number of files to transfer to a computer different from that used to perform the simulations. One example is when using a cloud service to perform simulations and the user needs to download result files to a local computer. Downloading only the essential files for postprocessing will help minimizing connection costs.

This section summarizes the essential files to create maps, graphics and animation in a OilFlow2D project. This assumes the existing project has the layers created to generate the OilFlow2D files such as *Trimesh*, etc.

The following table presents the various graphics and animations that can be created with OilFlow2D and the output files necessary for each graph.

> **Note**
> The are the native result output files of the model, that are always in SI units. Although are not directly used for graphic output, they can be helpful to regenerate the post processing files if you delete post processing files described in this section.

Results vs Time Maps
1..
2..
3..
Pollutant Concentration vs Time Maps
1..
2..Oil or Plastic location and properties for the OilFlow2D model. Time maps and animations.
1..
2..
3..
Maximum Result Maps
1..
Time-to-Depth Maps
1..
Hazard Intensity Maps
1..
Animations, Cross Sections and Profiles

1.

2..
3..

4. for the PL module. ## General Output Files

This section describes the content of each output file.

12.1.2 Output times .outfiles file

This file includes a list of times corresponding to each output interval. The following is an example of the content of a typical file:

```
time_0000_00_00_00.exp
time_0000_00_06_00.exp
time_0000_00_12_00.exp
time_0000_00_18_00.exp
time_0000_00_24_00.exp
time_0000_00_30_00.exp
time_0000_00_36_00.exp
```

12.1.3 Output times for the Oil Spill on Water model .outfilesoilw file

This file includes a list of times corresponding to each output interval generated when running the oil spill on water model. The following is an example of the content of a typical file:

_0000_02_00_00
_0000_02_15_00
_0000_02_30_00
_0000_02_45_00
_0000_03_00_00
_0000_03_15_00
_0000_03_30_00
_0000_03_45_00

12.1.4 Run Options Summary .outi and .oute files

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. The file is in metric units and in English units. Part of a typical output is as follows:

```
================================================================================
                RiverFlow2D Plus - Release  CPU  6.11.01
                                   Build    NOV 04 2018
================================================================================
             TWO-DIMENSIONAL FINITE-VOLUME RIVER DYNAMICS MODEL
                  (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                            ALL RIGHTS RESERVED
                            RUN DATE: 14/DEC/2018
================================================================================
================================================================================
            GRAPHICAL USER INTERFACE
  QGIS was used to process RiverFlow2D input files.
================================================================================
            TIME CONTROL OPTIONS
================================================================================
  Simulation time                :     2.000 hrs.
  Report output interval         :     0.100 hrs.
  Use variable time-steps        : ON.
  Courant-Friederich-Lewy (CFL)  :     1.000
  Start simulation               : Time = 0.
  Manning n multiplier           :      1.00
  Input units                    : Metric.
  Depth tolerance for dry bed  :    -1.000 [model controlled].
  Wetting-drying method          : Plus.
================================================================================
            MODULE OPTIONS
================================================================================
  Sediment Transport        : OFF.
  Mud/Debris Flow           : OFF.
  Pollutant Transport       : OFF.
================================================================================
            COMPONENT OPTIONS
================================================================================
  Bridges                   : OFF.
  Culverts                  : OFF.
  Internal rating tables    : OFF.
  Gates                     : OFF.
  Dams                      : OFF.
  Piers                     : OFF.
  Rainfall/evaporation      : OFF.
  Sources/Sinks             : OFF.
  Weirs                     : OFF.
  Wind                      : OFF.
================================================================================
            MESH DATA
================================================================================
 Number of elements/cells       :     1965
 Number of nodes                :     1048
 Total mesh area                :    0.551E+00 km2.
 Average element area           :    0.280E+03 m2.
 Average element size (approx.) :       23.682 m.
 Element with minimum area      :         1814
 Minimum element size (approx.) :       13.655 m.
 Minimum area                   :       93.224 m2.
 Element with maximum area      :       122
 Maximum element size (approx.) :       34.653 m.
 Maximum area                   :      600.408 m2.
 Minimum element angle          :       32.080
 Element with minimum angle     :         1965
 Maximum bed elevation             :       56.470
```

```
================================================================================
             OPEN BOUNDARIES
================================================================================
  Number of Open Boundaries:    2

   Number of nodes on open boundary  1:      21
   node   bc type  bc file name
     75      12     0.03
    117      12     0.03
     74      12     0.03
     73      12     0.03
     72      12     0.03
     71      12     0.03
     70      12     0.03
     69      12     0.03
     68      12     0.03
     67      12     0.03
     66      12     0.03
     65      12     0.03
     64      12     0.03
     63      12     0.03
     62      12     0.03
     61      12     0.03
     60      12     0.03
     59      12     0.03
     58      12     0.03
     57      12     0.03
     56      12     0.03

   Number of nodes on open boundary  2:       7
   node   bc type  bc file name
      6       6     QIN.DAT
    110       6     QIN.DAT
      5       6     QIN.DAT
      4       6     QIN.DAT
      3       6     QIN.DAT
      2       6     QIN.DAT
      1       6     QIN.DAT

  Boundary    1 has uniform flow condition.

    Cross section for boundary No.    1

 Node  Distance Bed elevation
           m.         m.
        75      0.00    163.47
       117     60.55    163.78
        74    121.09    164.00
        73    221.09    164.00
        72    321.09    163.87
        71    421.09    163.74
        70    521.09    163.24
        69    621.09    159.01
        68    721.09    157.06
        67    821.09    155.51
        66    921.09    153.79
        65   1021.09    154.51
        64   1121.09    159.43
        63   1221.09    161.44
        62   1321.09    161.47
        61   1421.09    161.27
        60   1521.09    162.54
        59   1621.09    164.35
        58   1721.09    167.54
        57   1821.09    173.44
        56   1921.09    177.04
```

12.1.5 Mesh Data and Mesh Metrics .meshouti and .meshoute files

Mesh data is written to files with extensions: (metric units) and (English units). These files provide comprehensive information about the triangular-cell mesh. The following table summarizes the available output.

- **Number of cells:** Total number of cells in the mesh
- **Number of nodes:** Total number of nodes in the mesh

- **X:** x-coordinate of node
- **Y:** y-coordinate of node
- **BEDEL:** Initial bed elevation
- **INITIAL_WSE:** Initial fluid surface elevation
- **BC ID:** Boundary condition code
- **BC File:** Boundary condition file name
- **Node1, Node2, Node3:** Nodes numbers of each cell in counterclockwise order
- **Manning's n:** Manning' n roughness coefficient
- **Area:** Cell area
- **Angle:** Minimum angle in cell
- **Total mesh area:** Sum of areas of all cells on the mesh
- **Average cell area:** Total mesh area divided by number of cells
- **Average cell size:** Average size of cells on mesh
- **Cell with minimum area:** Smallest cell
- **Minimum cell size:** Approximate linear size of smallest cell
- **Minimum cell area:** Area of smallest cell
- **Cell with maximum area:** Largest cell
- **Maximum cell size:** Approximate linear size of largest cell
- **Maximum cell area:** Area of largest cell
- **Minimum cell angle:** Smallest cell internal angle
- **Cell with minimum angle:** Cell that has the smallest internal angle

This file also reports the list of acute cells that have an internal angle of less than 5 degrees. If there are acute cells, the model will give an error message and will not be able to execute.
An excerpt of a typical file format is shown below:

```
================================================================================
================================================================================
                RiverFlow2D Plus - Release  CPU  6.11.01
                                   Build    NOV 04 2018
================================================================================
             TWO-DIMENSIONAL FINITE-VOLUME RIVER DYNAMICS MODEL
                  (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                            ALL RIGHTS RESERVED
                            RUN DATE: 06/DEC/2018
================================================================================
================================================================================
            MESH DATA
================================================================================
 Number of elements/cells       :     1803
 Number of nodes                :      963

   Node        X           Y       BEDEL    Initial WSE  BCType      BC File Name
               (m)         (m)       (m)       (m)
      1   798610.25   309314.51    183.62      0.00      6                          QIN.DAT
      2   798682.26   309383.90    166.34      0.00      6                          QIN.DAT
      3   798754.27   309453.28    161.56      0.00      6                          QIN.DAT
   ...
     73   798854.66   306009.70    164.00      0.00     12                             0.03
     74   798761.72   306046.62    164.00      0.00     12                             0.03
     75   798649.18   306091.32    163.47      0.00     12                             0.03
  ...
    962   799630.89   307402.41    168.31      0.00      0                                0
    963   799885.60   305990.72    161.14      0.00      0                                0

  ELEMENT/CELL CONECTIVITY AND MANNING's n

  Element    Node1    Node2    Node3  Manning's n     Area        Angle
                                                     m2          deg.
       1      897      160      400      0.035       2827.00     40.48
       2      363      160      897      0.035       3048.55     36.34
       3      623      225      273      0.035       2975.26     46.67
       4      124      462      619      0.035       2615.70     48.62
...
    1801      858      857      370      0.035       2301.06     54.04
    1802      718      935       57      0.035       1679.56     45.25
    1803      949       76       75      0.035       1586.29     31.70

================================================================================
 Total mesh area                :    0.532E+01 km2.
 Average element area           :    0.295E+04 m2.
 Average element size (approx.) :       76.828 m.
 Element with minimum area      :         1466
 Minimum area                   :     1088.114 m2.
 Minimum element size (approx.) :       46.650 m.
 Element with maximum area      :          158
 Maximum element size (approx.) :      109.155 m.
 Maximum element area           :     5957.432 m2.
 Minimum element angle          :       31.699
 Element with minimum angle     :         1803
 Maximum bed elevation          :        0.000
 Minimum bed elevation          :        0.000


================================================================================
  ACUTE ELEMENT REPORT (Elements with an internal angle less than 22.5 degrees.)

                     (Elements with an internal angle less than 5 degree are m
 arked with *<5.)
================================================================================

  The mesh has no acute angle elements.


================================================================================
              EXTERNAL BOUNDARY NODES (COUNTERCLOCKWISE)
================================================================================
  Number of Nodes on Boundary:          122
    Node        X             Y
                m.            m.
     121    798648.537    309256.390
     109    798686.825    309198.266
 ...
     121    798648.537    309256.390
```

12.1.6 Run Summary .rout file

Run summary report is written to file with extension:. 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), and volume conservation error (%).

12.1.6.1 Example of a .rout file

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Nov  4 2018
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                      (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 06/Dec/2018
================================================================================

================================================================================
                           MODEL RUN REPORT
================================================================================
    TIME          CPU Time       Ave.DT       Open Boundary          Q      Vol. Error       Qs        Mass Error
                                   (s)                              (m3/s)      (%)          (m3/s)        (%)
0000:00:06:00   0000:00:00:00     2.222            01           14199.83   0.000e+000       0.00      0.000e+000
                                                   02               0.00   0.000e+000       0.00      0.000e+000
0000:00:12:00   0000:00:00:00     1.809            01           27385.05   0.000e+000       0.00      0.000e+000
                                                   02               0.00   0.000e+000       0.00      0.000e+000
0000:00:18:00   0000:00:00:00     1.565            01           40561.07   5.035e-014       0.00      5.035e-014
                                                   02            6297.06   5.035e-014       0.00      5.035e-014
0000:00:24:00   0000:00:00:00     1.422            01           53797.19   0.000e+000       0.00      0.000e+000
                                                   02           28126.91   0.000e+000       0.00      0.000e+000
0000:00:30:00   0000:00:00:00     1.332            01           66987.93   0.000e+000       0.00      0.000e+000
                                                   02           45701.26   0.000e+000       0.00      0.000e+000
0000:00:36:00   0000:00:00:00     1.268            01           67000.00   1.653e-014       0.00      1.653e-014
                                                   02           62257.67   1.653e-014       0.00      1.653e-014
0000:00:42:00   0000:00:00:01     1.222            01           67000.00   3.240e-014       0.00      3.240e-014
                                                   02           65813.14   3.240e-014       0.00      3.240e-014
0000:00:48:00   0000:00:00:01     1.190            01           67000.00   0.000e+000       0.00      0.000e+000
                                                   02           66716.17   0.000e+000       0.00      0.000e+000
0000:00:54:00   0000:00:00:01     1.166            01           67000.00   0.000e+000       0.00      0.000e+000
                                                   02           66585.47   0.000e+000       0.00      0.000e+000
0000:01:00:00   0000:00:00:01     1.148            01           67000.00   3.219e-014       0.00      3.219e-014
                                                   02           66985.45   3.219e-014       0.00      3.219e-014
0000:01:06:00   0000:00:00:02     1.133            01           67000.00   4.828e-014       0.00      4.828e-014
                                                   02           66998.03   4.828e-014       0.00      4.828e-014
0000:01:12:00   0000:00:00:02     1.122            01           67000.00   6.438e-014       0.00      6.438e-014
                                                   02           66996.58   6.438e-014       0.00      6.438e-014
0000:01:18:00   0000:00:00:02     1.111            01           67000.00   0.000e+000       0.00      0.000e+000
                                                   02           67000.48   0.000e+000       0.00      0.000e+000
0000:01:24:00   0000:00:00:02     1.103            01           67000.00   0.000e+000       0.00      0.000e+000
                                                   02           66999.88   0.000e+000       0.00      0.000e+000
0000:01:30:00   0000:00:00:02     1.096            01           67000.00   3.219e-014       0.00      3.219e-014
                                                   02           66650.54   3.219e-014       0.00      3.219e-014
0000:01:36:00   0000:00:00:03     1.090            01           67000.00   1.609e-014       0.00      1.609e-014
                                                   02           66652.01   1.609e-014       0.00      1.609e-014
0000:01:42:00   0000:00:00:03     1.085            01           67000.00   3.219e-014       0.00      3.219e-014
                                                   02           67001.08   3.219e-014       0.00      3.219e-014
0000:01:48:00   0000:00:00:03     1.080            01           67000.00   1.609e-014       0.00      1.609e-014
                                                   02           66651.90   1.609e-014       0.00      1.609e-014
0000:01:54:00   0000:00:00:03     1.076            01           67000.00   4.828e-014       0.00      4.828e-014
                                                   02           66648.49   4.828e-014       0.00      4.828e-014
0000:02:00:00   0000:00:00:03     1.072            01           67000.00   0.000e+000       0.00      0.000e+000
                                                   02           66651.47   0.000e+000       0.00      0.000e+000
            MAXIMUM VALUES AT CELLS FILES CREATED.   3.1250000E-02
            FLOOD HAZARD AT CELLS FILES CREATED   1.5625000E-02
            PROGRAM EXCECUTION AND OUTPUT PROCESS COMPLETED.
```

12.1.7 Maximum Value Tabular .maxi and .maxe Files

These files report maximum nodal values of velocity module, depth, water surface elevations, and bed changes over the complete simulation. is in metric units and in English units. The reported variables are described in the following tables:

- **CELL:** Cell number; -; -

- **2:** VELOCITY; Maximum velocity magnitude $\sqrt{U^2 + V^2}$; ft/s; m/s
- **3:** DEPTH; Maximum water depth; ft; m
- **4:** WSEL; Maximum water surface elevation; ft; m
- **6:** DEPTHxVEL; Maximum product of depth and velocity; ft^2/s; m^2/s
- **7:** SHEAR STRESS; Maximum shear stress; lb/ft^2; Pa
- **8:** IMPACT FORCE; Maximum unit impact force; lb/ft; N/m
- **CELL:** Cell number; -; -
- **2:** VELOCITY; Maximum velocity magnitude $\sqrt{U^2 + V^2}$; ft/s; m/s
- **3:** DEPTH; Maximum water depth; ft; m
- **4:** WSEL; Maximum water surface elevation; ft; m
- **5:** DEPTHxVEL; Maximum product of depth and velocity; ft^2/s; m^2/s
- **6:** BED ELEV.; Maximum bed elevation; ft; m
- **7:** MIN BED ELEV.; Minimum bed elevation; ft; m
- **8:** EROS. DEPTH; Maximum erosion depth; ft; m
- **9:** DEPOS. DEPTH; Maximum deposition depth; ft; m
- **10:** SHEAR STRESS; Maximum shear stress; lb/ft^2; Pa
- **11:** IMPACT FORCE; Maximum unit impact force per unit width; lb/ft; N/m

A typical output file follows:

```
================================================================================
================================================================================
                RiverFlow2D Plus - Release  CPU  6.11.01
                                   Build    NOV 04 2018
================================================================================
             TWO-DIMENSIONAL FINITE-VOLUME RIVER DYNAMICS MODEL
                  (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                           ALL RIGHTS RESERVED
                           RUN DATE: 14/DEC/2018
================================================================================

  ALL COLUMNS INDICATE TO MAXIMUM VALUES AT THE CORRESPONDING CELL IN SI UNITS


      CELL   VELOCITY      DEPTH        WSEL         DEPTHxVEL   SHEAR STRESS   IMPACT FORCE
             (m/s)          (m)         (m)          (m2/s)        (Pa)          (kg/m)
        1     3.851       9.021       168.865        34.738      0.856E+02      0.533E+06
        2     5.563       4.271       168.024        22.461      0.270E+03      0.208E+06
        3     6.428      17.949       179.659       114.008      0.192E+03      0.230E+07
        4     6.385       8.082       174.646        51.608      0.244E+03      0.650E+06
        5     5.120       5.624       167.624        28.796      0.177E+03      0.303E+06
        6     5.526       5.352       167.666        29.572      0.210E+03      0.304E+06
        7     5.567      15.771       179.474        87.580      0.149E+03      0.171E+07
        8     6.077      13.357       177.966        81.176      0.188E+03      0.137E+07
        9     6.780      10.889       177.887        73.443      0.250E+03      0.108E+07
       10     4.487      10.128       175.067        45.447      0.112E+03      0.707E+06
       11    10.115      13.293       169.890       129.237      0.547E+03      0.212E+07
       12     5.344       5.811       174.766        31.052      0.191E+03      0.332E+06
       13     5.374       5.765       174.959        30.977      0.194E+03      0.329E+06
       14     9.451      13.711       170.477       127.030      0.453E+03      0.210E+07
       15     3.272      14.252       168.286        46.510      0.553E+02      0.115E+07
       16     7.696      19.229       173.812       147.994      0.275E+03      0.295E+07
       17     7.014       4.225       167.569        15.600      0.137E+04      0.145E+06
       18     7.090      22.327       178.037       127.759      0.282E+03      0.318E+07
       19     5.706      18.560       180.811       105.900      0.148E+03      0.229E+07
       20     4.415      28.418       179.450        73.028      0.128E+03      0.415E+07
       21     6.554      14.428       179.029        94.027      0.251E+03      0.163E+07
       22     8.153      15.484       170.203       126.137      0.323E+03      0.220E+07
       23     3.789       5.325       174.986        20.141      0.119E+03      0.215E+06
       24     2.564       3.099       168.720         5.561      0.724E+02      0.571E+05
```

12.1.8 Observation Point Output

These files report time series of results at observation points. The program finds cell where the observation point point is located and writes the result time series of the following variables:

- **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

The file name for each cell is:
for metric units and
for English units.
Where is the name given to the observation point. For example: is the file name for time series results of. An example of this file is shown below.

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build May 16 2018
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                     (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                  ALL RIGHTS RESERVED
                                  RUN DATE: 14/Oct/2018
================================================================================

 RESULTS FOR CELL:     6781 OBSERVATION POINT ID: ptObs1
   LOCATED AT COORDINATE: (   799401.88),(   305706.13)

      TIME         U         V     VELOCITY    DEPTH      WSEL    BEDEL_ORI    BEDEL   DELTA_BED  FROUDE     QSX          QSY          QS
     (hours)    (ft/s)    (ft/s)   (ft/s)      (ft)       (ft)      (ft)       (ft)      (ft)              (ft2/s)      (ft2/s)      (ft2/s)
     0.10000     0.000     0.000     0.000     0.000    154.219    154.219    154.219     0.000     0.000   0.00E+00     0.00E+00     0.00E+00
     0.20000     0.000     0.000     0.000     0.000    154.219    154.219    154.219     0.000     0.000   0.00E+00     0.00E+00     0.00E+00
     0.30000     0.000     0.000     0.000     0.000    154.219    154.219    154.219     0.000     0.000   0.00E+00     0.00E+00     0.00E+00
     0.40000     0.000     0.000     0.000     0.000    154.219    154.219    154.219     0.000     0.000   0.00E+00     0.00E+00     0.00E+00
     0.50000    -2.754    -6.195     6.780     4.999    159.218    154.219    154.219     0.000     0.535     0.000        0.000        0.000
     0.60000    -3.817    -8.471     9.291     7.505    161.724    154.219    154.219     0.000     0.598     0.000        0.000        0.000
     0.70000    -4.055    -9.247    10.097     8.462    162.681    154.219    154.219     0.000     0.612     0.000        0.000        0.000
     0.80000    -4.091    -9.706    10.533     9.085    163.304    154.219    154.219     0.000     0.616     0.000        0.000        0.000
     0.90000    -4.107   -10.049    10.856     9.589    163.808    154.219    154.219     0.000     0.618     0.000        0.000        0.000
     1.00000    -4.103   -10.332    11.117    10.043    164.262    154.219    154.219     0.000     0.618     0.000        0.000        0.000
     1.10000    -4.130   -10.516    11.298    10.358    164.577    154.219    154.219     0.000     0.619     0.000        0.000        0.000
     1.20000    -4.153   -10.550    11.338    10.422    164.641    154.219    154.219     0.000     0.619     0.000        0.000        0.000
     1.30000    -4.164   -10.560    11.352    10.435    164.654    154.219    154.219     0.000     0.620     0.000        0.000        0.000
     1.40000    -4.172   -10.564    11.358    10.440    164.659    154.219    154.219     0.000     0.620     0.000        0.000        0.000
     1.50000    -4.185   -10.569    11.367    10.443    164.662    154.219    154.219     0.000     0.620     0.000        0.000        0.000
     1.60000    -4.199   -10.576    11.379    10.446    164.665    154.219    154.219     0.000     0.621     0.000        0.000        0.000
     1.70000    -4.213   -10.582    11.390    10.450    164.669    154.219    154.219     0.000     0.621     0.000        0.000        0.000
     1.80000    -4.236   -10.577    11.394    10.451    164.670    154.219    154.219     0.000     0.621     0.000        0.000        0.000
     1.90000    -4.253   -10.590    11.412    10.454    164.673    154.219    154.219     0.000     0.622     0.000        0.000        0.000
     2.00000    -4.270   -10.606    11.434    10.457    164.676    154.219    154.219     0.000     0.623     0.000        0.000        0.000
```

12.1.9 Hot Start `2binitialized.hotstart` File (Version 10, 2026)

The 2 binitialized . hotstart file restarts a simulation from previously computed results when the **Hot start** option is selected. By default, the file contains the values for the last report written by the model. The source-output time code selects the file used as initial conditions, while the time and output index set the restarted simulation values:

- TIME: restart time in seconds.

- DDDD_HH_MM_SS: time code for the source cell_time_DDDD_HH_MM_SS.textout file.
- IDX: restart output index.

The results in the referenced cell_time_DDDD_HH_MM_SS.textout file are used as initial conditions for all mesh cells when the simulation restarts. For example, if the simulation stops after 5 hours and the report interval is 1 hour, 2 binitialized . hotstart contains:

18000 0000_05_00_00 5

OilFlow2D can restart from any existing report time by reading the initial conditions from the cell_time_DDDD_HH_MM_SS.tex file identified in 2 binitialized . hotstart. To restart from a time other than the last calculated report, edit 2 binitialized . hotstart with the desired time in seconds, its corresponding time code, and the output index. For example, to hot start from hour 3 (10,800 seconds) when the report interval is 1 hour, use:

10800 0000_03_00_00 3

The **Hot start** option is useful for establishing initial conditions common to a series of simulations, such as different return periods. For example, run the model with a constant-discharge inflow until it converges to steady state. To begin a new scenario at time zero using cell_time_0000_24_00_00.textout as the initial condition, retain its source-output time code and set TIME and IDX to 0:

0 0000_24_00_00 0

When OilFlow2D runs with **Hot start** enabled, the data in cell_time_0000_24_00_00.textout defines the initial conditions. You may keep 2 binitialized . hotstart and the selected time-output file in a separate directory and copy them to the project folder for each scenario.
Note: cell_time_DDDD_HH_MM_SS.textout files are tied to the mesh used to create them. If you modify the mesh in any way, use a file created for that mesh.

12.1.10 Mass Balance Output File

The
file reports on the global mass/volume balance throughout the simulation. The file content varies depending on the module used according to the following format:
OilFlow2D–Hydrodynamics
Column 1: Time
Column 2: Accumulated water volume inflow
Column 3: Accumulated water volume outflow
Column 4: Internal water volume
Column 5: Accumulated rain/evaporation water volume
Column 6: Accumulated infiltration water volume
OilFlow2D–PL
Column 1: Time
Column 2: Accumulated water volume inflow
Column 3: Accumulated water volume outflow
Column 4: Internal water volume
Column 5: Accumulated rain/evaporation water volume
Column 6: Accumulated infiltration water volume

Column 7: Accumulated solute_1 volume inflow
Column 8: Accumulated solute_1 volume outflow
Column 9: Internal solute_1 volume
Column 10: Accumulated uptake solute_1 volume
⋮ Column 11+k: Accumulated solute_k volume inflow
Column 12+k: Accumulated solute_k volume outflow
Column 13+k: Internal solute_k volume
Column 14+k: Accumulated uptake solute_k volume

OilFlow2D–UD

Column 1: Time
Column 2: Accumulated water volume inflow
Column 3: Accumulated water volume outflow
Column 4: Internal water volume
Column 5: Accumulated rain/evaporation water volume
Column 6: Accumulated infiltration water volume

OilFlow2D

Column 1: Time
Column 2: Accumulated oil volume inflow
Column 3: Accumulated oil volume outflow
Column 4: Internal oil volume
Column 5: Accumulated intake/evaporation oil volume
Column 6: Accumulated infiltration oil volume

OilFlow2D–HT

Column 1: Time
Column 2: Accumulated oil volume inflow
Column 3: Accumulated oil mass inflow
Column 4: Accumulated oil volume outflow
Column 5: Accumulated oil mass outflow
Column 6: Internal oil volume
Column 7: Internal oil mass
Column 8: $NULL$
Column 9: $NULL$
Column 10: $NULL$
Column 11: $NULL$

12.2 Component Output Files

12.2.1 Booms .OUTBOOMS Output File

When considering Booms in the OilFlow2D oil-on-water spill model creates an output file with extension: , that reports on the oil or plastic volumes in bbl and m^3 retained by each boom and for each output interval:

```
OIL/PLASTIC BOOM RETENTION BALANCE (BBL and m3)

       TIME      BOOM_1      BOOM_2      BOOM_1      BOOM_2
       (hr)      (bbl)       (bbl)         (m3)        (m3)
       0.10        0.00        0.00         0.00        0.00
       0.20      201.27        0.00        32.00        0.00
       0.30     3295.86      176.11       524.00       28.00
       0.40     4100.96      578.66       652.00       92.00
       0.50     4453.19     1006.37       708.00      160.00
       0.60     4830.57     1509.55       768.00      240.00
       0.70     4981.53     1987.58       792.00      316.00
       0.80     5107.33     2440.45       812.00      388.00
       0.90     5207.96     2717.20       828.00      432.00
       1.00     5333.76     3069.43       848.00      488.00
       1.10     5459.56     3295.86       868.00      524.00
       1.20     5610.51     3924.84       892.00      624.00
       1.30     5836.94     4327.39       928.00      688.00
       1.40     6038.22     4679.62       960.00      744.00
       1.50     6214.33     5283.44       988.00      840.00
       1.60     6491.08     5585.35      1032.00      888.00
       1.70     6767.84     6113.70      1076.00      972.00
       1.80     6893.63     6465.93      1096.00     1028.00
       1.90     7195.54     6742.68      1144.00     1072.00
       2.00     7296.18     7120.07      1160.00     1132.00
```

12.2.2 Culvert CULVERT_culvertID.out Output Files

For each culvert, OilFlow2D creates an output file named: , where is the text provided by the user to identify the culvert. Report includes discharge for each report interval and the water surface elevations (WSEL1, WSEL2) at each culvert end as shown:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Jun 29 2018
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 20/Aug/2018
================================================================================

================================================================================
Results for Culvert no.:                1 Culvert ID: Ret01
================================================================================
 Time       Qc       WSEL1     WSEL2
 hrs.             m3/s      m.        m.
0.10000   0.000   843.631    843.572
0.20000   14.427   844.737    843.572
0.30000   16.703   844.220    844.091
0.40000   17.093   843.993    844.321
0.50000   17.264   844.496    843.926
0.60000   20.183   844.151    844.228
0.70000   20.222   843.699    844.605
0.80000   20.292   844.911    843.615
0.90000   18.133   843.631    844.699
1.00000   21.038   843.929    844.507
```

12.2.3 Internal Rating Table IRT_irtID.out Files

For each Internal Rating Table, OilFlow2D creates an output file named: , where is the text provided by the user to identify the Internal Rating Table. Report includes discharge for each report interval as shown:

```
================================================================================
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
```

12.2.4 Weir Output .weiri and .weire Files

These files report results for each weir and for each output interval. File extension is for metric units and for English units. Output includes the following information:

- **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:

```
================================================================================
 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
```

12.3 Cross Section and Profile Output Files

12.3.1 General Cross Section .xseci and .xsece Files

When using the *Output results* for *cross sections* option, the model will generate files with extensions and , that report results along user provided cross sections. For each output interval and for each user defined cross section, the bed elevation, depth, water surface elevation, depth average velocity, and Froude number are written to file in metric and English units. A typical file is as follows:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Nov  4 2018
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 07/Dec/2018
================================================================================

 CROSS SECTION RESULTS IN SI UNITS

  CROSS SECTION NO.:   1 CROSS SECTION ID: X1

(     243546.87,       94395.30), (     243406.27,       94246.14)
ELEM    STATION    BEDEL     DEPTH     WSEL     VELOCITY  FROUDE     QS
          (m)       (m)       (m)       (m)       (m/s)           (m2/s)
  259     33.13     51.61      2.31     53.92      3.73      0.78  0.000000
  589     39.34     51.52      2.47     53.99      3.63      0.74  0.000000
 1733     57.97     49.68      4.26     53.94      4.09      0.63  0.000000
 1676     72.47     48.98      5.09     54.08      3.58      0.51  0.000000
 1584     78.68     48.59      5.43     54.02      3.74      0.51  0.000000
 1731     89.03     48.84      5.09     53.93      3.94      0.56  0.000000
 1848     99.38     49.06      4.88     53.93      3.76      0.54  0.000000
 1841    107.67     49.21      4.70     53.91      3.83      0.56  0.000000
 1393    120.09     49.45      4.41     53.86      3.57      0.54  0.000000
 1654    138.72     49.44      4.45     53.89      3.41      0.52  0.000000
 1793    140.79     49.59      4.53     54.13      2.32      0.35  0.000000
  184    144.94     51.55      1.87     53.42      4.73      1.11  0.000000
 Q =   1905.999 m3/s.
```

When running only hydrodynamics the and files will display the cross section water discharge.

12.3.2 Cross Section Hydrograph .xsech and .xsecsed Files

These files will only be generated using the post processing Plot OilFlow2D results button on the *Graphic Output Options* panel. When using the *Output results for cross sections* option, the model will generate files with extension and, that report a hydrograph table for each cross section. A typical path.xsech

file is as follows:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Nov  4 2018
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 07/Dec/2018
================================================================================

 Hydrograph for cross sections in m^3/s, time in hours

  Time        Q-X1     Q-x2     Q-x3

0.000000      -0.000      -0.000      -0.000
0.100000     198.264      -0.000      -0.000
0.200000     387.976     180.700      -0.000
0.300000     564.022     407.099     134.082
0.400000     757.734     596.830     435.154
0.500000     950.410     810.896     656.584
0.600000    1142.998    1017.487     901.355
0.700000    1333.144    1224.109    1120.433
0.800000    1523.201    1416.349    1308.080
0.900000    1712.219    1612.098    1501.255
1.000000    1901.146    1804.103    1705.973
1.100000    1905.654    1877.983    1858.916
1.200000    1905.948    1884.967    1888.445
1.300000    1905.992    1885.975    1892.804
1.400000    1905.999    1886.126    1893.624
1.500000    1906.000    1886.320    1892.845
1.600000    1905.999    1886.168    1893.636
1.700000    1905.999    1886.166    1893.388
1.800000    1905.999    1886.166    1893.357
1.900000    1905.999    1886.165    1893.359
2.000000    1905.999    1886.165    1893.363
```

12.3.3 Profile .prfi and .prfe Files

When using the *Output results for profiles* option, the model will generate files with extensions and , 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. is in metric and in English units. An example output is shown below:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Nov  4 2018
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                       ALL RIGHTS RESERVED
                                       RUN DATE: 07/Dec/2018
================================================================================

 PROFILE RESULTS IN SI UNITS

 TIME: 0000 days,00 hours,00 min.,00 secs.

 PROFILE NO.:   1 PROFILE ID: Perfil1

 ELEM   DISTANCE    BEDEL     DEPTH      WSEL    VELOCITY  FROUDE
           (m)       (m)       (m)       (m)       (m/s)
   527     17.52     49.16      0.50     49.66      0.00      0.00
  1458     35.04     48.91      0.75     49.66      0.00      0.00
  1092     52.56     48.85      0.00     48.85      0.00      0.00
  1731     70.08     48.84      0.00     48.84      0.00      0.00
  1707     87.60     49.15      0.00     49.15      0.00      0.00
  1298    105.13     49.01      0.00     49.01      0.00      0.00
   517    122.65     49.11      0.00     49.11      0.00      0.00
  1568    140.17     49.01      0.00     49.01      0.00      0.00
  1402    157.69     48.96      0.00     48.96      0.00      0.00
   967    175.21     48.93      0.00     48.93      0.00      0.00
  1678    192.73     48.76      0.00     48.76      0.00      0.00
  1736    210.25     48.78      0.00     48.78      0.00      0.00
   182    227.77     48.82      0.00     48.82      0.00      0.00
   162    245.29     48.97      0.00     48.97      0.00      0.00
   238    262.81     48.88      0.00     48.88      0.00      0.00
    64    278.94     49.13      0.00     49.13      0.00      0.00
    25    295.07     49.16      0.00     49.16      0.00      0.00
   176    311.20     48.99      0.00     48.99      0.00      0.00
```

12.4 Output Files for QGIS Post-processing

12.4.1 General Results at Cells

These ASCII files allow seamless transfer to QGIS Geographic Information System software. These files use the extension and are named as follows:
Where dddd is days, hh is hours, mm is minutes and ss seconds. For example
corresponds to a file 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-cell mesh as shown:

- **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; -; -
- **13:** Bed shear stress*:; lb/ft^2; Pa & $\tau = \gamma H S_f = \gamma \left(Un/k\right)^2 / H^{1/3}$ &

- **14:** Accumulated rainfall volume; ft^3; m^3
- **15:** Accumulated infiltration volume; ft^3; m^3
- **16:** Manning's n; -; -

12.4.2 Oil Spill on Land Considering Heat Transfer Concentration Files (OilFlow2D Overland Spills Module)

These ASCII files contains oil properties for each cell when the user selects the option to compute heat transfer in the OilFlow2D model. These files use the extension and are named as follows:
Where dddd is days, hh is hours, mm is minutes and ss seconds. For example
corresponds to a file for time: 1 day, 12 hours, 1 minute and 34 seconds.
The files contains NELEM lines with results for each cell in the triangular-cell mesh as shown:

- **1:** Temperature; °C or °F
- **2:** Density; kg/m^3 or lb/ft^3
- **3:** Viscosity; Pa.s or lb.s/in^2
- **4:** Yield stress; Pa or lb/in^2

The following is an extract of a typical file:

```
24.285166  896.773201  0.127555  388.067542
33.863397  890.767650  0.084793  11.684067
24.291867  896.768999  0.127518  387.241991
48.416306  881.642976  0.047768  0.000000
24.271945  896.781491  0.127630  389.696434
30.217557  893.053592  0.098751  45.954962
43.904228  884.472049  0.056749  0.000000
46.777124  882.670743  0.050825  0.000000
24.264588  896.786103  0.127671  390.602726
24.429900  896.682453  0.126746  370.236305
24.363673  896.723977  0.127116  378.395478
```

12.4.3 Pollutant Concentration Files (PL Module)

These ASCII files contains pollutant transport module results. These files use the extension and are named as follows:
Where dddd is days, hh is hours, mm is minutes and ss seconds. For example
corresponds to a file 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 (NP_MAX). Then follows NELEM lines with results for each cell in the triangular-cell mesh as shown:

- **1:** Concentration for solute 1; Same as in BC's
- **2:** Concentration for solute 2; Same as in BC's
- **...:** ...; ...
- **NP_MAX:** Concentration for solute NP_MAX; Same as in BC's

The following file is an example of a typical file:

```
3
0.000000  0.000000  0.000000
0.000000  0.000000  0.000000
0.000000  0.000000  0.000000
0.202378  0.000000  0.000000
0.326602  0.000000  0.000000
0.291721  0.000000  0.000000
0.000000  0.000000  0.000000
...
```

In this example, the has 3 pollutants.

12.4.4 Oil and Plastics Output Files (OilFlow2D Spills On Water and Plastics Modules)

12.4.4.1 Particles in Mesh Files

These ASCII files report the oil or plastic particle coordinates, and properties for each report interval. These files use the extension and are named as follows:
Where dddd is days, hh is hours, mm is minutes and ss seconds. For example
corresponds to a file for time: 1 day, 12 hours, 1 minute and 34 seconds.
The format for these files is as follows. The first line indicates the total number of particles NP representing the oil or plastic. Then follows NP lines with results for each particle as shown:

- **1:** Time since released; hours
- **2:** Spill number; 1, 2,..., NSpillSites
- **3:** Cell in which the particle is located for this time; -
- **4:** X coordinate of the particle; m or ft
- **5:** Y coordinate of the particle; m or ft
- **6:** Z coordinate of the particle; m or ft
- **7:** Particle volume; m^3 or ft^3
- **8:** Particle density; Specific gravity

- **9:** Particle viscosity; cPoise (0.001 Pa.s) or (lb. s/ft^2)
- **10:** Particle state.
- Active inside the mesh (flowing)
- On shore
- Went out of the mesh
- On bottom
- Evaporated.

& -

The following file is an example of a typical file :

```
 1000
 0.000       1      108    583104.440   2859074.590       0.000    20.000    0.825
15.0     1
 0.000       1       91    583048.358   2859239.876       0.000    20.000    0.825
15.0     1
 0.000       1      124    583054.107   2859039.172       0.000    20.000    0.825
15.0     1
 0.000       1      124    583025.906   2859012.304       0.000    20.000    0.825
15.0     1
 0.000       1      154    582984.811   2858954.262       0.000    20.000    0.825
15.0     1
 0.400       1     2089    583076.449   2859329.372       0.000    20.000    0.825
15.0     1
 0.400       1     2171    583077.807   2859406.086       0.000    20.000    0.825
15.0     1
 0.800       1      145    583011.099   2858829.013       0.000    20.000    0.825
15.0     1
 0.800       1       66    582971.325   2858846.586       0.000    20.000    0.825
15.0     1
 1.100       1    20438    582205.981   2857441.104       0.000    20.000    0.825
15.0     1
 1.100       1    20453    582184.220   2857537.876       0.000    20.000    0.825
15.0     1
 1.300       1    20353    582250.333   2857832.535       0.000    20.000    0.825
15.0     1
 1.300       1    20733    582673.118   2858336.860       0.000    20.000    0.825
15.0     1
 1.500       1      117    583186.693   2858579.932       0.000    20.000    0.825
15.0     1
```

```
 1.500       1       73     583148.156   2858460.752       0.000     20.000     0.825
15.0     1
```

Note that in no data cells, particle coordinates X and Y are equal to -9999.

12.4.4.2 Water Hydrodynamics and Oil-Plastic Volumes in Mesh Files

These ASCII files report the oil or plastic volumes for each spill point at all mesh cells for each report interval. These files use the extension and are named as follows:
Where dddd is days, hh is hours, mm is minutes and ss seconds. For example
corresponds to a file for time: day 0, 6 hours, 1 minute and 54 seconds.
The format for these files is as follows. The first line indicates the total number of cells NELEM and the number of spill points NSPILLS. Then follows NELEM lines with results for each cells as shown on table :

- **Water velocity component in x direction U:** ft/s; m/s
- **2:** Water velocity component in y direction V; ft/s; m/s
- **3:** Water velocity magnitude $\left|\vec{U}\right| = \sqrt{U^2 + V^2}$; ft/s; m/s
- **4:** Water surface elevation; ft; m
- **5:** WaterDepth H; ft; m
- **6:** Bed elevation; ft; m
- **7:** Oil volume per unit area in cell for spill 1; ft; m
- **8:** Oil volume per unit area in cell for spill 2; ft; m
- **NSPILLS + 6:** Oil volume per unit area in cell for spill NSPILLS; ft; m

The following file is an example of a typical file:

```
    8260  2
-0.414E+00  0.343E+00  0.538E+00  0.353E+01  0.846E+01  -0.493E+01  0.375E-01  0.000E+00
-0.344E+00  0.303E+00  0.458E+00  0.353E+01  0.343E+01  0.984E-01  0.250E-01  0.000E+00
-0.344E+00  0.288E+00  0.449E+00  0.353E+01  0.364E+01  -0.115E+00  0.250E-01  0.000E+00
-0.392E+00  0.331E+00  0.513E+00  0.353E+01  0.813E+01  -0.460E+01  0.375E-01  0.000E+00
-0.321E+00  0.288E+00  0.431E+00  0.353E+01  0.666E+01  -0.313E+01  0.250E-01  0.000E+00
-0.377E+00  0.351E+00  0.515E+00  0.353E+01  0.550E+01  -0.197E+01  0.625E-01  0.000E+00
-0.378E+00  0.276E+00  0.467E+00  0.353E+01  0.603E+01  -0.250E+01  0.375E-01  0.000E+00
-0.337E+00  0.302E+00  0.453E+00  0.353E+01  0.826E+01  -0.473E+01  0.625E-01  0.000E+00
-0.360E+00  0.298E+00  0.468E+00  0.353E+01  0.813E+01  -0.460E+01  0.250E-01  0.000E+00
-0.441E+00  0.328E+00  0.550E+00  0.353E+01  0.813E+01  -0.460E+01  0.000E+00  0.000E+00
-0.343E+00  0.330E+00  0.476E+00  0.353E+01  0.653E+01  -0.300E+01  0.375E-01  0.000E+00
-0.573E-01  0.788E-02  0.578E-01  0.353E+01  0.470E+00  -0.306E+01  0.000E+00  0.000E+00
-0.247E+00  0.199E+00  0.317E+00  0.353E+01  0.200E+01  0.153E+01  0.250E-01  0.000E+00
-0.267E+00  0.269E+00  0.379E+00  0.353E+01  0.438E+01  -0.850E+00  0.500E-01  0.000E+00
     . . .
```

12.4.4.3 Maximum Oil-Plastic Volume per Unit Area File

This ASCII files report maximum oil or plastics volume per unit area allow seamless transfer to QGIS Geographic Information System software. This file use the extension and are named as follows: The format for these file is as follows. The first line contains the number of cells (NELEM), and the number of spills (NSPILLS). Then follows NELEM lines with each column indicating the maximum oil volume per unit area for each spill as shown on table. Dry cells are indicated with the number -9999.000.

- **1:** Maximum oil volume per unit area for spill 1; ft; m
- **2:** Maximum oil volume per unit area for spill 2; ft; m
- **...:** ...; ...; ...
- **NSPILLS:** Maximum oil volume per unit area for spill NSPILLS; ft; m

The following file is an example of a typical file:

```
        7086              2
2.5281593E-05  4.2135565E-04
-9999.000  -9999.000
-9999.000  -9999.000
2.7736165E-05  -9999.000
8.1205413E-05  6.7670504E-04
2.3327330E-04  1.1663549E-03
1.1482510E-04  4.7843312E-04
2.0842110E-04  2.9774143E-03
...
```

12.4.4.4 Oil Global Mass Balance .OUTOILVOL

This ASCII file reports the oil global mass balance throughout the simulation.
The file is named and includes for each time the following values:

- IN MESH: Oil volume that is moving.
- OUT MESH: Oil volume that has exit through the model open boundaries.
- ON SHORE: Oil volume on the mesh closed boundaries.
- ON BOTTOM: Oil volume on the bottom.
- TOTAL: Total oil volume that should equal the sum of IN MESH, OUT MESH, ON SHORE, and ON BOTTOM.

The volumes are given in barrels (BBL) and m^3.
The following file is an example of a typical file:

```
================================================================================
================================================================================
OilFlow2D  -  Release  CPU  8.04
Build  SEP  12  2022
================================================================================
TWO-DIMENSIONAL  FINITE-VOLUME  OIL  FLOW  MODEL
```

```
(R) TRADEMARK 2009-2022 Hydronia, LLC.
ALL RIGHTS RESERVED
RUN DATE: 22/SEP/2022
================================================================================
```

TIME (h)	IN MESH (m3)	OUT MESH (m3)	ON SHORE (m3)	ON BOTTOM (m3)	TOTAL (m3)
0.010	1020.044	0.000	0.000	0.000	1020.044
0.020	1040.040	0.000	0.000	0.000	1040.040
0.030	1060.035	0.000	0.000	0.000	1060.035
0.040	1080.030	0.000	0.000	0.000	1080.030
0.050	1011.443	0.000	0.000	88.600	1100.043
0.060	797.030	0.000	0.000	323.001	1120.031
0.070	598.218	0.000	0.000	541.815	1140.033

...

12.4.5 Maximum Value Files

These ASCII files report maximum values of velocity module, depth and water surface elevations, and other and allow seamless transfer to QGIS Geographic Information System software. These files use the extension and are named as follows:
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 6 by default. There follows NELEM lines with velocity module, depth and water surface elevation for each cell as shown:

- **1:** Maximum velocity magnitude $\sqrt{U^2 + V^2}$; ft/s; m/s
- **2:** Maximum depth; ft; m
- **3:** Maximum water surface elevation; ft; m
- **4:** Maximum depth x 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
- **9:** Maximum bed shear stress; lb/ft^2; Pa
- **10:** Maximum impact force per unit width; lb/ft; N/m
- **11:** Limiting DT times; -; -

12.4.6 Time-to-Depth at Cells Output File

The file reports the time at which certain depths are reached during the simulation and allow seamless transfer to QGIS Geographic Information System software. The time-to-depth files have the following name:
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. When the cell remains dry o depth is below 0.30 m the reported value is -1. Time is always given in hours.

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. When the cell remains dry o depth is below 1ft the reported value is -1.

The inundation time is computed as the total time during the simulation that cell depth is greater than 0. If the cell gets wet, then dries out and gets wet again, the intermediate dry period is not considered.

- **Time to 0.30 m (Metric) o 1 ft. (English)*:** h.
- **2:** Time to 0.50 m (Metric) o 2 ft. (English)*; h.
- **3:** Time to 1 m (Metric) o 3 ft. (English)*; h.
- **4:** Time to maximum depth*; h.
- **5:** Inundation time; h.
- **6:** Arrival time; h.

12.4.7 Hazard Intensity Values at Cells Output File

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, Australia and in the UK. The file can be used to create hazard maps in the QGIS Geographic Information System software. These files use the extension and are named as follows:

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:

- **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
- **12:** Custom Water Flood; $0, 1, 2,$ and 3
- **13:** Custom Debris Flow; $0, 1, 2,$ and 3
- **14:** Australia Flood Hazard; $0, 1, 2, 3,$ and 4
- **15:** ECWHM Equivalent Clear Water Hazard Map; $0 - 8$

12.4.7.1 USBR Hazard Levels

The USBR Hazard includes five attributes corresponding of hazard level for houses, mobile homes, vehicles, adults and children based on the United States Bureau of Reclamation classification of flood hazards (USBR, 1988). The attributes can get the values of 1, 2 o 3 depending on the hazard level summarized in the following table:
& Low-danger zone
- **2:** Judgment zone - **3:** High-danger zone
For further details about the USBR Hazard classification, consult USBR (1988).

12.5 VTK Output Files for Paraview

OilFlow2D model will output files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of OilFlow2D results. ParaView (https://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 using ParaView's batch processing capabilities.

13 OilFlow2D Tools

13.1 Automated Batch Execution of Model Simulations

You can perform multiple OilFlow2D runs using batch scripts. When controlling model runs through batch scripts, it is recommended to enable the *Automatic Close of Model Windows* output option in the DIP interface to ensure uninterrupted execution.
There are two available methods for performing multiple runs using batch scripts:

- Using a MS-DOS script in a file.
- Using a script written in the Python computer programming language.

> **Note**
>
> Note: The script files described in this chapter can be downloaded using the following link: BatchRun-RF2D-OF2D.zip

13.1.1 Using a MS-DOS script to perform multiple batch runs

```
C:
cd "C:\Program Files\Hydronia\OilFlow2D"
OilFlow2Dm5 "D:\OilFlow2D\Projects\ProjectA\Scenario1\Run1" > "D:\OilFlow2D\Proje
OilFlow2Dm5 "D:\OilFlow2D\Projects\ProjectA\Scenario2\Run2" > "D:\OilFlow2D\Proje
```

> **Warning**
>
> The redirect log filename must not match <projectname>.log. The OilFlow2D engine writes its

own log file using that exact name, and redirecting to the same path locks the file for the duration of the command, preventing the OilFlow2D Plus viewer from launching. Use a different suffix such as _batch.log (as shown above) or omit the redirect entirely.
Comment lines in MS-DOS batch files must start with REM or ::. The % character begins variable expansion, so any line that starts with % will be parsed and executed as a command rather than ignored as a comment.

13.1.2 Using a Python script within QGIS to perform multiple batch runs

This script runs within the QGIS Python Console, but has the disadvantage that blocks QGIS while running models.

```
# -*- coding: utf-8 -*-
from subprocess import call

oilflow2d_path = r"C:\Program Files\Hydronia\OilFlow2D\OilFlow2Dm5 "

dat_paths = [
            r"C:\Users\hydronia\Documents\OilFlow2D_QGIS\ExampleProjects\OilSpillLandTutoria
            r"C:\Users\hydronia\Documents\OilFlow2D_QGIS\ExampleProjects\OilSpillWaterTutor
             ]

for path in dat_paths:
    prjPath = oilflow2d_path + path
    call(prjPath)
```

13.1.3 Using a Python script through a batch file to perform multiple runs

This Python script also runs from within QGIS, but creates a .BAT file that does not block QGIS while the models are executing.

```
# -*- coding: utf-8 -*-
import tempfile
import subprocess
tmpdir = tempfile.mkdtemp(prefix='of2qgis')
temp = os.path.join(tmpdir, "temp.bat")

dat_paths = [
            r"C:\Users\hydronia\Documents\OilFlow2D_QGIS\ExampleProjects\OilSpillLandTutoria
            r"C:\Users\hydronia\Documents\OilFlow2D_QGIS\ExampleProjects\OilSpillWaterTutor
             ]
```

```
with open(temp, "w") as f:
  f.write("C:\n")
  f.write('cd "C:\Program Files\Hydronia\OilFlow2D"\n')
  for path in dat_paths:
      f.write('OilFlow2Dm5 "' + path + '" > "' + QFileInfo(path).absolutePath +
       '/Run.log" \n')

subprocess.Popen(temp, shell=True)
```