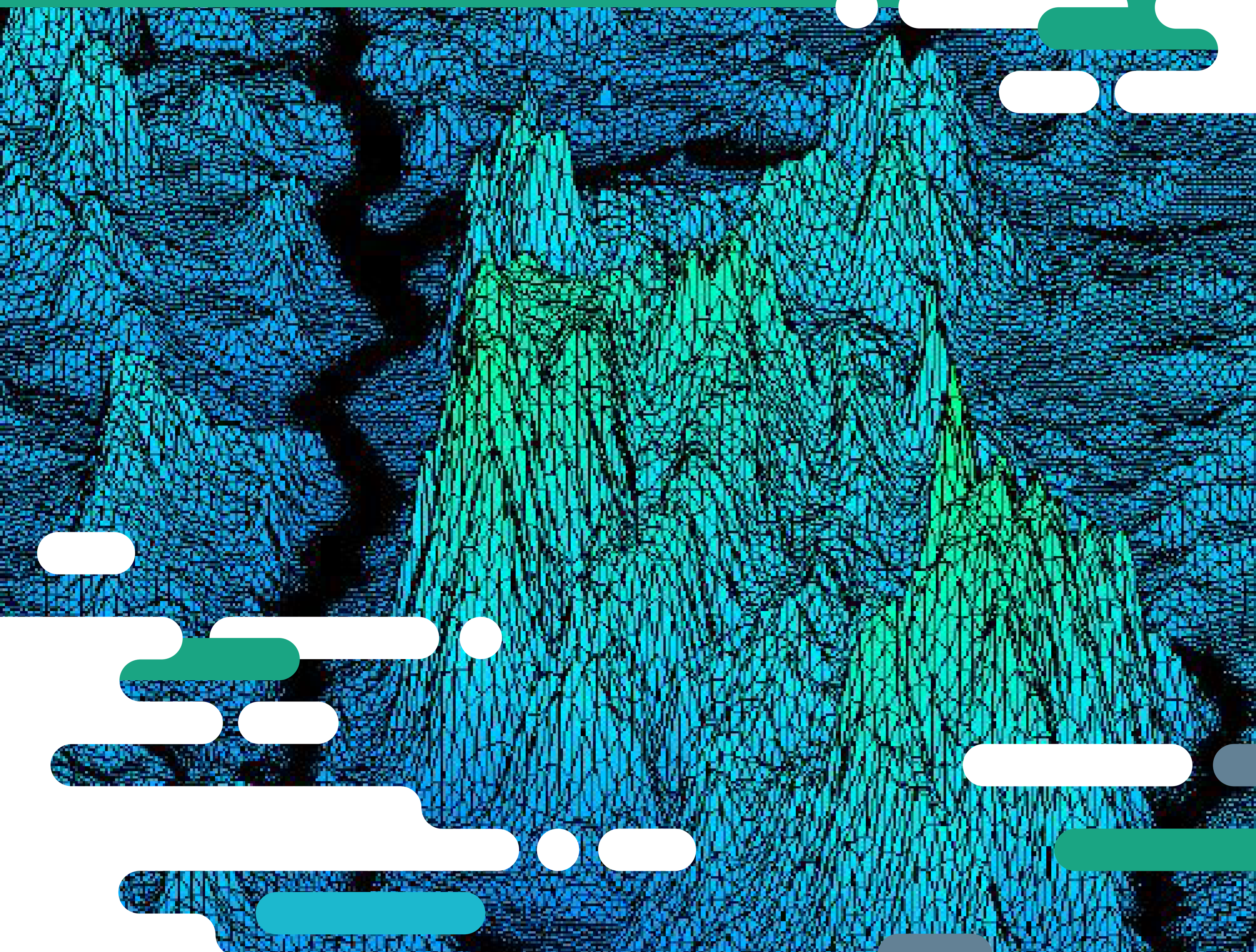
HydroBID

IDB

HydroBID Flood Reference Manual

Two-dimensional Flood and River Dynamics Model
QGIS Version



Water and Sanitation Division
February 2019

MINISTERIO DE ASUNTOS EXTERIORES, UNIÓN EUROPEA Y COOPERACIÓN
aecid
Cooperación Española
FCAS Fondo de Cooperación para Agua y Saneamiento

Autor: Hydronia LLC.

Keywords: Water, Floods, River, Manual, Water Resources.

Contents

List of Figures

List of Tables

1 — Introduction

HydroBID Flood is a combined hydrologic and hydraulic, mobile bed and pollutant transport finite-volume model for rivers, estuaries and floodplains based on the RiverFlow2D model[1]. It is part of Hydronia suite of models that includes OilFlow2D, and OilFlow2D GPU. RiverFlow2D can route floods in rivers and simulate inundation over floodplains and complex terrain at high resolution and with remarkable speed, stability, and accuracy. The use of adaptive triangular-cell meshes enables the flow field to be resolved around key features in any riverine environments.

This version of HydroBID Flood includes a Graphical User Interface (GUI) based upon a plug in developed by Hydronia for the Open Source Geographical Information System QGIS (www.qgis.org). The plugin was partially funded by the InterAmerican Development Bank. The integration of the RiverFlow2D model and the QGIS software system provides interactive functions to generate and refine the flexible mesh used by RiverFlow2D, 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 HydroBID Flood components, allowing the user to efficiently manage the entire modeling process. HydroBID Flood offers a comprehensive set of visualization tools including map rendering, animations, and exporting graphs in shapefile format and Google Earth.

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

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

[1] RiverFlow2D, and RiverFlow2D GPU are copyrighted by Hydronia, LLC. 2011-2019. OilFlow2D™ is a registered trademark of Hydronia, LLC.

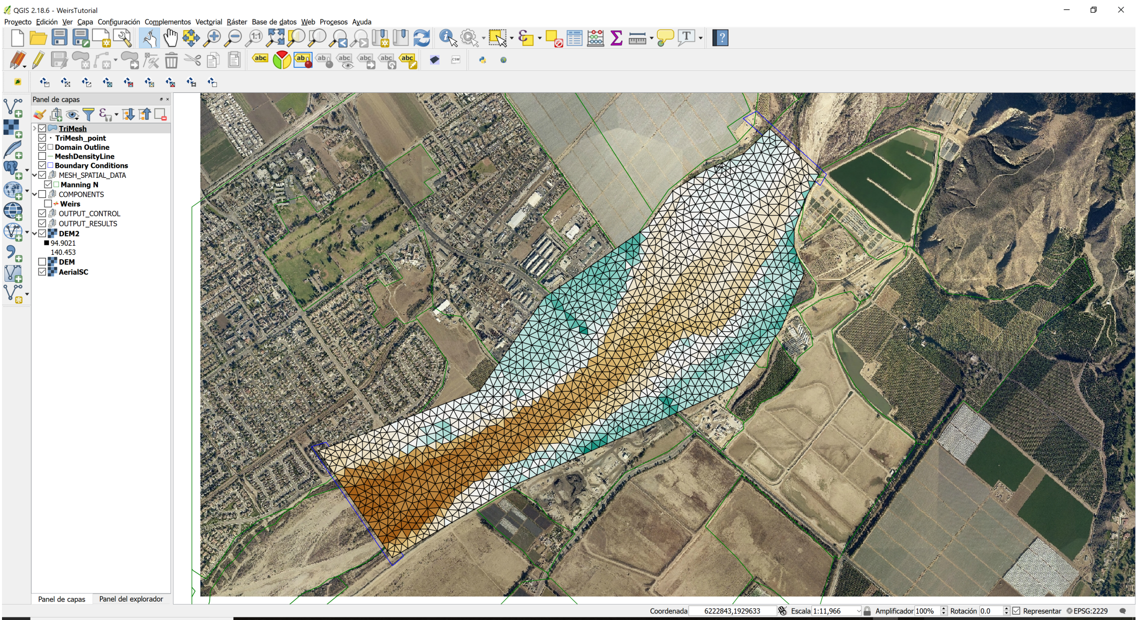


Figure 1.1 – RiverFlow2D triangular-cell mesh.

1.1 Summary of HydroBID Flood Features and Capabilities

1.1.1 Mesh Generator

- Automatic generation of flexible triangular-cell mesh
- Mesh refinement along density polylines
- Use of different DEMs for user selected areas
- Spatially interpolation of DEM elevations to cells

1.1.2 Numerical Engine

- Spatial discretization using triangular elements/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 >600X faster simulations using NVIDIA GPU Graphic Cards

1.1.3 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
- Internal hydraulic structures
- Sources and sinks
- Spatially distributed rainfall and evaporation
- Spatially distributed infiltration
- Spatially distributed wind stress
- Depth dependent Manning's n

1.1.4 Input Data Formats

- Metric or English units
- ASCII X, Y, Z
- ESRI ASCII grid files
- USGS DEM
- ESRI shapefiles
- Autodesk DXF
- TIFF, GIF, JPG, etc. raster's
- Any raster or vector data format accepted by QGIS.

1.1.5 Initial Conditions

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

1.1.6 Boundary Conditions

- Water discharge hydrograph
- Water discharge and water elevation vs time
- Water elevation vs time
- Uniform flow
- Rating tables
- Free outflow
- Inflow suspended sediment concentrations (ST Module)
- Inflow solid discharge for bed load option (ST Module)
- Inflow pollutant concentrations (PL Module)

1.1.7 Output Options

- Results at cross sections and profiles
- Observation points: time series at user selected locations
- Dynamic plots while the model runs
- Velocity field, depth and water surface elevations
- Bed shear stress
- Froude Number
- Accumulated rainfall
- Accumulated infiltration
- Erosion and deposition depths
- Bed elevation changes
- Sediment fluxes

- 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

1.1.8 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

1.1.9 Output for Hazard Assessments

- United States Bureau of Reclamation (USBR) Hazard for Homes
- United States Bureau of Reclamation (USBR) Hazard for Vehicles
- United States Bureau of Reclamation (USBR) Hazard for Adults
- United States Bureau of Reclamation (USBR) Hazard for Children
- Swiss method for flooding
- Swiss method for debris flow
- Austrian method for river flooding
- Austrian method for torrents Tr = 10 yrs.
- Austrian method for torrents Tr = 100 yrs.
- UK method
- Australia flood hazard

1.1.10 Sediment Transport Module (ST)

- Separate calculation of suspended and bed load sediment transport
- Multiple size fractions
- Bed-changes (erosion-deposition)
- 10 sediment-transport formulas
- Sediment transport over rigid bed
- Limited maximum bed elevation gradient
- Maximum erosion depth areas
- Coupled or decoupled computation

1.1.11 Mud and Debris Flow Module (MD)

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

1.1.12 Pollutant Transport Module (PL)

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

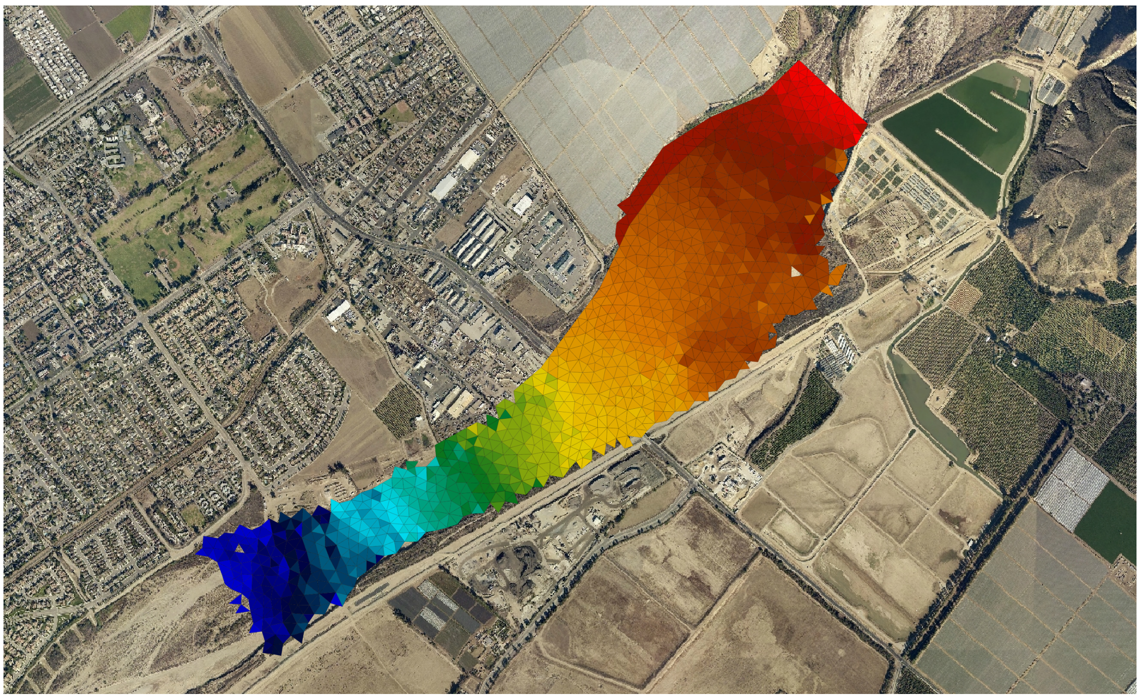


Figure 1.2 – Plot showing water surface elevations computed with HydroBID Flood.

2 — Installing HydroBID Flood

RiverFlow2D requires QGIS version 2.18. Presently Hydronia is developing the migration to QGIS 3.x. This section will assist you to setup HydroBID Flood to enable using the it from QGIS.

2.1 Hardware Requirements

HydroBID Flood is supported on 64-bit computers running MS-Windows 7, 8.1, 10 Operating Systems. It is recommended to have a minimum of 4 GB of RAM and at least 10 GB of free hard disk space. HydroBID Flood is capable of running in modern Intel single processor computers. If multiple-core processors (Duo, Quad, etc.) are available, the model can execute in parallel processor mode, thereby running much faster than in single processor computers. In addition, if you have a RiverFlow2D GPU license, the model can take advantage of NVIDIA Graphic Processing Unit (GPU) cards to run up to 600 times faster than in single-processor computers.

2.2 Installing HydroBID Flood

To install RiverFlow2D, please follow these steps.

1. When installing from a DVD, please insert HydroBID Flood installation disk into your DVD drive, double click on the setup file (`HydroBID_Flood_Setup.exe`) and follow the on-screen instructions. If you are installing from a setup file link, please download the file and double-click on it.
2. If you are installing on a PC running Windows 7 or later, you must be logged on the PC as an administrator before you begin the installation.
3. A separate document explains how to activate your software for the first time.

2.3 HydroBID Flood Documentation

Find HydroBID Flood documentation including this manual in the following folder:

\RiverFlow2D_QGIS\Documentation usually installed under ...\Documents or \My Documents (see Figure 2.1).

Also under \RiverFlow2D_QGIS, you will find example projects, videos, and other useful resources.

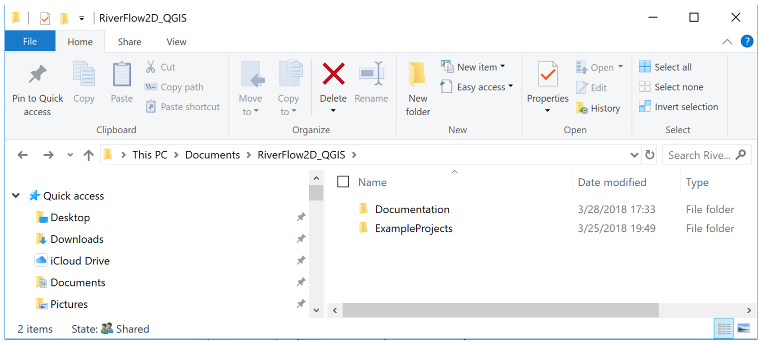


Figure 2.1 – Location of HydroBID Flood Documentation folder.

2.4 HydroBID Flood Technical Support

If you have any questions or require assistance using HydroBID Flood, please send an email to our support team at: contact@hydrobidlac.org. Please make sure you visit our web site www.hydrobidlac.org regularly to find out about new products and news about the software.

2.5 HydroBID Flood Tutorials

The best way to get acquainted and using HydroBID Flood capabilities is following the tutorials. In the accompanying HydroBID Flood there are tutorials to get started with HydroBID Flood, and for several of the model components. Each tutorial includes a set of files that you can use to do each exercise.

3 — Overview of HydroBID Flood

When you create a new HydroBID Flood 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 3.1. Description of each layer is included in Table 3.1.

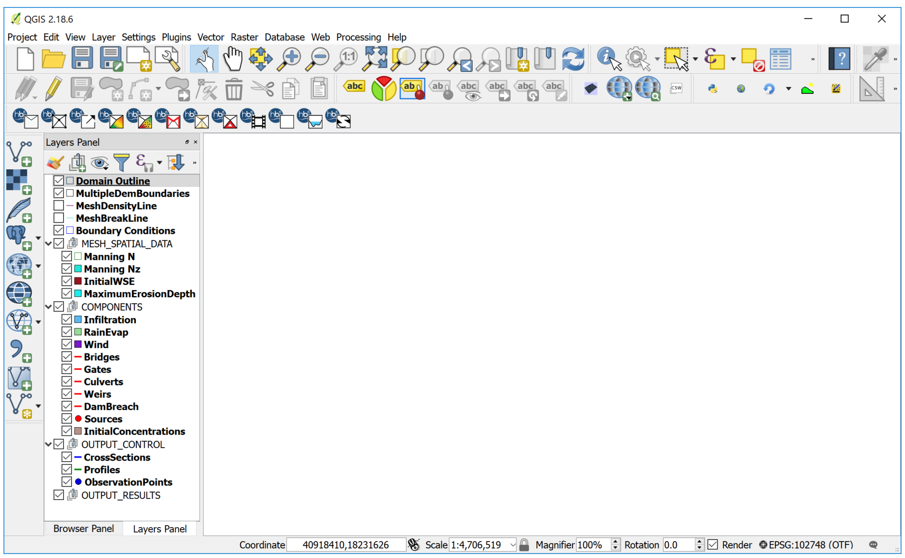
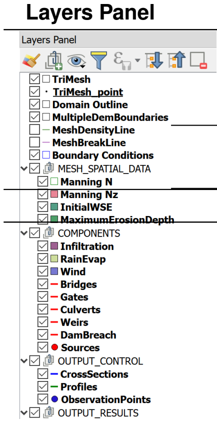


Figure 3.1 – Standard layers created when using the New HydroBID Flood Project command.

Table 3.1 – HydroBID Flood layers.

Layers Panel	Layer	Type	Content
	TriMesh	Polygon	Contains the mesh triangular cells. It is automatically created by the mesh generation program.
	TriMesh_points	Point	Contains the mesh nodes. It is automatically created by the mesh generation program.

Continued on next page

Table 3.1 – continued from previous page

Layers Panel	Layer	Type	Content
	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).
	MaximumErosionDepth	Polygon	Container for polygons with Maximum Erosion Depth (MED) attribute. When using the Sediment Transport ST module, the model will not allow erosion to reduce the bed elevation below the initial bed elevation minus MED.
	COMPONENTS		
	Infiltration	Polygon	Defines areas of different Infiltration parameters.
	RainEvap	Polygon	Container for areas associated with a rainfall intensity and evaporation.

Continued on next page

Table 3.1 – continued from previous page

Layers Panel	Layer	Type	Content
	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.
	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.
	OUTPUT_CONTROL		
	CrossSections	Line	Container for lines that define cross sections where the model will write results including discharge for each report interval.
	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 layers with model results that will be incorporated by the program when creating specific graphics with model results.

4 — Mesh Generation in HydroBID Flood

The basis of HydroBID Flood computational engine, RiverFlow2D 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 4.1).

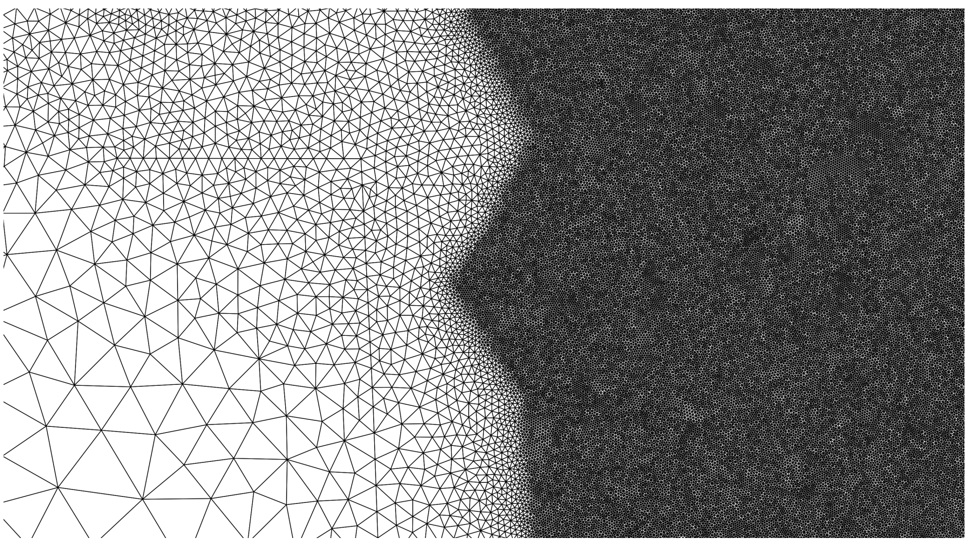


Figure 4.1 – Typical HydroBID Flood flexible mesh.

The fundamental computational unit in the RiverFlow2D model is the triangular cell, where velocities, depth, and other variables are computed.

There are several tools in HydroBID Flood that can be used to control the mesh generation. These tools make use of spatial objects and parameters that you can enter in the *Domain Outline*, *MeshDensityLine*, and *MeshBreakLine* layer.

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 4.2).

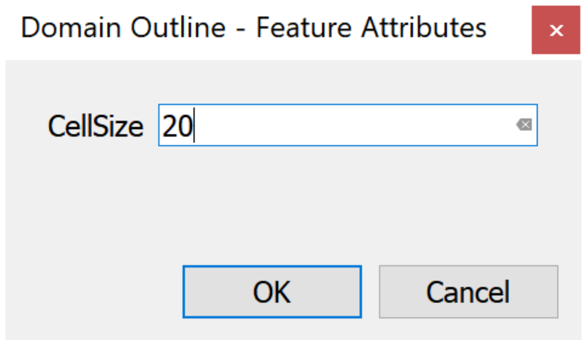


Figure 4.2 – *CellSize* dialog.

Figure 4.3 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.

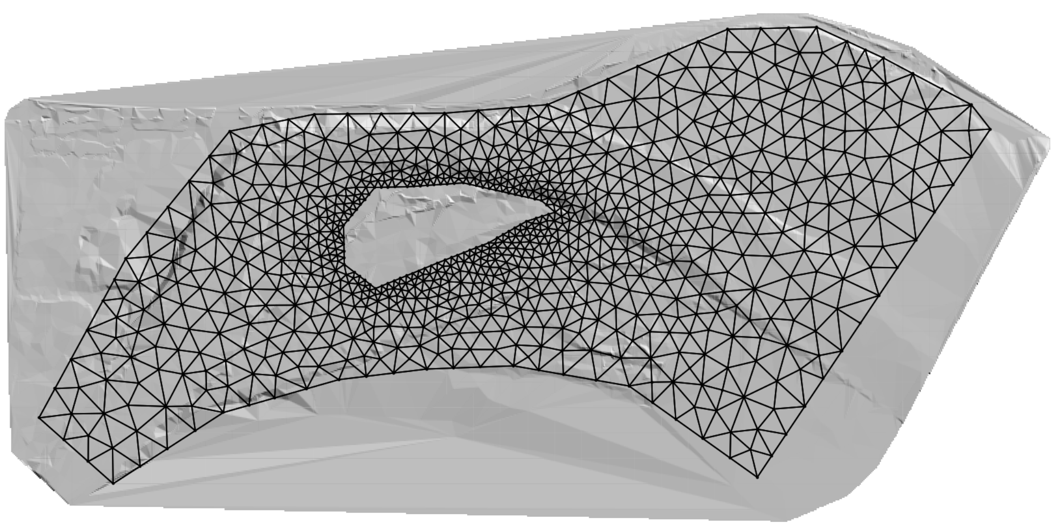


Figure 4.3 – Mesh generated based on an external polygon with *CellSize* = 50ft , and an internal polygon with *CellSize* = 10ft, both entered on the *Domain Outline* layer.

The *MeshDensityLine* layer 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 (see Figure 4.4).

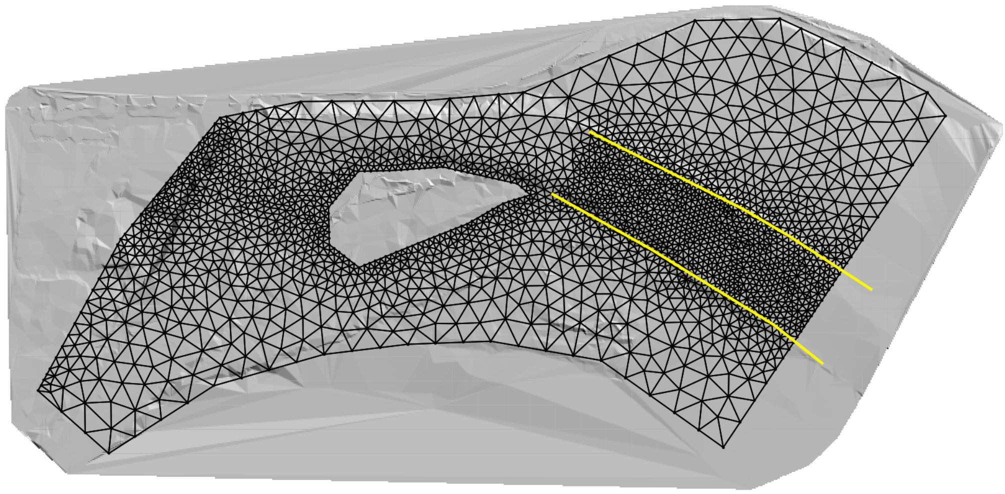


Figure 4.4 – Mesh generated based on the polygons of Figure 4.3 adding the two polylines on the *MeshDensityLine* layer.

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 *MeshDensity-Line* layer, but in this case the lines force the mesh generator to create nodes along the lines. Therefore, they act as hard breaklines (see Figure 4.5).

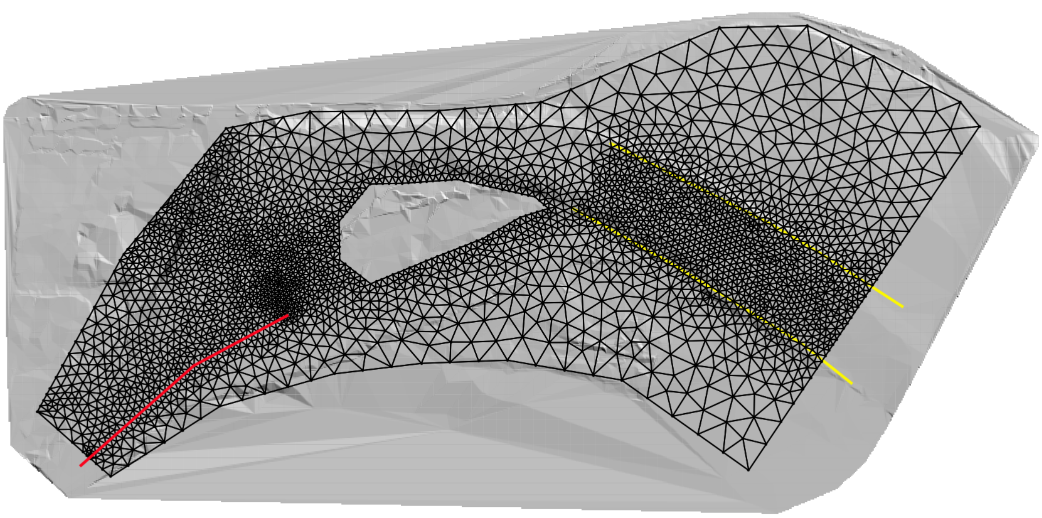


Figure 4.5 – Mesh generated based on the polygons and polylines of Figure 4.4 adding the one polylines on the *MeshBreakLine* layer. Note how unlike in the MeshDenstityLine layer, the polylines entered in the *MeshBreakLine* layer force the mesh to have nodes along the polylines.

In addition to the control offered by the spatial objects entered in the Domain Outline, *Mesh-DensityLine*, and *MeshBreakLine* layers, other layers can be used to adjust the mesh alignment and resolution. For instance, Bridges, Gates, and Weirs components are entered as polylines on the respective layers and all of them have a *CellSize* attribute and act as hard breaklines.

4.1 Boundary Conditions

Data to impose open boundary conditions in HydroBID Flood 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

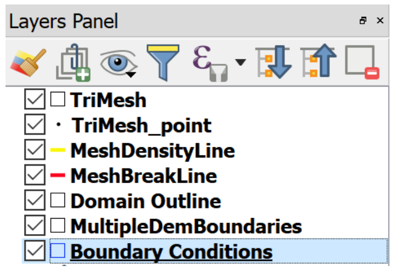


Figure 4.6 – QGIS Layer Panel showing the *Boundary Conditions* layer selected.

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

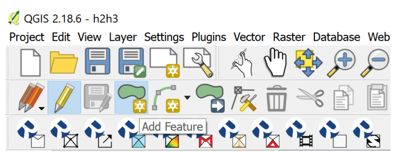


Figure 4.7 – Select Toggle Editing (pencil) and Add Feature.

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

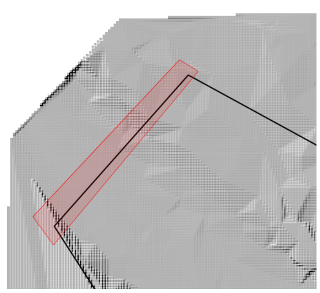


Figure 4.8 – Inflow Boundary Condition Polygon.

To complete entering the polygon, right click and the following dialog will appear where as an example we have selected the open boundary as Inflow, Discharge vs. time, and the data will be written to the `QInflow.dat`.

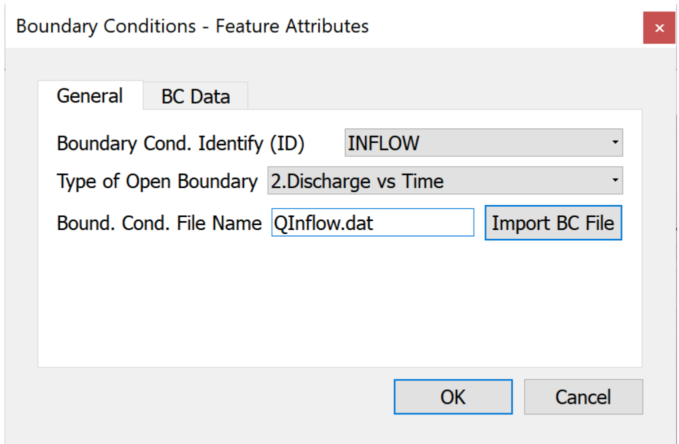


Figure 4.9 – Boundary Condition Dialog.

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

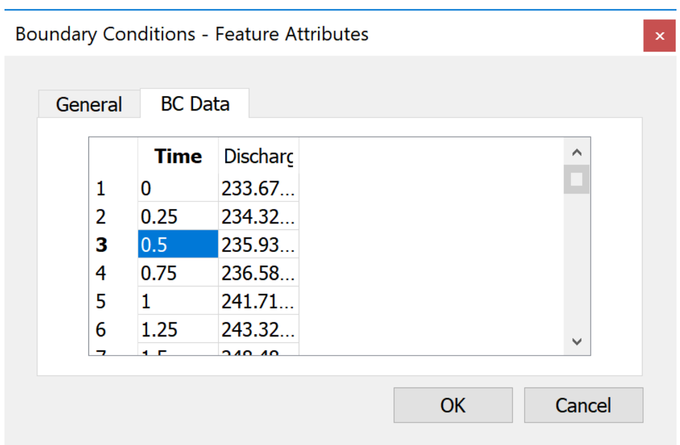


Figure 4.10 – Boundary Condition BC Data panel.

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.

4.2 Mesh Spatial Data

4.2.1 Mannings' n

To assign spatially varied Manning's n coefficients in HydroBID Flood 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

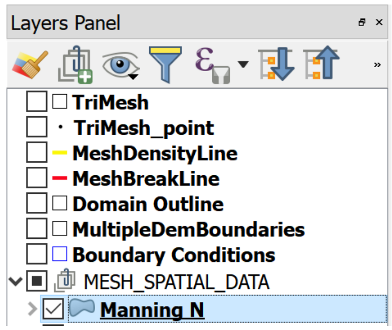


Figure 4.11 – QGIS Layer Panel showing the Manning N layer selected.

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

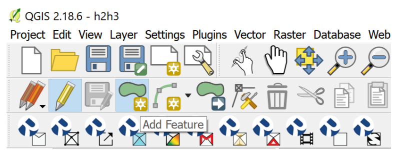


Figure 4.12 – Select Toggle Editing (pencil) and Add Feature.

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 the following dialog will appear where as an example we have selected the open boundary as Inflow, Discharge vs. time, and the data will be written to the `QInflow.dat`.

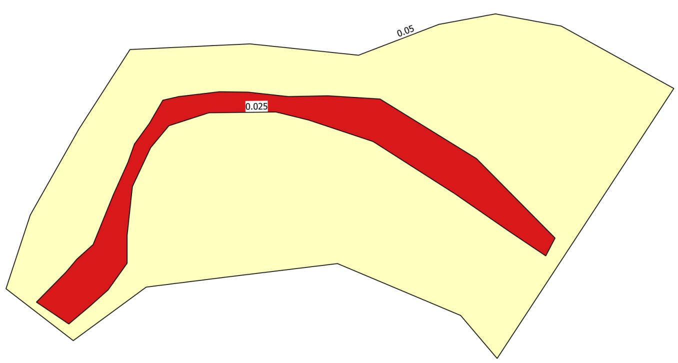


Figure 4.13 – Manning's n polygon.

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.

5 — RiverFlow2D Model Equations

5.1 Hydrodynamic Model

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

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

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

5.2 Assumptions of the Hydrodynamic Model

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

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

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

5.3 Unsteady Flow Model

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

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

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

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

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

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

where the bed slopes of the bottom level z_b are

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

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

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

with n the roughness coefficient.

5.4 Finite-Volume Numerical Solution

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

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

where $\mathbf{E} = (\mathbf{F}, \mathbf{G})$ and $\mathbf{n} = (n_x, n_y)$ is the outward unit normal vector to the volume Ω. In order to obtain a numerical solution of system (5.6) the domain is divided into computational cells, Ω_i, using a fixed mesh. Assuming a piecewise representation of the conserved variables

(Figure 5.1) and an upwind and unified formulation of fluxes and source terms (Murillo & García-Navarro 2010*b*)

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

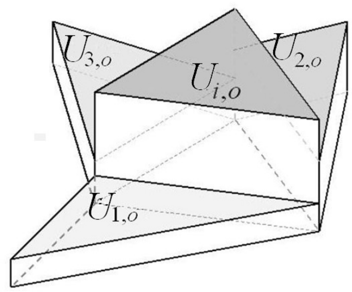


Figure 5.1 – Piecewise uniform representation of the flow variables.

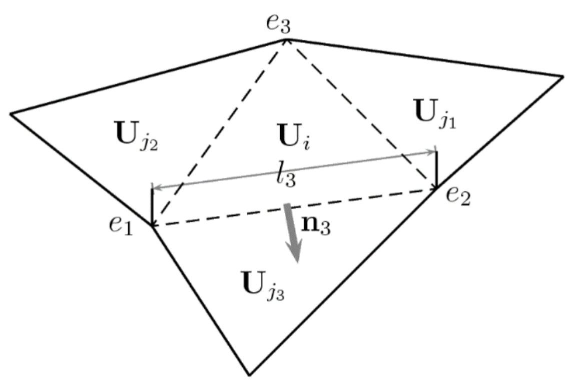


Figure 5.2 – Cell parameters.

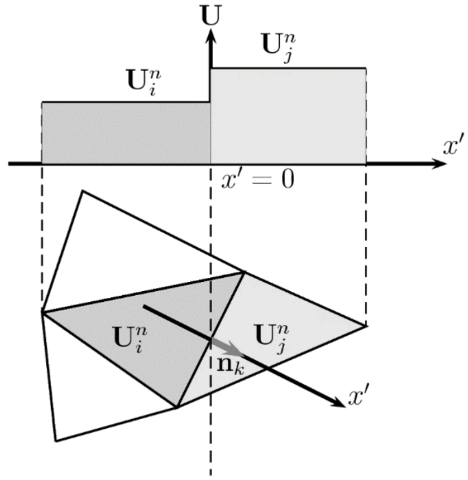


Figure 5.3 – Cell parameters.

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

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

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

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

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

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

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

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

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

Godunov method (Godunov 1959)

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

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

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

(5.12) can be rewritten as

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

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

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

with

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

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

5.4.1 Numerical Optimizations

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

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

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

5.4.2 Stability Region

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

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

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

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

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

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

5.5 Open Boundary Conditions

There are two main boundary condition types that can be used in HydroBID Flood: Open boundaries where flow can enter of leave the modeling area and closed boundaries that are solid no-flow walls (see Figure 5.4). There is no restriction on the number of inlet or outlet boundaries. This section describes the open boundary conditions.

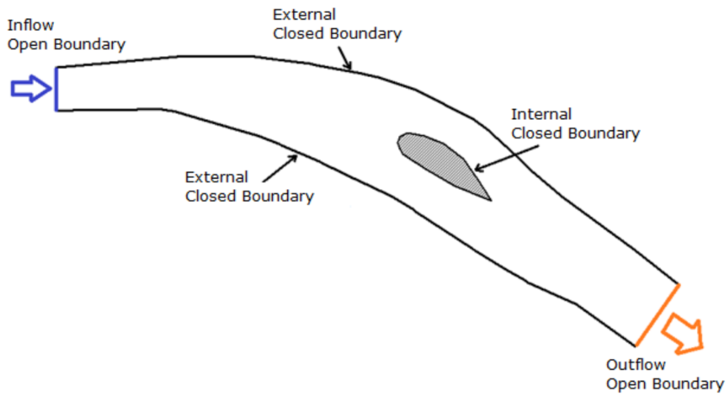


Figure 5.4 – Open and closed boundary conditions.

HydroBID Flood 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 HydroBID Flood simulation. Shallow water equation theory indicates that for two-dimensional subcritical flow it is required to provide at least one condition at inflow boundaries and one for outflow boundaries. For supercritical flow all conditions must be imposed on the inflow boundaries and no boundary condition should be imposed at outflow boundaries. The table below helps determining which conditions to use for most applications.

Table 5.1 – Required boundary conditions.

FLOW REGIME AT BOUNDARY	INFLOW BOUNDARY CONDITION	OUTFLOW BOUNDARY CONDITION
Subcritical	Q or Velocity	Water Surface Elevation
Supercritical	Q and Water Surface Elevation	None

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.

Table 5.2 – Open Boundary condition types.

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

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 5.5). Setting two or more open conditions without this separation will lead to incorrect detection of the open boundaries.

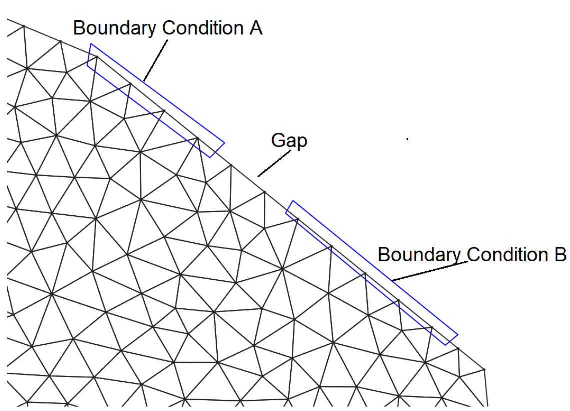


Figure 5.5 – Required gap between adjacent open boundary conditions.

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

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

5.5.1.1 Water Discharge Converted in Velocities (BCTYPE 6)

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

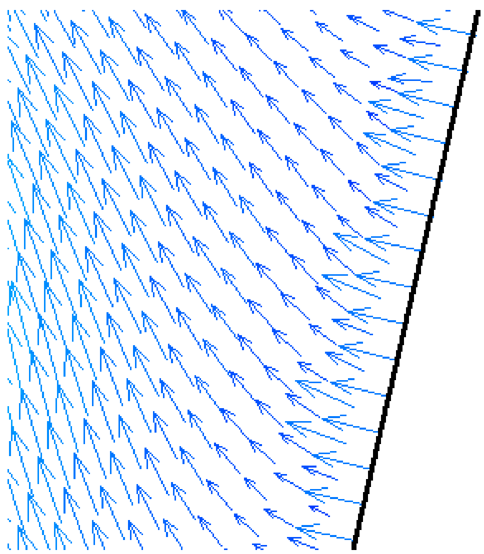


Figure 5.6 – Inflow water discharge imposed as velocities (BCTYPE 6).

5.5.2 Discharge Rating Table (BCTYPE 9)

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

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

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

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

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

5.5.4 Uniform Flow Boundary Condition (BCTYPE 12)

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

5.5.5 Numerical Implementation of Open Boundaries

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

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

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

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

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

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

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

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

The cells in the outlet discharge region are defined by

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

for supercritical flow, all the following eigenvalues are positive:

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

in consequence, no extra information is required.

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

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

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

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

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

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

5.5.5.1 Inlet discharge boundary

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

5.5.5.2 Simple cases

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

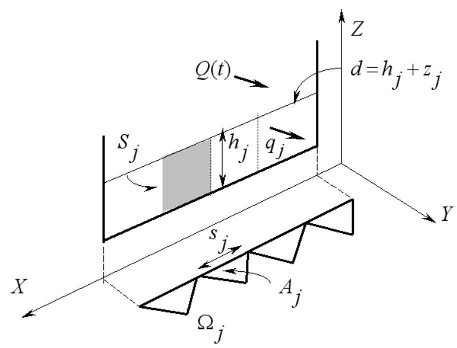


Figure 5.7 – Rectangular inlet cross section.

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

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

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

5.5.5.3 Complex cases

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

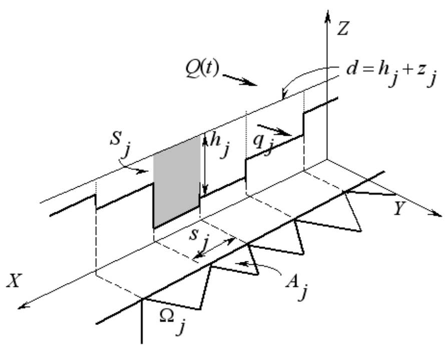


Figure 5.8 – Irregular inlet cross section.

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

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

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

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

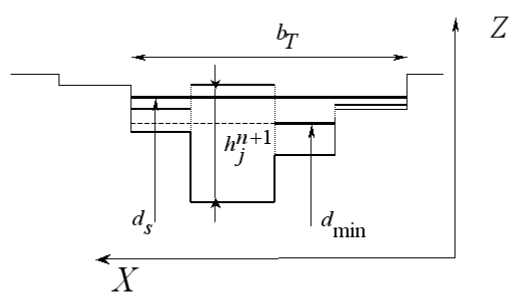


Figure 5.9 – Evaluation of d_{min}.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

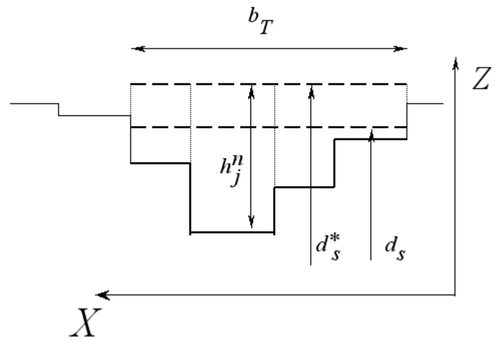


Figure 5.10 – New water level for the inlet section.

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

5.5.5.4 Outlet boundaries

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

5.5.6 Closed Boundaries

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

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

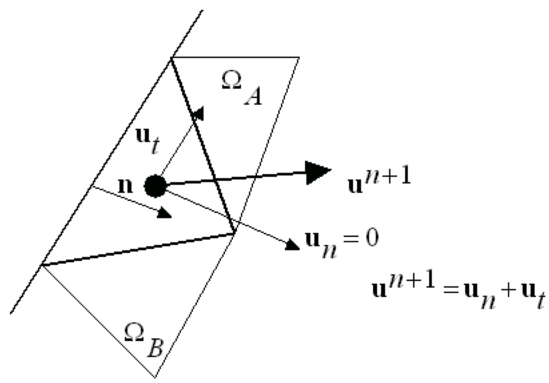


Figure 5.11 – Solid wall condition.

5.6 Dry/Wet Cell Modeling

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

5.6.1 Cell definitions Based on Dry and Wet Conditions

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

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

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

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

1. At the beginning of each time-step all cells are classified as wet or dry according to the definition.

2. If a cell is dry and completely surrounded by dry cells, it is removed from the computations and velocity components are set to zero for the ongoing time step.

3. All the internal cell edges are classified as active or inactive according to the definition.

4. Wet/dry cell edge contributions are computed assuming the edge is a solid boundary and the velocities on both sides are set to zero.

5. The rest of the cell edge contributions are computed according to the numerical scheme as described above.

6. Wet cells and dry cells surrounded by at least one wet cell are retained in the computation and solved with the updating scheme using the contributions from the cell edges.

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

5.7 Volume Conservation

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

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

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

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

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

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

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

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

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

with the total amount of water existing at time t

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

as follows

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

This is usually expressed in relative terms as follows:

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

5.8 Manning's n roughness Coefficients

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

5.9 Sediment Transport Model: RiverFlow2D ST Module

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

5.9.1 Model Equations

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

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

where

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

are the conserved variables with h representing the water depth, $q_x = hu$ and $q_y = hv$ the unit discharges, with (u, v) the depth averaged components of the velocity vector $\mathbf{u}$ along the x and y coordinates respectively and ϕ_p, with $p = 1, ..., N_p$ representing the scalar depth-averaged concentration of the N_p different sediments transported in suspension. The term z is the bed level and p_p is the bed porosity of each p bed sediment. The sum of $\sum_{p=1}^{N_p}\left[z_p(1-p_p)\right]$ is motivated for the possibility of a heterogeneous soil, where different fractions of material may coexist.

On the other hand the fluxes are given by

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

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

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

with $p_{b,x}, p_{b,y}$ and $\tau_{b,x}, \tau_{b,y}$ are the pressure force along the bottom and the bed shear stress in the

x and y direction respectively, with ρ_w the water density. The former can be formulated in terms of the bed slopes of the bottom level z

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

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

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

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

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

and is evaluated as

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

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

5.9.2 Sediment Transport Laws

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

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

5.9.2.1 Bed Load

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

Mass conservation

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

Momentum conservation in x direction

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

Momentum conservation in y direction

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

Bed elevation changes

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

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

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

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

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

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

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

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

Table 5.3 – Summary of bed load transport formulas.

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

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

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

5.9.3 Boundary Conditions for the Bed Load Sediment Model

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

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

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

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

5.9.4 Suspended Load

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

Mass conservation

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

Momentum conservation in x direction

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

Momentum conservation in y direction

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

Sediment mass of the fluid layer for the suspended sediment p

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

The term ϕ_{*_p} defines the equilibrium volume concentration, which is obtained through a solid transport discharge law. The term ϕ_p contains the information about the suspension sediment quantity which is transported. Both concentrations take in consideration the presence of the settling velocity, ω_{s_p}. The settling velocity can be computed through several laws as a function of the sediment density and the grain diameter.

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

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

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

Table 5.4 – Formulas to calculate sediment settling velocity ω_s.

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

5.9.4.1 Suspended Load Formulas

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

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

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

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

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

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

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

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

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

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

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

  with

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

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

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

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

with

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

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

5.9.5 Boundary Conditions for the Suspended Sediment Model

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

3

0 1000. 0.001 0.002

2 67000. 0.001 0.002

100 67000. 0.001 0.002

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

5.9.6 Geomorphological Collapse

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

5.9.7 Sediment Transport Finite-Volume Numerical Solution

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

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

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

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

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

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

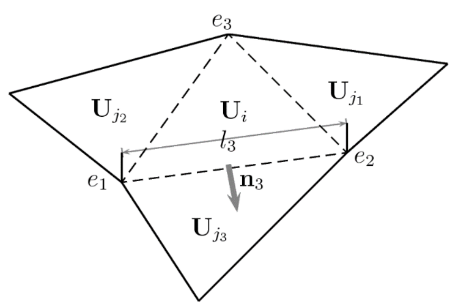


Figure 5.12 – Cell parameters.

Assuming a piecewise representation of the conserved variables

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

Equation (5.69) is written as

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

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

5.9.8 Entering Data for the Sediment Transport Model

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

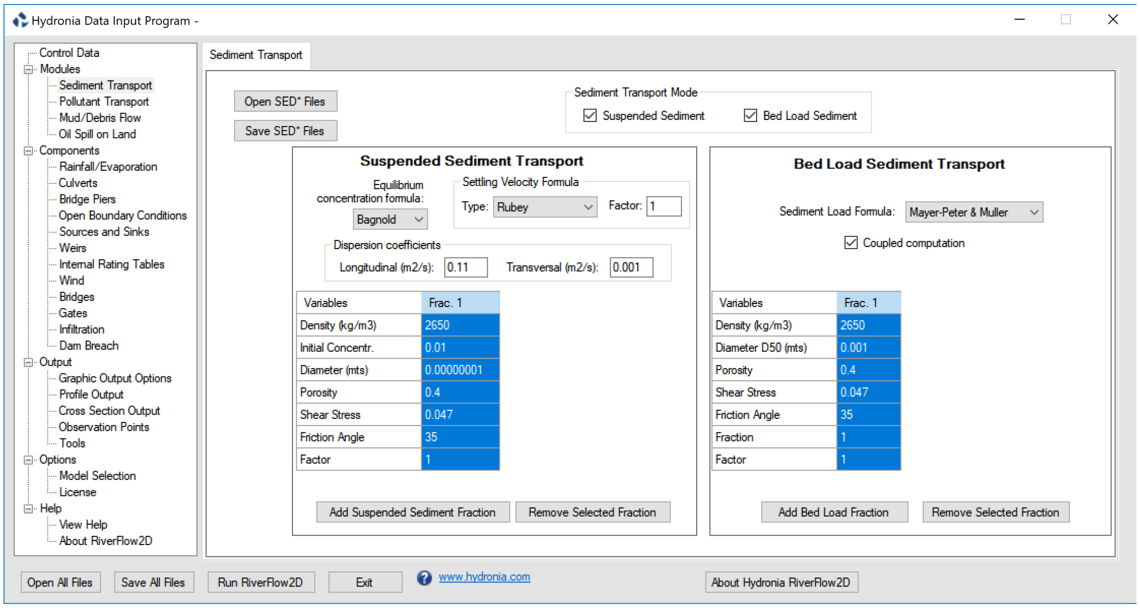


Figure 5.13 – Hydronia Data Input Program Sediment Transport panel.

5.9.9 Assumptions of the Sediment Transport Model

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

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

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

5.10 Mud and Debris Flow Model: RiverFlow2D MD module

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

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

5.10.1 Assumptions of the Mud and Debris Flow Model

The model involves the following assumptions:

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

5.10.2 Rheological Formulation

The single-phase rheological formulation in RiverFlow2D MD accounts for different friction terms that represent a variety of hyperconcentrated non-Newtonian fluids (see Figure 5.14).

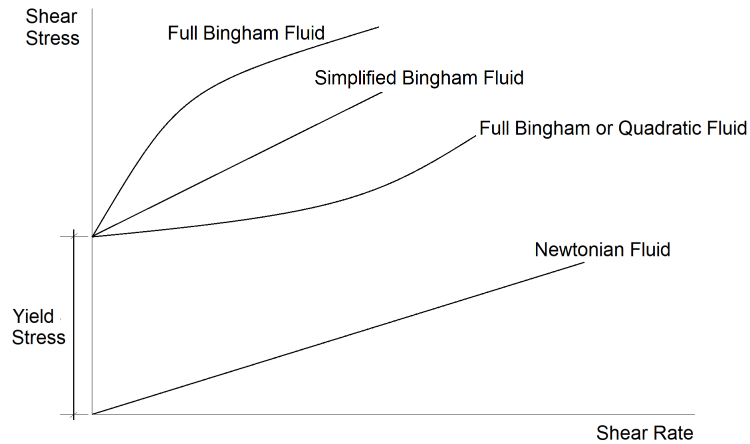


Figure 5.14 – Rheological diagram of some common non-Newtonian fluids.

The friction term in equation (5.1) is depth-averaged. Hence, the equations that describe the tangential forces generated by the stresses can be lumped into the same mathematical formula despite of having a different nature. In this way, several shear stresses can be considered to cover a wide range of mud and debris flow conditions, including:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Table 5.5 – Rheological flow resistance formulations used in HydroBID Flood MD.

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

5.10.3 Entering Data for the Mud and Debris Flow Model

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

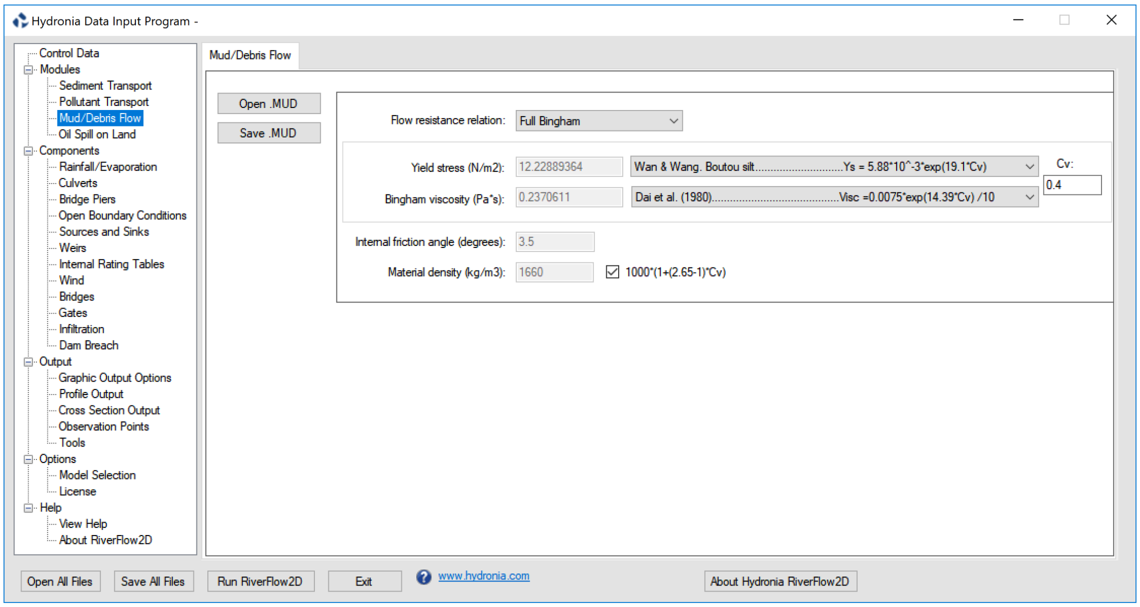


Figure 5.15 – Hydronia Data Input Program Mud/Debris Flow panel.

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

In addition to having the possibility to enter any of the fluid parameters from know data, the *Mud/Debris Flow* panel provides functions to determine yield stress, and viscosity based on formulas recommended in a number of publications. Figure 5.16 presents the formulas available to set the yield stress as a function of the fluid volumetric concentration C_v. Also Figure 5.17 lists the formulas provided to determine the fluid viscosity from the given C_v.

Wan & Wang. Boutou silt.............................Ys = 5.88*10^-3*exp(19.1*Cv)
Wan & Wang. Tangu silt I and II..................Ys = 5.88*10^-3*exp(32.1*Cv)
Wan & Wang. Zhengzhou loess..................Ys = 25.78*10^2**Cv^5.4
Wan & Wang. Zhengzhou loess..................Ys = 98.9810^2**Cv^5.4
Wan & Wang. Kaoline bentonite.................Ys = 1.28*10^3*Cv
Julien (1998) for sands...................................Ys =0.1*exp(3*(Cv-0.05))
Julien (1998) for 95% silt and 5% clays........Ys =0.1*exp(13*(Cv-0.05))
Julien (1998) for 70% silts and 30% clays.....Ys =0.1*exp(23*(Cv-0.05))
O'Brien (1988). Aspen pit...............................Ys = 0.181*exp(25.7*Cv) /10
O'Brien (1988). Aspen pit 4............................Ys = 2.72*exp(10.4*Cv) /10
O'Brien (1988). Aspen natural soil................Ys = 0.152*exp(18.7*Cv) /10
O'Brien (1988). Aspen mine fill.....................Ys = 0.0473*exp(21.1*Cv) /10
O'Brien (1988). Aspen natural soil source....Ys = 0.0383*exp(19.6*Cv) /10
O'Brien (1988). Aspen mine fill source.........Ys = 0.291*exp(14.3*Cv) /10
O'Brien (1988). Glenwood 1...........................Ys = 0.0345*exp(20.1*Cv) /10
O'Brien (1988). Glenwood 2...........................Ys = 0.0765*exp(16.9*Cv) /10
O'Brien (1988). Glenwood 3...........................Ys = 0.000707*exp(29.8*Cv) /10
O'Brien (1988). Glenwood 4..........................Ys =0.00172*exp(29.5*Cv) /10
Dai et al. (1980)..Ys = 2.6*exp(17.48*Cv) /10
Kang and Zhang (1980)..................................Ys = 1.75*exp(7.82*Cv) /10
Quian et al. (1980)..Ys = 0.00136*exp(21.2*Cv) /10
Chien and Ma (1958)......................................Ys = 0.0588*exp(19.1*Cv) /10
Fei (1981)..Ys = 0.166*exp(25.6*Cv) /10

Figure 5.16 – Yield stress formulas as a function of volumetric concentration C_v

Julien (1998) for sands..............................Visc =0.001*(1+2.5*Cv+exp(10*(Cv-0.05)))
Julien (1998) for silts and clays................Visc =0.001*(1+2.5*Cv+exp(23*(Cv-0.05)))
O'Brien (1988) Aspen pit 1........................Visc = 0.036*exp(22.1*Cv) /10
O'Brien (1988) Aspen pit 4........................Visc = 0.0538*exp(14.5*Cv) /10
O'Brien (1988) Aspen natural soil..............Visc = 0.00136*exp(28.4*Cv) /10
O'Brien (1988) Aspen mine fill....................Visc = 0.128*exp(12*Cv) /10
O'Brien (1988) Aspen natural soil source...Visc = 0.000495*exp(27.1*Cv) /10
O'Brien (1988) Aspen mine fill source........Visc = 0.000201*exp(33.1*Cv) /10
O'Brien (1988) Glenwood 1........................Visc = 0.00283*exp(23*Cv) /10
O'Brien (1988) Glenwood 2........................Visc = 0.648*exp(6.2*Cv) /10
O'Brien (1988) Glenwood 3........................Visc = 0.00632*exp(19.9*Cv) /10
O'Brien (1988) Glenwood 4........................Visc =0.000602*exp(33.1*Cv) /10
Iida (1938)..Visc =0.0000373*exp(36.6*Cv) /10
Dai et al. (1980)...Visc =0.0075*exp(14.39*Cv) /10
Kang and Zhang (1980)............................Visc =0.0405*exp(8029*Cv) /10

Figure 5.17 – Viscosity formulas as a function of volumetric concentration C_v

The program also calculates the fluid density ρ as a function of C_v and the water density ρ_w using the following formula:

$$\rho = \rho_w[(1 + (1.65)C_v] \tag{5.79}$$

5.11 Pollutant Transport Model: HydroBID Flood PL Module

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

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

5.11.1 Model Equations

Although HydroBID Flood PL can handle multiple pollutants simultaneously, for the sake of clarity in this section the transport of only one solute is presented coupled to the 2D model. The solute transport equations will be expressed in a conservative form, assuming that the velocities and the water depth may not vary smoothly in space and time.

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

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{H}(\mathbf{U}) + \mathbf{R}(\mathbf{U}) + \mathbf{D}(\mathbf{U}) \tag{5.80}$$

where

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

$$\mathbf{F} = \left(q_x, \tfrac{q_x 2}{h} + \tfrac{1}{2}gh2, \tfrac{q_x q_y}{h}, h\phi u\right)^T, \qquad \mathbf{G} = \left(q_y,\ \tfrac{q_x q_y}{h},\ \tfrac{q_y 2}{h} + \tfrac{1}{2}gh2,\ h\phi v\right)^T \tag{5.81}$$

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

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

$$\mathbf{R} = (0, 0, 0, -Kh\phi)^T \qquad \mathbf{D} = \left(0, 0, 0, \vec{\nabla}(Dh\vec{\nabla}\phi)\right)^T \tag{5.82}$$

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

5.11.2 Pollutant Transport Finite-Volume Numerical Solution

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

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

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

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

$$q_k^{\downarrow} = q_i + \sum_{m=1}^{3} \left(\widetilde{\lambda}^{-} \, \widetilde{\gamma} \, \widetilde{\mathbf{e}}_1 \right)_k^m \tag{5.83}$$

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

$$(h\phi)_i^{n+1} = (h\phi)_i^n - \frac{\Delta t}{A_i} \sum_{k=1}^{N_E} (q\phi)_k^{\downarrow} l_k \tag{5.84}$$

where

$$\phi_k^{\downarrow} = \begin{cases} \phi_i & if \quad q_k^{\downarrow} > 0 \\ \phi_j & if \quad q_k^{\downarrow} < 0 \end{cases} \tag{5.85}$$

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

From a physical point of view, the new solute mass at a fixed cell can be seen as exchanging water volumes with certain concentration through the neighboring walls and mixing them (finite-volume Godunov's type method) with the former mass existing in the previous time (Figure 5.18).

According to this philosophy, the outlet boundary cells will require a special treatment when applying this technique in order to extract the corresponding solute mass through the boundary walls. For this reason, it is necessary to define $q^{\downarrow} = (h\mathbf{u}\cdot\mathbf{n})_{BC}$ and $\phi^{\downarrow} = \phi_{BC}$ at the boundary wall and to include this contribution for the updating of the boundary cell BC (see Figure 5.19).

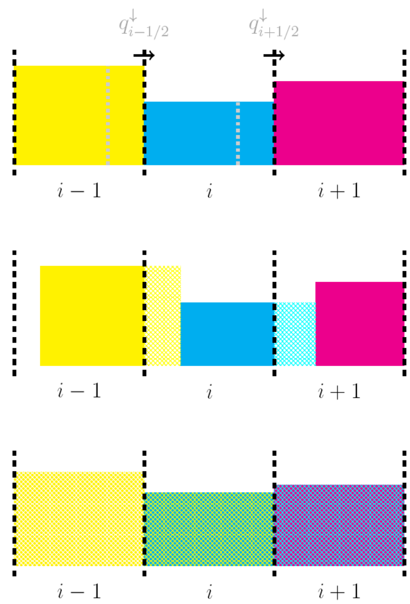


Figure 5.18 – Physical representation of solute mass exchange between cells with $q^{\downarrow}_{i-1/2}, q^{\downarrow}_{i+1/2} > 0$.

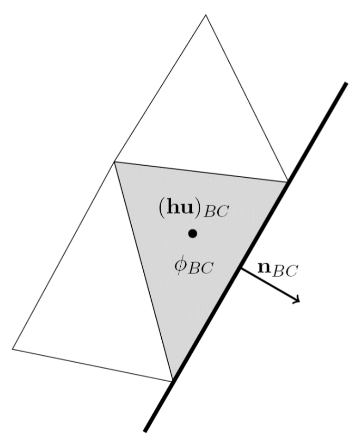


Figure 5.19 – Extraction of mass solute in an outlet boundary cell.

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.

5.11.3 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.

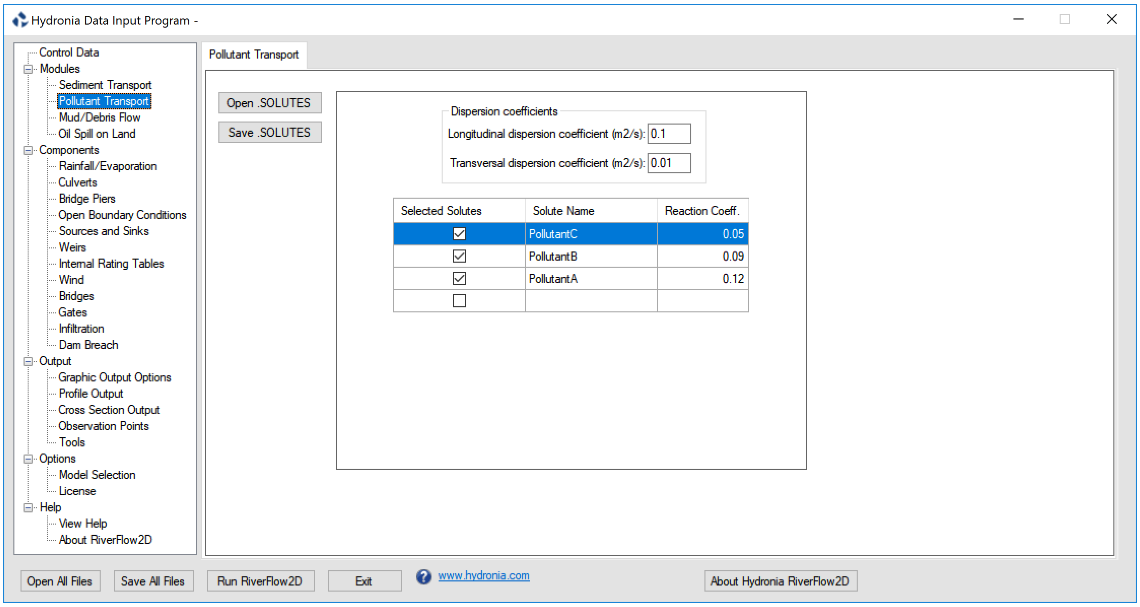


Figure 5.20 – Hydronia Data Input Program *Pollutant Transport* panel.

5.11.4 Assumptions of the Pollutant Transport Model

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

1. There is no predetermined limit to the number of pollutants.

2. Interaction between solutes and between each solute and water are assumed to be first order reactions.

3. All inflow boundaries where either discharge or water elevation is imposed must provide a concentration time series for each pollutant.

5.12 Water Quality Model: WQ Module

The configurations of this module follow the structure given in the table 5.6:

Table 5.6 – State variables used to simulate each option in the quality module

	Option: 1	2	3	4
State variable				
Ammonium nitrogen $(NH_4^+ - N)$			✔	✔
Nitrate Nitrogen $(NO_3^- - N)$			✔	✔
Inoganic phosphorus (IP)				✔
Phytoplankton carbon $(PHYT)$				✔
Carbonaceous biological oxygen demand $(CBOD)$	✔	✔	✔	✔
Dissolved oxygen (DO)	✔	✔	✔	✔
Organic nitrogen (ON)			✔	✔
Organic phosphorus (OP)				✔
Temperature (T)		✔		✔
Total coliform bacteria (TC)				✔

5.12.1 Hydrodynamic equations and convection-diffusion-reaction equation

The flow of water with a free surface can be described by using equations that conserve mass and momentum:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial y} = \mathbf{H}(\mathbf{U}) \tag{5.86}$$

with:

$$\begin{aligned}
\mathbf{U} &= (h, q_x, q_y)^T \\
\mathbf{F} &= \left(q_x, \frac{q^2x}{h} + \frac{1}{2}gh^2, \frac{q_x q_y}{h}\right)^T \\
\mathbf{G} &= \left(q_y, \frac{q_x q_y}{h}, \frac{q^2y}{h} + \frac{1}{2}gh^2\right)^T \\
\mathbf{H} &= (0, gh(S_{0x} - S_{fx}), gh(S_{0y} - S_{fy}))^T
\end{aligned} \tag{5.87}$$

being **U** $=$ Conserved variables

F,G $=$ Fluxes

H $=$ Source terms

$q_x = uh$

$q_y = vh$

$(u, v) =$ Average components of velocity vector **u** along the x and y

$h =$ depth

While for the transport equation it is:

$$\frac{\partial(h\phi_i)}{\partial t} + \frac{\partial(hu\phi_i)}{\partial x} + \frac{\partial(hv\phi_i)}{\partial y} = E\frac{\partial}{\partial x}\left(h\frac{\partial\phi_i}{\partial x}\right) + E\frac{\partial}{\partial y}\left(h\frac{\partial\phi_i}{\partial y}\right) \pm hR_i \pm f_i \tag{5.88}$$

where ϕ_i is the average concentration of each state variable, E longitudinal diffusion coefficient, f_i point and non-point sources, y R_i represents the formation or consumption of each constituent.

The term R_i is established according to the Petersen matrix. The matrix is composed of processes (rows) and state variables (columns), with elements within the matrix that include stoichiometric coefficients that establish the relationships between the components in the individual processes. The general matrix to simulate the options in the table 5.6, it will be defined according to the tables 5.7, 5.8, 5.9

Variable $i \rightarrow$ Process $j \downarrow$	ϕ_1 NH_4^+	ϕ_2 NO_3^-	ϕ_3 IP	ϕ_4 PHYT	ϕ_5 CBOD	ϕ_6 DO	ϕ_7 ON	ϕ_8 OP	ϕ_9 T	ϕ_{10} TC	Process rate
1. CBOD and oxygen Reaeration						1					$k_a \theta_2^{T-20} (\phi_{sat} - \phi_6)$
2. C-oxidation					-1	-1					$k_D \theta_D^{T-20} \left(\frac{\phi_6}{k_{BOD}+\phi_6} \right) \phi_5$
3. C-settling					-1						$\frac{v_{s3}(1-f_{d5})}{h} \phi_5$
4. Sediment O_2 demand						-1					$\frac{SOD}{h} \theta_S^{T-20}$
5. OP- mineralization			1					-1			$k_{83} \theta_{83}^{T-20} \left(\frac{\phi_4}{k_{PHY}+\phi_4} \right) \phi_8$
6. OP-settling								-1			$\frac{v_{s3}(1-f_{d8})}{h} \phi_8$
7. ON- mineralization	1						-1				$k_{71} \theta_{71}^{T-20} \left(\frac{\phi_4}{k_{PHY}+\phi_4} \right) \phi_7$
8. ON-settling							-1				$\frac{v_{s3}(1-f_{d7})}{h} \phi_7$
9. Nitrification (low O_2 limitation)	-1	1				-$\frac{64}{12}$					$k_{12} \theta_{12}^{T-20} \left(\frac{\phi_6}{k_{NIT}+\phi_6} \right) \phi_1$
10. Denitrification (high O_2 limitation)		-1			-$\frac{5}{4}\frac{32}{14}$						$k_{2D} \theta_{2D}^{T-20} \left(\frac{k_{NO_3}}{k_{NO_3}+\phi_6} \right) \phi_2$
11. Phyt. grow (Phosphorus limtation)			$-a_{PC}$	1		b^a					$G_{P1} \phi_4$
12. Phyt. resp. (Phosphorus limtation)				-1		$-\frac{32}{12}$		a_{PC}			$k_{1R} \theta_{1R}^{T-20} \phi_4$
13. Phyt. death (Phosphorus limtation)			$a_{PC}(1-f_{OP})$	-1	a_{OC}			$a_{PC} f_{OP}$			$k_{1D} \theta_{1R}^{T-20} \phi_4$
14. Phyt. settling (Phosphorus limtation)				-1							$\frac{v_{S4}}{h} \phi_4$
15. Phyt. grow (Nitrogen limtation)	$-a_{NC} P_{NH_4}$	$-a_{NC}(1 - P_{NH_4})$		1	a_{OC}	b^a					$G_{P1} \phi_4$
16. Phyt. resp. (Nitrogen limtation)				-1		$-\frac{32}{12}$	a_{NC}				$k_{1R} \theta_{1R}^{T-20} \phi_4$
17. Phyt. death (Nitrogen limtation)	$a_{NC}(1 - f_{ON})$			-1			$a_{NC} f_{ON}$				$k_{1D} \theta_{1R}^{T-20} \phi_4$
18. Phyt. settling (Nitrogen limtation)				-1							$\frac{vs4}{h} \phi_4$
19. Heat budgets									1		$\frac{K_{ch}(T_e - \phi_9)}{\rho c_p h}$
20. Total coliform bacteria										1	$k_{TC} \phi_{10}$

Table 5.7 – Peter matrix of processes

Ammonium preference factor(P_{NH_4}):
$P_{NH_4} = \left(\frac{\phi_2}{(k_{mN} + \phi_1)(k_{mN} + \phi_2)} \right) \phi_1$

Phytoplankton growth rate. (G_{p1}):
$G_{p1} = k_{1c} X_{RT} X_{RI} X_{RN}$

Phytoplancton temperature adjustment:
$X_{RT} = \theta_{1C}^{T-20}$

Phytoplankton light limitation.(X_{RI}):
$X_{RI} = \frac{e}{k_e h} f \left[exp \left\{ -\frac{I_0}{I_S} exp\left(-k_e h\right) \right\} - exp \left(\frac{I_0}{I_S} \right) \right]$

phytoplankton nutrient limitation.(X_{RN}):
$X_{RN} = min \left(\frac{DIN}{k_{mN} + DIN}, \frac{DIP}{k_{mP} + DIP} \right)$

Total coliform:
$k_{TC} = -\left(0.8 + 0.02S\right) 1.07^{\phi_9 - 20} + \frac{\alpha I_0/24}{k_e h} \left(1 - e^{-k_e h}\right) - F_p \frac{v_{s_{10}}}{h}$

Re-aeration:
$^{*}k_a = 3.93 \frac{u^{0.5}}{h^{1.5}}$

Saturation oxygen:
$^{**}\phi_{sat} = 0.0035\phi_9^2 - 0.3369\phi_9 + 14.407$

Five day BOD:
$DBO_5 = \phi_5 (1 - e^{-5k_{dbot}})$

Dissolved inorganic phosphorus:
DIN $= \phi_1 + \phi_2$

Dissolved inorganic nitrogen:
DIP= ϕ_3

Equilibrium river temperature:
$T_e = T_d + \frac{I_0}{K_{ch}}$

Constant:
$b^a = \frac{32}{12} + \frac{48}{14} \frac{14}{12} (1 - P_{NH4})$

Table 5.8 – Addtional equations

Parameters	Expression
Heat exchange coefficient K_{ch}	$K_{ch} = 4.5 + 0.05\phi_9 + \beta f(U_w) + 0.47 f(U_w)$
Wind function $f(U_w)$	$f(U_w) = 9.2 + 0.46U_w^2$
Coefficient β	$\beta = 0.35 + 0.015T_v + 0.0012T_v^2$
Average temperature T_v	$T_v = (\phi_9 + T_d)/2$
Dew point temperature T_d	$T_d = 237.3[T_a^* + ln(r_h)]/[17.27 - ln(r_h) - T_a^*]$
	$T_a^* = 17.27T_a/(237.3 + T_a)$

Table 5.9 – Empirical formulas for computing surface heat exchange coefficient

Description	Parameter	Recommend
Oxygen to carbon ratio	a_{OC}	2.76
Nitrogen to carbon ratio	a_{NC}	0.25
Phosphorus to carbon ratio	a_{PC}	0.025
Half-saturation for O_2 limitation on de-oxygenation	k_{BOD}	0.5
Carbonaceous de-oxygenation rate at 20 °C	k_D	0.16-0.21
Half-saturation for N-limitation on phyto. uptake	k_{mN}	0.025
Half-saturation for P-limitation on phyto. uptake	k_{mP}	0.001
Half-saturation for O_2 limitation on nitrification	k_{NIT}	2.0
Nitrification rate at 20 °C	k_{12}	0.09-0.13
Phytoplankton growth rate	k_{1C}	2
Phytoplankton death rate	k_{1D}	0.02
Phytoplankton respiration rate	k_{1R}	0.125
Denitrification rate at 20°C	k_{2D}	0.09
ON-mineralization rate at 20°C	k_{71}	0.075
OP-mineralization rate at 20°C	k_{83}	0.22
Half-saturation for O_2 limitation on denitrification	k_{NO_3}	0.1
Half-saturation for phyto. limit. on mineralization	k_{PHY}	0-1
Light extinction coefficient in water column	k_e	0.1-5
The laboratory ”bottle” deoxygenation rate cosntant	k_{dbot}	0.2-0.5
Settling velocity of organic matter	v_{S3}	-
Settling velocity of phytoplankton	v_{S4}	0.1
Settling velocity of TC	v_{10}	0.4
Saturated light intensity	I_S	200-500
Surface light energy	I_0	-
BOD temperature coefficient	θ_D	1.047
Re-aeration temperature coefficient	θ_2	1.024-1.028
Nitrification temperature coefficient	θ_{12}	1.08
Denitrification temperature coefficient	θ_{2D}	1.08
Phytoplankton grow temperature coefficient	θ_{1C}	1.066
Phytoplankton respiration temperature coefficient	θ_{1R}	1.045
ON-mineraliztion temperature coefficient	θ_{71}	1.08
OP-mineralization temperature coefficient	θ_{83}	1.08
SOD temperature coefficient	θ_S	1.08
Sediment oxygen demand	SOD	0.2-4
Photoperiod (fraction of day)	f	0.5
Fraction of the bacteria	F_p	0.871
Fraction of dead phytoplankton recycled to ON	f_{ON}	0.5
Fraction of dead phytoplankton recycled to OP	f_{OP}	0.5
Fraction dissolved CBOD	f_{d5}	0.5
Fraction dissolved ON	f_{d7}	0.5
Fraction dissolved OP	f_{d8}	0.5
Saturation oxygen	ϕ_{sat}	**
Salinity	S	-
A proportionality	α	-
Wind velocity	U_w	-
Air temperature	T_a	-
Relative humidity	r_h	-

Table 5.10 – Description of all parameters used WQ

6 — Code Parallelization

6.1 RiverFlow2D CPU

RiverFlow2D 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 RiverFlow2D dynamically distributes the computational workload between as many processors or cores as are available. In this way the model optimizes its computations to the particular architecture of each computer.

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

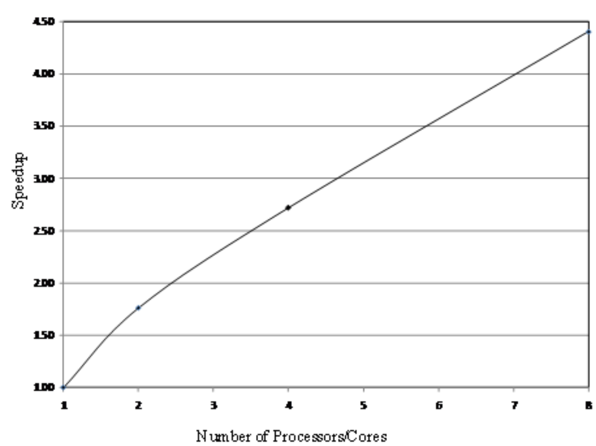


Figure 6.1 – Speed up using RiverFlow2D parallelized code as a function of number of processor cores.

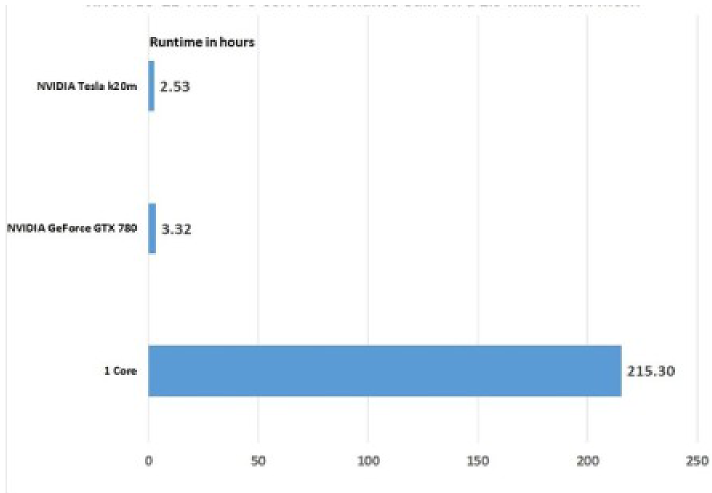


Figure 6.2 – Speed up using RiverFlow2D parallelized code as a function of number of processor cores.

6.2 RiverFlow2D GPU

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

7 — Hydraulic and Hydrologic Components

HydroBID Flood 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:

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

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

- **Bridges:** account for general geometry bridges including pressure flow and overtopping.
- **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.

7.1 Bridges Component

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

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

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

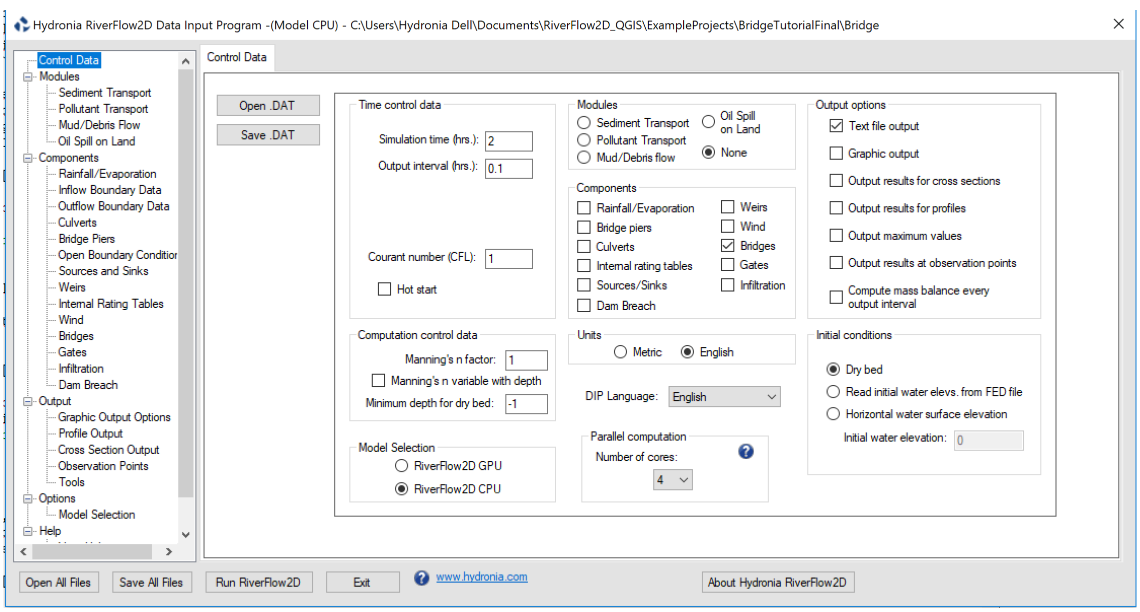


Figure 7.1 – Hydronia Data Input Program *Control Data* panel with the Bridges component selected.

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

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

7.1.1 Bridge Geometry Data File

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

Line 1: Number of points defining polylines.

NP

NP lines with these entries:

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

...

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

The relationship between the four polylines must be as follows:

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

7.1.1.1 Example of the Bridge Cross Section File

The following table is an example one of the geometry file that schematically represents the bridge in 7.2.

NP ⇨ 23

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

Figure 7.2 – Front view of a bridge cross section.

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

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BEDELEV	R	-	m or ft	Bed elevation. Must be the lowest elevation for all polylines at a given point.
DECKELEV	R	-	m or ft	Elevation of the bridge deck. Must be the highest elevation for all polylines at a given point.
LOWCHORD	R	-	m or ft	Elevation of the lower bridge deck. LOWCHORD must be larger or equal to ZLOWER and smaller or equal to DECKELEV for a particular point. The space between LOWCHORD and DECELEV is 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.

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

7.1.2 Bridge Calculations

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

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

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

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

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

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

and is used to represent bridges.

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

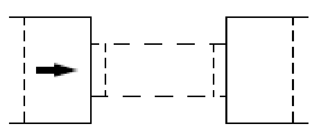


Figure 7.3 – Top view of a bridge showing the cross sections of interest. Only two piers are depicted for simplicity.

7.1.2.1 Energy dissipation in bridges

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

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

where ΔH_c and ΔH_e are the contraction and expansion losses respectively

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

where m is a typical value for the contraction coefficient, $m = 0.62$ (Ratia et al. 2014) and the areas A_1 to A_4 refer to effective cross sectional flow area. The numbering of areas is shown in Figures 7.3. Area 1 is a section upstream of the bridge while area 4 is a downstream section. Areas 2 and 3 are sections inside the bridge, near the entrance and exit respectively.

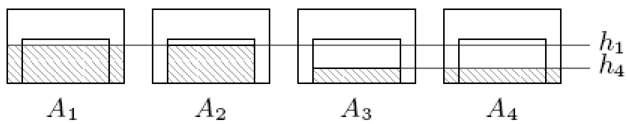


Figure 7.4 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in free surface bridges.

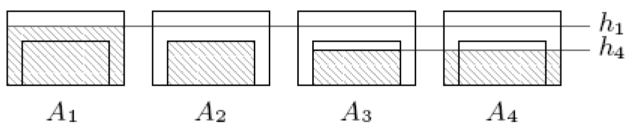


Figure 7.5 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in a partially submerged bridges.

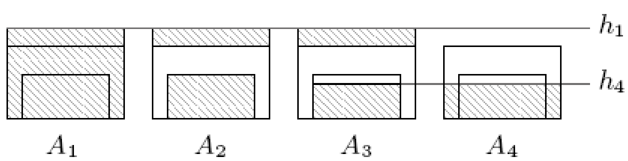


Figure 7.6 – Simple example of A_1,A_2, A_3 and A_4 used to calculate head loss in fully submerged bridges.

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

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

with Q_1 and Q_4 the total discharges in areas A_1 and A_4, expressed as a function of the water surface elevation, $d = h + z$. Different regimes can be described. Figure 7.4 shows a skecth of the areas considered in the free surface case, Figure 7.5 shows the equivalent areas for partially submerged bridges and Figure 7.6 for fully submerged bridges.

7.1.2.2 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 $\mathbf{S}_{\mathbf{n}b}$ at the edge of the RP where the bridge exists. The source term $\mathbf{S}_{\mathbf{n}b}$ is formulated as

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

with

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

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

Computation of ΔH in a real mesh is done as follows. The bridge is defined on cell edges (bold line in Figure 7.7), and the cells on both sides of these edges are considered to form two cross sections Γ_L and Γ_R (hatched cells in Figure 7.7). Note that it is possible to define bridges in arbitrary orientations and in structured/unstructured meshes.

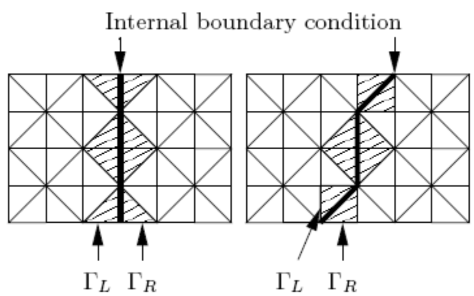


Figure 7.7 – Application of the scheme in triangular structured meshes. Normal bridge (left) and oblique bridge (right).

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 7.7. The discharge is computed as

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

and the cross sectional average water level surface is estimated as

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

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

7.1.2.3 Influence of the Bridge Width

The computation algorithm used in the Bridges component neglects the effect of the structure width (distance perpendicular to the bridge alignment) on the head loss. According to (Yarnell 1934*b*) and (Yarnell 1934*a*), the bridge width has a small influence on the flow variables such as water surface elevation and energy loss. Yarnell performed experiments in a laboratory flume with bridges having rectangular piers with width-to-length ratios (*w*:*l*) of 1:4, 1:7 and 1:13, where *w* is the pier dimension perpendicular to the flow direction and *l* the pier length parallel to the flow. Yarnell noted that the energy loss increased less than 10% for the configuration with longest piers. (Ratia et al. 2014) performed numerical simulations to confirm Yarnell's experiments using piers with the same with-to-length ratios and a wide range of approach discharges (see Figure 7.8).

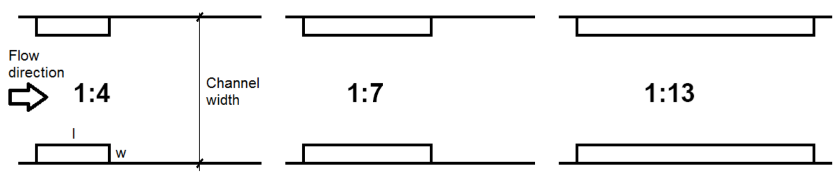


Figure 7.8 – Bridge pier proportions used to asses the influence of the structure width.

Numerical results indicate that the changes in total head loss across the structure are very similar for the three configurations (see Figure 7.9).

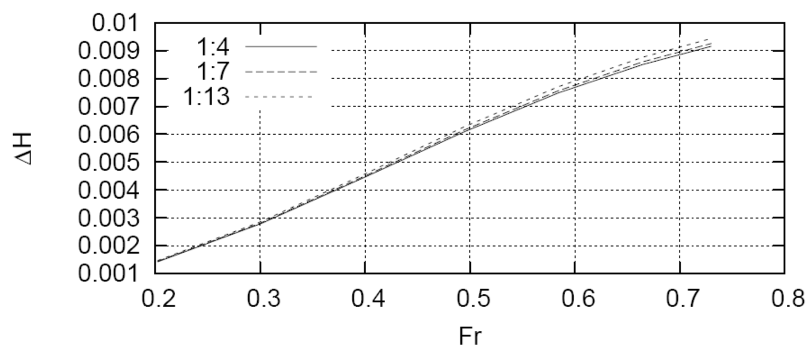


Figure 7.9 – Influence of the structure width on the total head change (ΔH) across the bridge as a function of the Froud number downstream.

7.2 Bridge Piers

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

To run a simulation with the *Bridge Piers* Component, you need to select the option in the *Control Data* panel of Hydronia Data Input Program as shown in Figure 7.10.

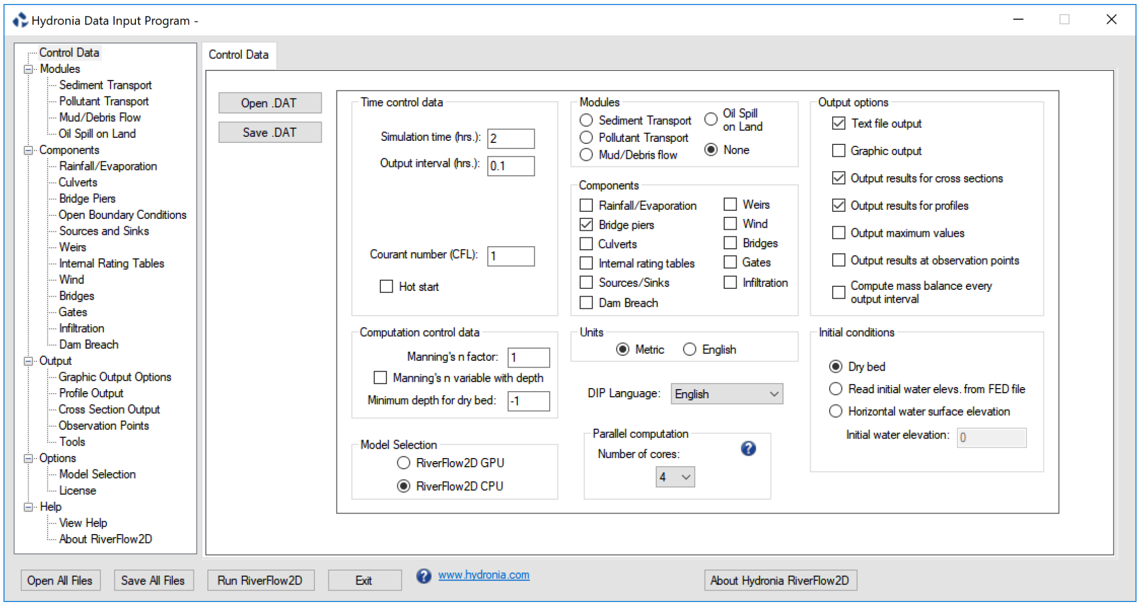


Figure 7.10 – Hydronia Data Input Program *Control Panel* dialog with the Bridge Piers component selected.

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

7.2.1 Bridge Pier Calculation

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

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

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

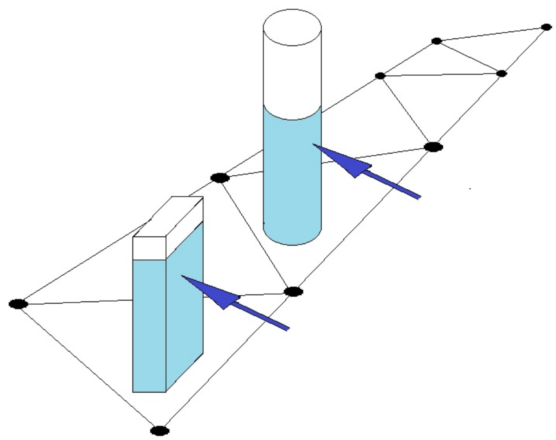


Figure 7.11 – Piers inside cells.

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

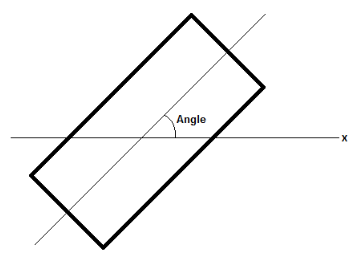


Figure 7.12 – Schematic view of a rectangular pier.

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

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

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

where A_e is the element area.

7.3 Culverts Component

The culvert component in HydroBID Flood 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 Hydronia Data Input Program dialog as shown in Figure 7.13.

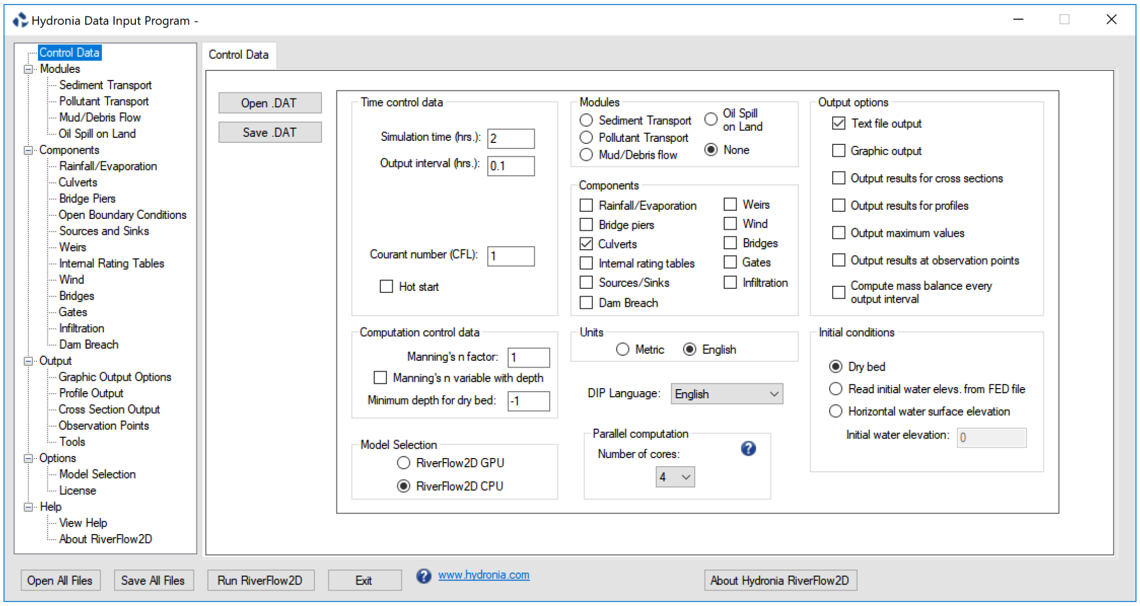


Figure 7.13 – Hydronia Data Input Program *Global Parameters* dialog with the Culverts Component selected.

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

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

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

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

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

2. Interpolate flow discharge from the rating table using the depth at the culvert inlet,
3. If depth at the culvert inlet is lower than minimum value in the rating table, then the discharge is assumed to be zero.
4. If depth at entrance is higher than maximum value in the rating table, then the discharge is assumed to be equal to that of the maximum depth.
5. The computed discharge is subtracted from the inlet element and added to the outlet element assuming instantaneous water volume transmission.

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

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

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

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

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

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

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

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

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

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

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

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

Continued on next page

Table 7.3 – continued from previous page

TYPE OF CULVERT	ENTRANCE DESCRIPTION*	ENTRANCE LOSS COEFFICIENT K_e
	Crown edge rounded to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 10° to 30° to barrel	
	Square-edged at crown	0.5
	Wingwalls parallel to embankment	
	Square-edged at crown	0.7

*See Table 7.5 for inlet configurations.

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

BARREL MATERIAL	BARREL SHAPE	INLET DESCRIPTION*	K'	M	C'	Y
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45° bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7° bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30° to 75° flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90° and 15° flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0° flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45° flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18° to 33.7° flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; 3/4 inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45° bevels	1.5752	0.6670	1.0101	0.8200
Concrete	Rectangular	Headwall; 33.7° bevels	1.5466	0.6670	0.8107	0.8650
Concrete	Rectangular	Headwall; 45° skew; 3/4 in chamfers	1.6611	0.6670	1.2932	0.7300
Concrete	Rectangular	Headwall; 30° skew; 3/4 in chamfers	1.6961	0.6670	1.3672	0.7050
Concrete	Rectangular	Headwall; 15° skew; 3/4 in chamfers	.7343	0.6670	1.4493	0.6800
Concrete	Rectangular	Headwall; 10-45° skew; 45° bevels	1.5848	0.6670	1.0520	0.7500

Continued on next page

Table 7.4 – continued from previous page

BARREL MATERIAL	BARREL SHAPE	INLET DESCRIPTION*	K'	M	C'	Y
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

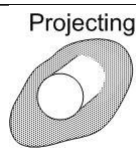
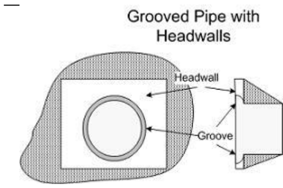
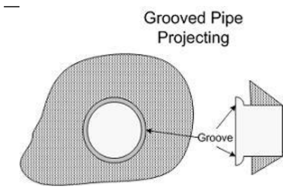
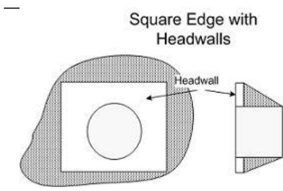
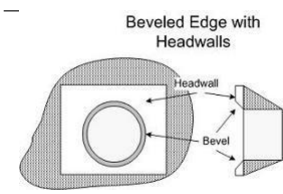
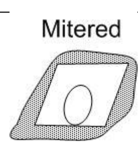
Continued on next page

Table 7.4 – continued from previous page

BARREL MATERIAL	BARREL SHAPE	INLET DESCRIPTION*	K’	M	C’	Y
Concrete	Rectangular	Headwall; 45°bevels	1.5752	0.6670	1.0101	0.8200

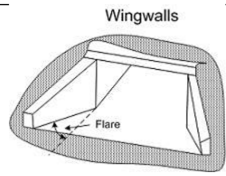
*See Table 7.5 for inlet configurations.

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

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

Continued on next page

Table 7.5 – continued from previous page

INLET CONFIGURATION	DESCRIPTION
	Wingwalls are used when the culvert is shorter than the embankment and prevents embankment material from falling into the culvert

The culvert computation algorithm works as follows:

1. If at least one of the culvert ends is wet, Determine the flow direction based on the water surface elevations at each culvert end.
2. Compute the culvert discharge using inlet control formulas.
3. Compute the culvert discharge using outlet control formulas.
4. Select the minimum discharge from the inlet and outlet control discharges.
5. If depth at the culvert inlet is lower than minimum value in the rating table, then the discharge is assumed to be zero.
6. The computed discharge is subtracted from the inlet element and added to the outlet element assuming instantaneous water volume transmission.

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

7.3.3 Assumptions of Culvert Calculations

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

7.4 Gates Component

The GATES components allows integrating gates inside the modeling region. Each gate needs to be defined in terms of its plan alignment, crest elevation (Zc), gate height ($Hgate$) and the time history of apertures (Ha) given as a table in a file associated to each structure (see Figure 7.14). Figure 7.15 shows the flow modes that can be calculated through gates that include submergence and overtopping.

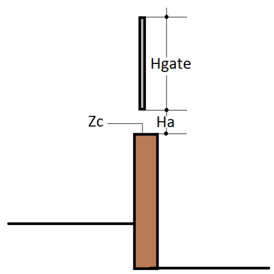


Figure 7.14 – Schematic cut view perpendicular to a gate structure.

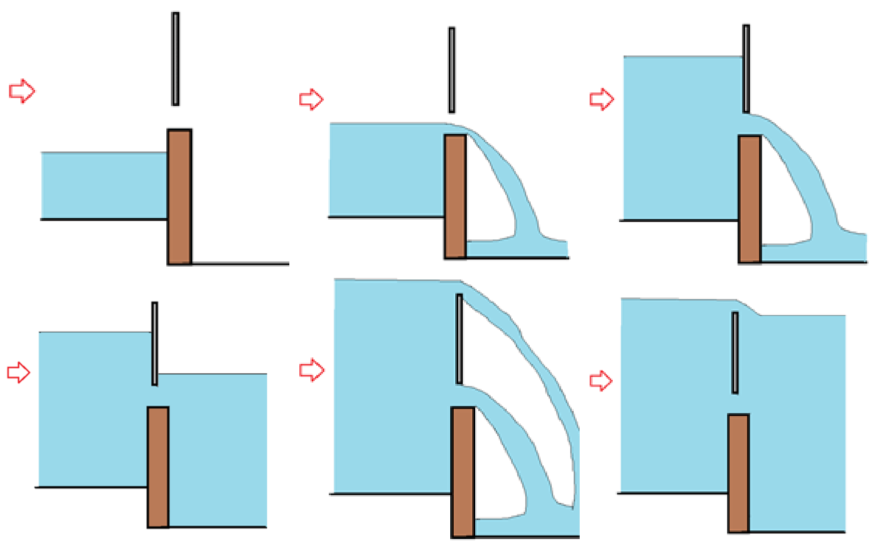


Figure 7.15 – Flow modes across gates.

To run a simulation with the gates component, you need to select the option in the *Control Data* panel of Hydronia Data Input Program as shown in Figure 7.16.

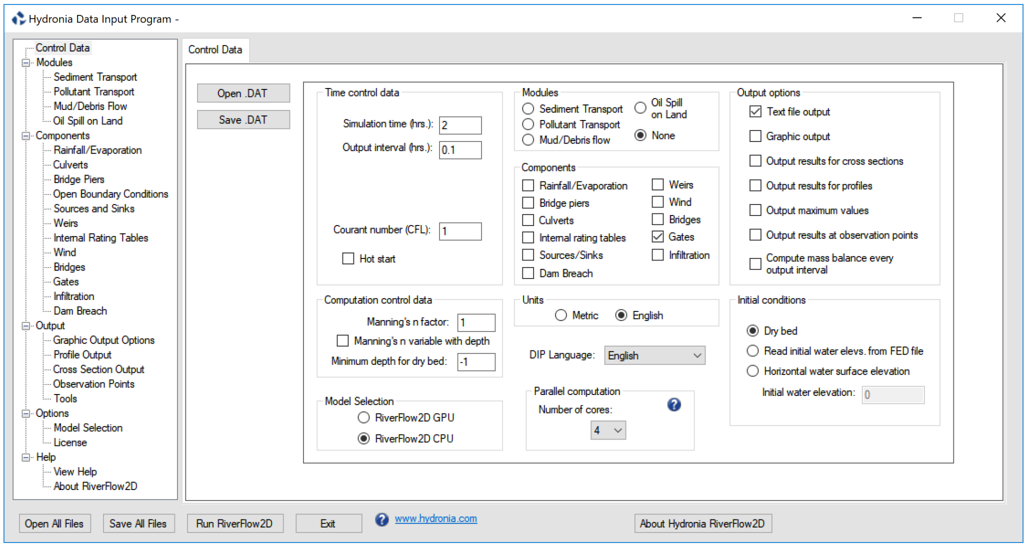


Figure 7.16 – Hydronia Data Input Program *Control Data* panel with the Gates Component selected.

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.

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

7.4.1 Gate Calculations

The gate is simulated by assuming that the discharge per unit breadth q crossing the gate is governed by the difference between the water surface level ($d = h + z$) on both sides of the gate, referred to as d_l upstream of the gate and d_r downstream of the gate, and by the allowable gate opening, G_o. Several situations are envisaged. In the case that $G_o = 0$ the gate behaves as a solid wall and no flow crosses the gate. When the gate opening is larger than the surface water level on both sides, it no longer influences the flow. In any other case, assuming that $d_l < d_r$, without loss of generality, two different flow situations can occur depending on the relative values of G_o, z_l, z_r, d_l and d_r. When $G_o + max(z_l, z_r) < min(d_l, d_r)$, Figure 7.17, the discharge is given by

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

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

When $G_o + max(z_l, z_r) > min(d_l, d_r)$, Figure 7.18, the discharge is given by

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

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

Figure 7.17 – Water levels for discharge under a gate in submerged conditions formulated as in (G1).

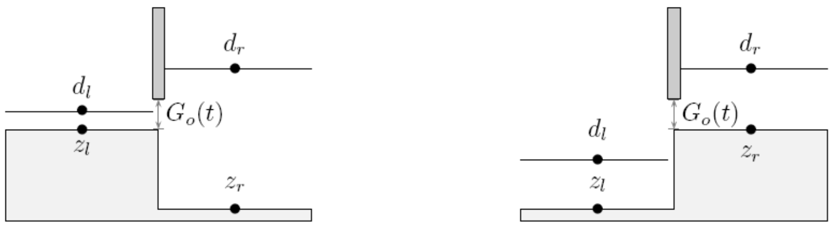


Figure 7.18 – Water levels for discharge under a gate in submerged conditions formulated as in (G2).

7.5 Rainfall and Evaporation

This section and the following about Infiltration describe the Hydrologic computations implemented in HydroBID Flood. The component includes capabilities to integrate rainfall, evaporation and infiltration in the model simulations. You may use these components to perform hydrologic simulations with the following options:

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

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

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

In HydroBID Flood the rainfall imposed before calculating infiltration. This is an important because the infiltration capacity strongly depends on the rainfall intensity, as we will see in the next section.

7.6 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 HydroBID Flood, 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).

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 water depth to determine the infiltration rates. The SCS-CN method calculates infiltration as a function of the given rainfall and does not consider the pounded water.

7.6.1 Horton Infiltration Model

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

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

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

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

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

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

Continued on next page

Table 7.6 – continued from previous page

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

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

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

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

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

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

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

where t_p represents the ponding time.

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

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

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

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

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

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

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

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

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

7.6.2 Green-Ampt Infiltration Model

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

Table 7.8 – Mean values and standard deviation for Green-Ampt model parameters. Source: Rawls & Brakensiek 1983.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

7.6.3 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:

$$\frac{F}{S} = \frac{RO}{RV - I_a} \tag{7.33}$$

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

$$RV = RO + F + I_a \tag{7.34}$$

By combining (7.33) and (7.34) and taking into consideration that the runoff cannot begin until the initial abstraction has been met:

$$RO = \begin{cases} \frac{(RV-I_a)^2}{RV-I_a+S} & (RV > I_a) \\ 0 & (RV \leq I_a) \end{cases} \tag{7.35}$$

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

$$S = \frac{25400}{CN} - 254 \tag{7.36}$$

The initial abstraction is assumed proportional to S:

$$I_a = \alpha S \tag{7.37}$$

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 HydroBID Flood , and its influence in water runoff was studied in Caviedes et al. (Caviedes-Voullième, García-Navarro & Murillo 2012).

To determine appropriate Curve Numbers we recommend following the guidelines provided in (USDA 1986).

It is important to remark that SCS-CN method was not designed to consider time. Following (Caviedes-Voullième et al. 2012), when the method is implemented in a complex simulator, a time-advancing methodology is used. The method is not applied to the entire catchment. Runoff is calculated for every cell in every time step, using the cumulative rainfall since the beginning of the storm.

The SCS-CN method can be extended in order to estimate the temporal distribution of the water losses. By combining again (7.33) and (7.34) but solving for F:

$$F = \frac{S\,(RV - I_a)}{RV - I_a + S}, \qquad RV \geq I_a \tag{7.38}$$

By differentiating (7.38), taking into account that I_a and S are constant magnitudes, the following expression for the infiltration rate is obtained ((Chow et al. 1988)):

$$f = \frac{dF}{dt} = \frac{S^2 R}{RV - I_a + S} \tag{7.39}$$

being R the rainfall rate, defined as follows:

$$R = \frac{dRV}{dt} \tag{7.40}$$

7.6.3.1 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 (Chow et al. 1988) 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 7.9.

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

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

Traditionally (Chow et al. 1988), the Curve Number for dry or wet conditions has been recalculated in terms of the standard conditions according to Eqs. 7.41 and 7.42:

$$CN(I) = \frac{4.2CN(II)}{10 - 0.058CN(II)} \tag{7.41}$$

$$CN(III) = \frac{23CN(II)}{10 + 0.13CN(II)} \tag{7.42}$$

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

7.7 Wind Component

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

$$\mathbf{S} = (0,\ gh(S_{0x} - S_{fx}) + S_{wx},\ gh(S_{0y} - S_{fy}) + S_{wy})^T \tag{7.43}$$

where

$$S_{wx} = C_a \frac{\rho_a}{\rho_w} \mathcal{U} |\boldsymbol{\mathcal{U}}| \qquad S_{wy} = C_a \frac{\rho_a}{\rho_w} \mathcal{V} |\boldsymbol{\mathcal{U}}| \tag{7.44}$$

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

7.8 Internal Rating Tables

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

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

To run a simulation with Internal Rating Tables, you need to select the option in the *Control Data* panel of Hydronia Data Input Program shown in Figure 7.19.

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

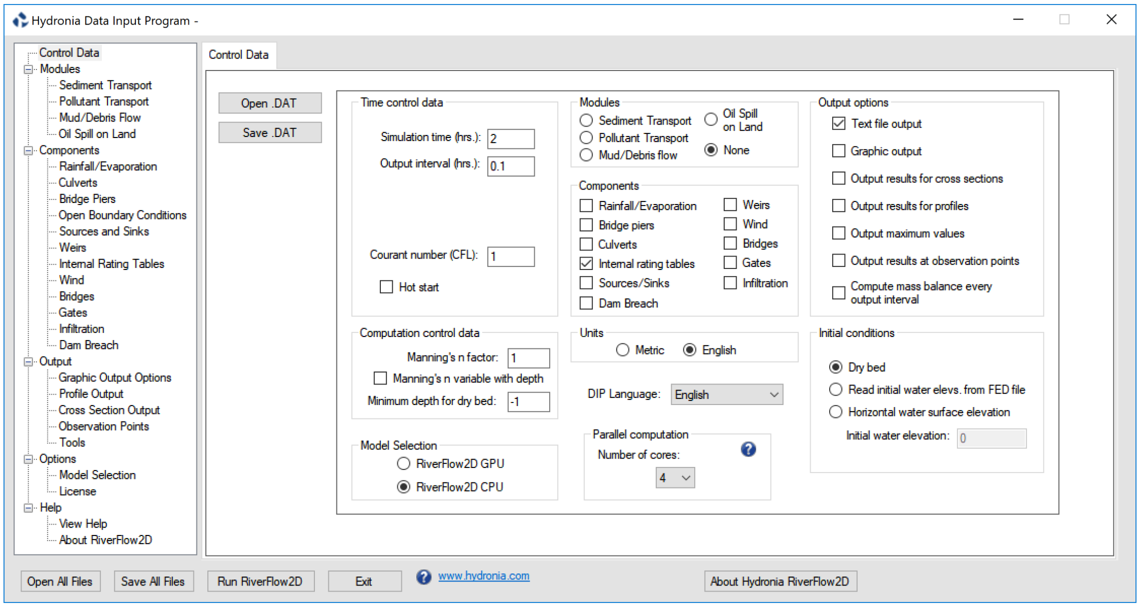


Figure 7.19 – Hydronia Data Input Program *Control Data* panel with the IRT component selected.

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

7.8.1 Internal Rating Table Calculations

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

The IRT calculation algorithm works as follows:

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

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

7.8.2 Assumptions of Internal Rating Table Calculations

The rating table does not account for outlet control.

7.9 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 Hydronia Data Input Program as shown in Figure 7.20.

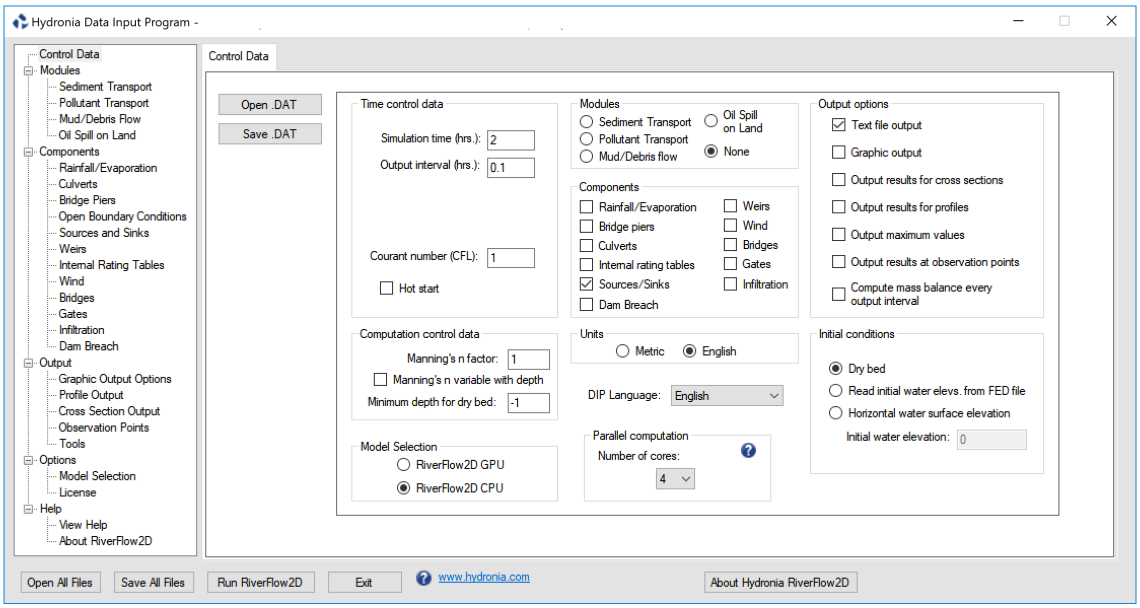


Figure 7.20 – Hydronia Data Input Program *Control Data* panel with the Sources and Sinks component selected.

Sources and Sinks data is entered in HydroBID Flood *Sources* layer.

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

7.10 Weirs

HydroBID Flood 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 Hydronia Data Input Program as shown in Figure 7.21.

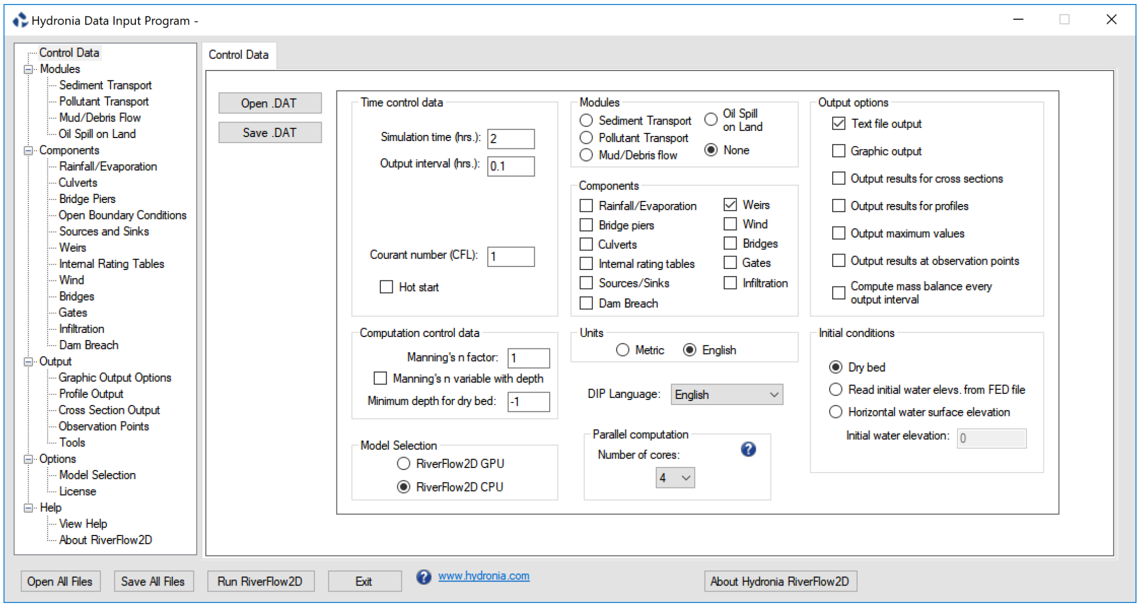


Figure 7.21 – Hydronia Data Input ProgramControl Data panel with the Weirs component selected.

Weir plan data is entered in the QGIS *Weirs* layer.

Since HydroBID Flood 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.

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

7.10.1 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:

$$d_w = h_{crest} + MAX\,(z_L, z_R) \tag{7.45}$$

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

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

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

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

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

$$q = C_f\,(d_r - d_l)^{3/2} \tag{7.46}$$

with C_f the weir discharge coefficient, with typical values around 1.84 for metric units, and 3.34 for English units.

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

$$q = C_f\,(d_r - H_w)^{3/2} \tag{7.47}$$

7.10.2 Assumptions of Weir Calculations

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

7.11 Dam Breach Modeling

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

In HydroBID Flood, the dam is defined as an internal boundary condition and modeled as a progressive trapezoid. For a complete parametrization of the breach, the next parameters and variables are used (see Figure 7.22):

- 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:

$$Q_b = KBH^{3/2} \tag{7.48}$$

where $H = h + z - H_v$, $H_v = z_{crest} - H_b$, $B(H) = b(t) + 2\frac{H(t)}{tg\alpha}$, and K is a discharge coefficient. K typical values are around 1.25 for metric units, and 3.1 for English units.

The discharge computed in (7.48) will be distributed among the cells included in the breach top length $B(H_b)$:

$$B(H_b) = b(t) + 2\frac{H_b(t)}{tg\alpha} \tag{7.49}$$

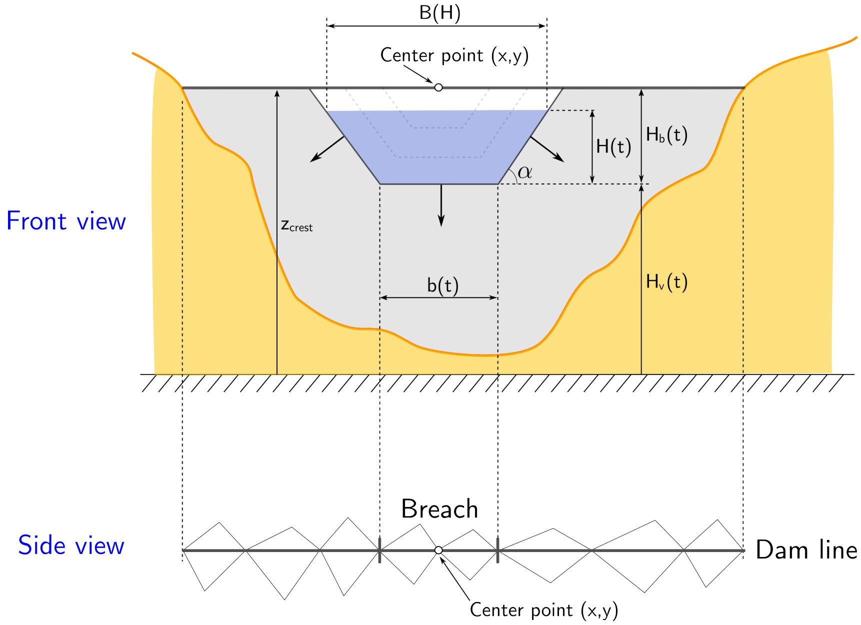


Figure 7.22 – Coordinate system for the dam breach representation.

8 — Hydronia Data Input Program (DIP)

Hydronia Data Input Program (DIP) user interface provides the tools to enter non-spatial data, and run RiverFlow2D. All changes introduced on the DIP will be saved to the native RiverFlow2D data files.

The DIP appears when you export the files to RiverFlow2D 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:

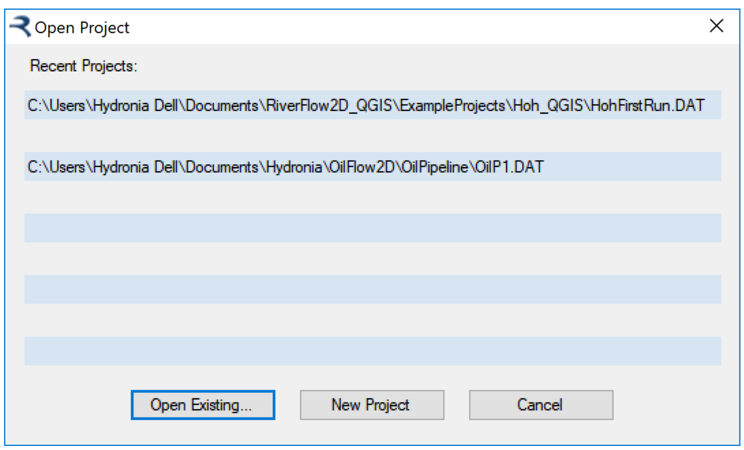


Figure 8.1 – Hydronia Data Input ProgramOpen Project Dialog.

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

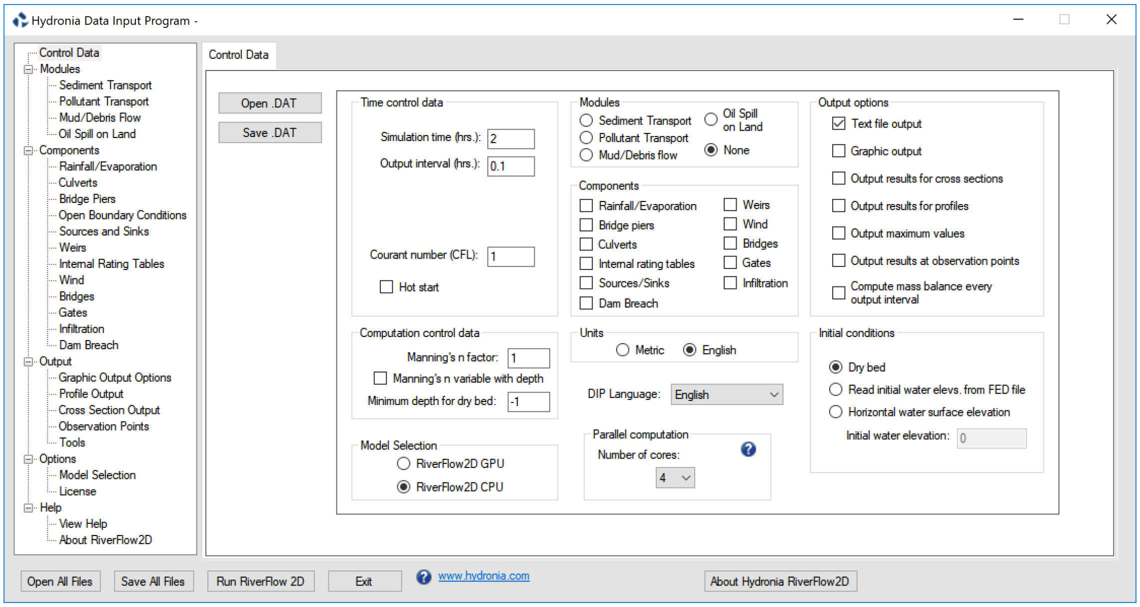


Figure 8.2 – Main Hydronia Data Input Program window.

Hydronia Data Input Programlets you select different model engines. Using the options the Model Selection frame you can select between RiverFlow2D CPU or RiverFlow2D GPU. Note that to run the GPU version you need the appropriate GPU hardware.

The following sections describe the panel dialogs of Hydronia Data Input Program.

8.1 Control Data Panel (.DAT files)

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

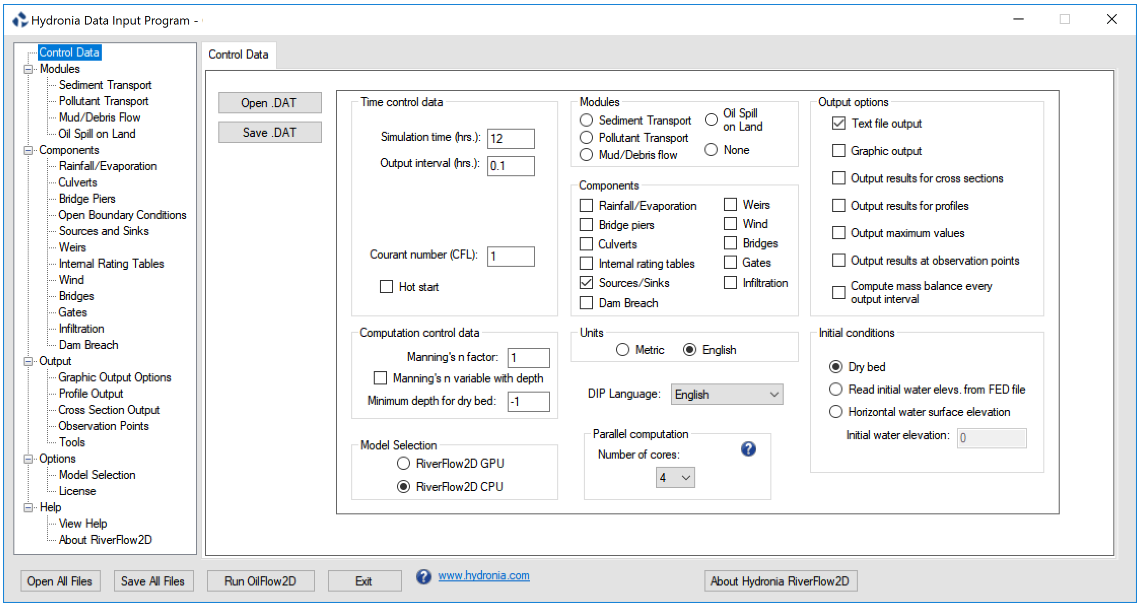


Figure 8.3 – *Control Data* Panel.

Table 8.1 – Buttons in the *Control Data* Panel.

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

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

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

Table 8.3 – Units Frame on the *Control Data* Panel.

CONTROL NAME	**DESCRIPTION**
Metric	Select this option to work in metric units. Coordinates are given in meters, velocities in m/s, discharge in m^3/s, etc. Text

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

CONTROL NAME	DESCRIPTION
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.

Table 8.5 – Output Options Data Frame on the *Control Data* Panel.

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

Table 8.6 – Components Data Frame on the *Control Data* Panel.

CONTROL NAME	DESCRIPTION
Rainfall/Evaporation	Option to activate rainfall and/or evaporation. The required data has to be entered in the Rainfall /Evaporation Panel. This data is written to `.LRAIN` file.
Bridge piers	Switch to allow accounting for pier drag force. The Bridge piers data can be edited in the Bridge Piers Panel. The data is written to `.PIERS` file.

Continued on next page

Table 8.6 – continued from previous page

CONTROL NAME	DESCRIPTION
Bridges	Switch to model Bridges using the bridge cross section geometry and accounting for energy losses. The data can be edited in the Bridges Panel. The data is written to `.BRIDGES` file.
Culverts	Switch indicating if one dimensional culverts will be used. The Culverts data can be edited in the Culverts Panel. The data is written to `.CULVERTS` file.
Gates	Switch to model gates. The data can be edited in the *Gates* Panel. The data is written to `.GATES` file.
Infiltration	Option to activate Infiltration loss calculations. The required data has to be entered in the Infiltration Panel. This data is written to `.LINF` file.
Internal rating tables	Switch to allow using internal rating tables. The data can be edited in the Internal Rating Tables Panel. The data is written to `.IRT` file.
Mud/Debris Flow	Option to activate the Mud and Debris Flow modeling. The data can be edited in the Mud/Debris Flow Panel. The data is written to the `.MUD` file.
Oil Spill on Land	Option to activate the Overland Oil Spill modeling. The data can be edited in the Oil Spill on Land Panel. The data is written to the `.OILP` file.
Pollutant Transport	Option to activate pollutant transport modeling. The pollutant transport data can be edited in the Pollutant Transport Panel. The data is written to the `.SOLUTES` file.
Sources/Sinks	Switch to indicate existence of sources or sinks. The sources/sinks data can be edited in the Sources/Sinks Panel. The data is written to `.SOURCES` file.
Weirs	Switch to indicate the existence of Weir computation. The Weir data can be edited in the Weirs Panel. The data is written to `.WEIRS` file.
Wind	This option activates the calculation of wind stress on the watersurface. The data can be edited in the Wind Panel. The data is written to `.WIND` file.
Sediment transport	Option to activate sediment transport modeling with erosion and deposition for a mobile bed. The sediment transport data can be edited in the Sediment Transport Panel. This data is written to the `.SEDS` and `.SEDB` files.

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

CONTROL NAME	DESCRIPTION
Dry bed	The simulation will start with a fully dry bed. For discharge boundary conditions, an arbitrary depth (>0.0) is assigned to calculate the inflow for the first time-step. Subsequently the flow depth at the boundary will be determined by the model.

Continued on next page

Table 8.7 – continued from previous page

CONTROL NAME	DESCRIPTION
Read initial water elevations from .FED file	Initial water surface elevations will be read from the .FED file. It is possible to assign a spatially variable initial water surface elevation in the Initial Conditions Layer.
Horizontal water surface elevation	Use this option to start a simulation with a user provided initial horizontal water surface elevation.
Initial water elevation	Initial water surface elevation on the whole mesh. If initial water elevation is set to -9999, the program will assign a constant water elevation equal to the highest bed elevation on the mesh.

8.2 Sediment Transport Panel (.SEDS and .SEDB Files)

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

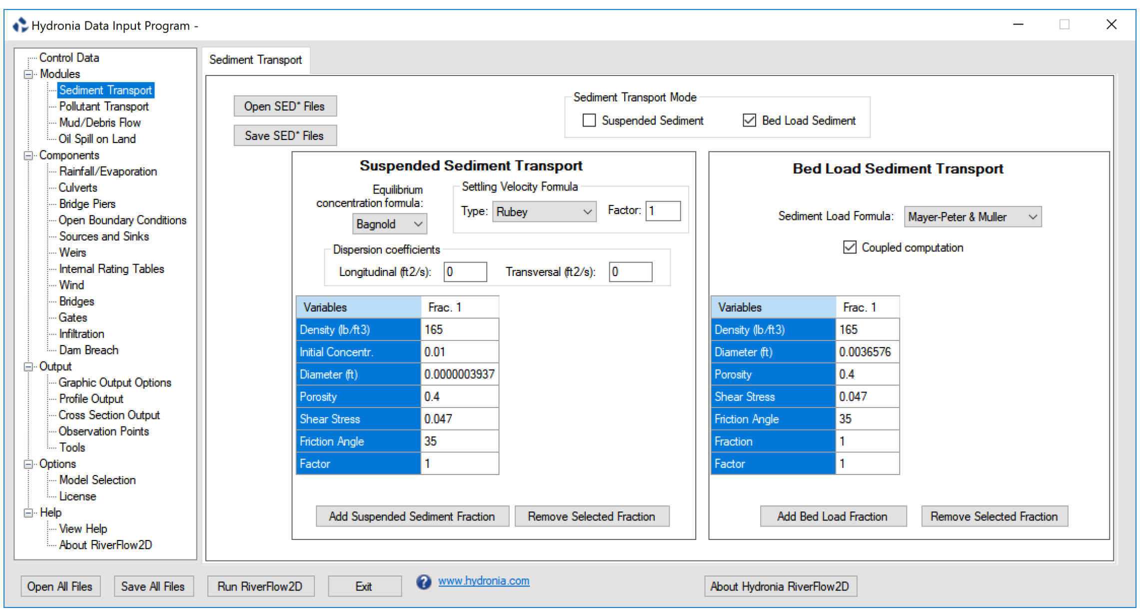


Figure 8.4 – Sediment Transport Panel.

Table 8.8 – Parameters on the Sediment transport Mode frame and buttons of the Sediment Transport Panel.

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

Table 8.9 – Parameters on the Suspended Sediment transport frame of the Sediment Transport Panel.

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

Continued on next page

Table 8.9 – continued from previous page

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

Table 8.10 – Parameters on the Bed Load Sediment transport frame of the Sediment Transport Panel.

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

8.2.0.1 Comments for the .SEDS and .SEDB Files

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

fraction entered.

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

8.3 Mud and Debris Flow Data Panel (.MUD File)

This panel allows entering mud and debris flow data for the RiverFlow2D model. To activate this panel, first select *Mud/Debris Flow* on the *Components Frame* of the *Control Data* Panel. Note that the program will determine Yield Stress, Viscosity and Density based on the Volumetric Concentration Cv and the publication selected from the drop down lists.

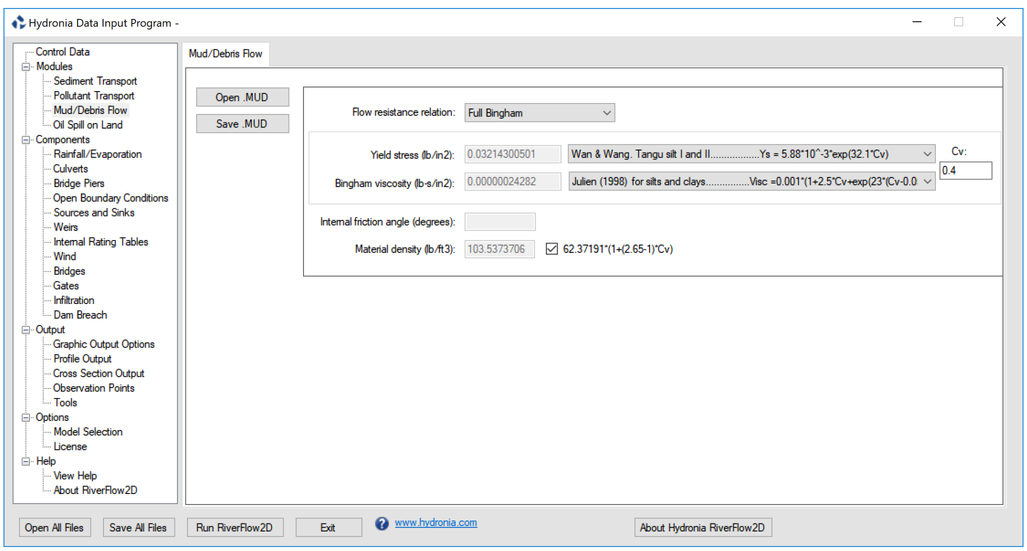


Figure 8.5 – Mud/Debris Flow Panel.

Table 8.11 – Parameters on the Mud/Debris Flow Panel.

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

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

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

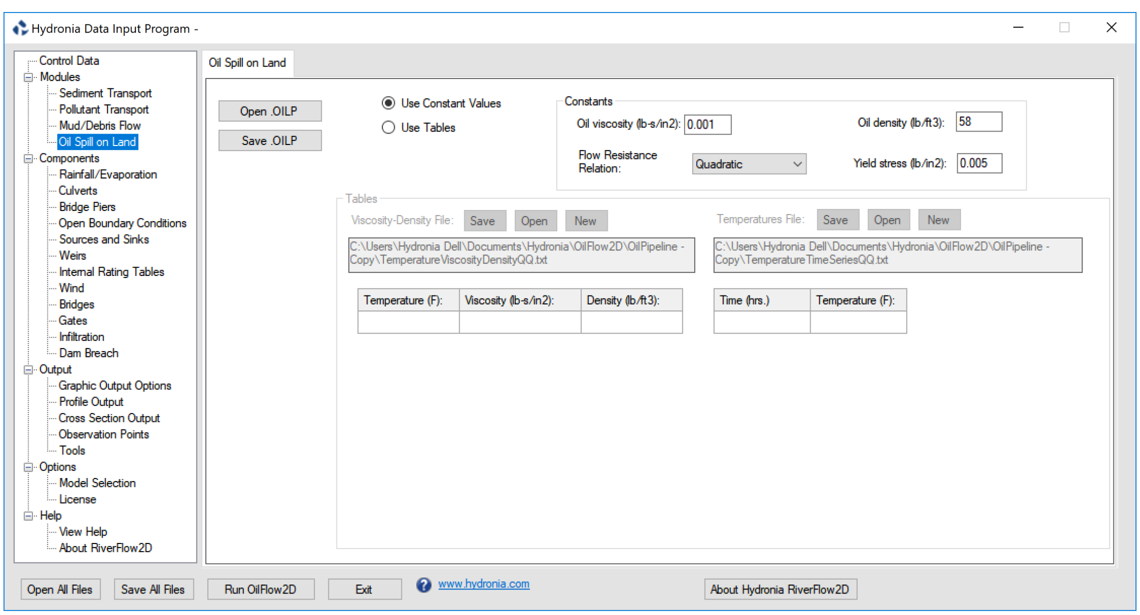


Figure 8.6 – Oil Spill on Land Panel (OilFlow2D model).

Table 8.12 – Parameters on the Oil Spill on Land Panel (OilFlow2D model).

CONTROL NAME	DESCRIPTION
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).

8.5 Graphic Output Options Tab (.PLT File)

This panel allows entering options to control HydroBID Flood output. To activate this panel, first select *Graphic Output Options* from the *Output* group on the left panel Hydronia Data Input Program.

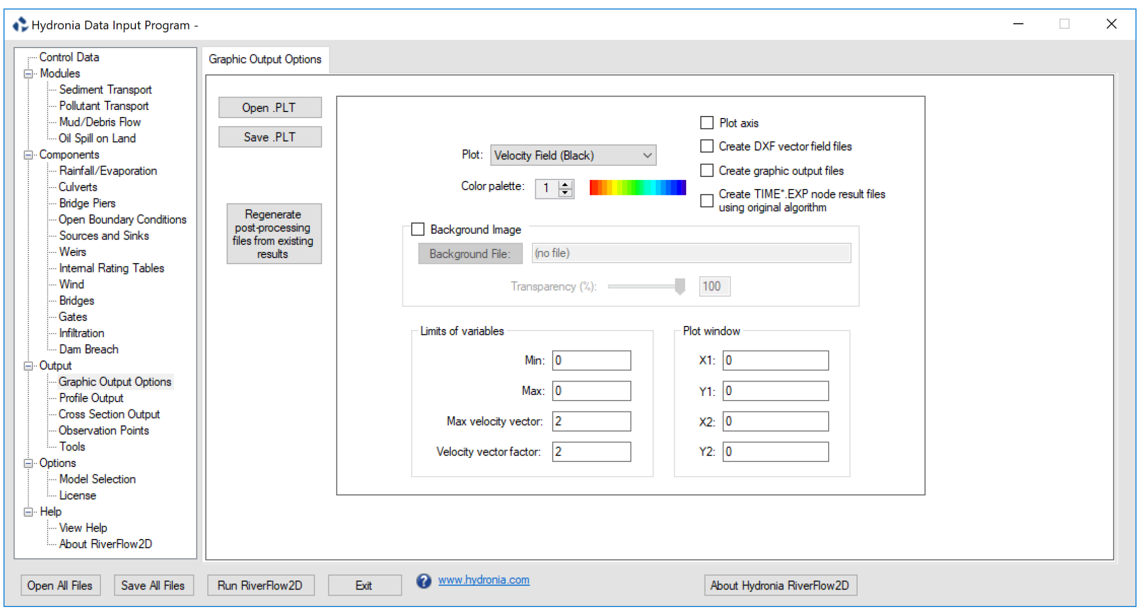


Figure 8.7 – Graphic Output Panel.

Table 8.13 – Parameters on the Graphic Output Option Panel.

CONTROL NAME	DESCRIPTION
Buttons	
Open `.PLT`	Opens an existing `.PLT` output data file.
Save `.PLT`	Saves only the graphic output data to a `.PLT` file.

Continued on next page

Table 8.13 – continued from previous page

CONTROL NAME	DESCRIPTION
Regenerate post-processing files from existing results	Use this button to create output files from existing simulations. The output `state*.out` files written during a previous simulation must be available.
Plot Frame	
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: `<ProjectName>_MESH.DXF`.
Create graphic output files	For future use.
Create `TIME*.EXP` node result files using original algorithm.	Using this option RiverFlow2D when running will generate `TIME*.EXP` files using an algorithm to compute nodal values from cell values that was available in versions older than 2018.
Background Image Frame	
Background image	For future use.
Background file	For future use.
Transparency	For future use.
List of variables Frame	
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.

8.6 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 `.PRFI` and `.PRFE` files. To activate this panel, first select the *Profile Output* in *Output* from the *Output* group on the left panel of Hydronia Data Input Program.

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

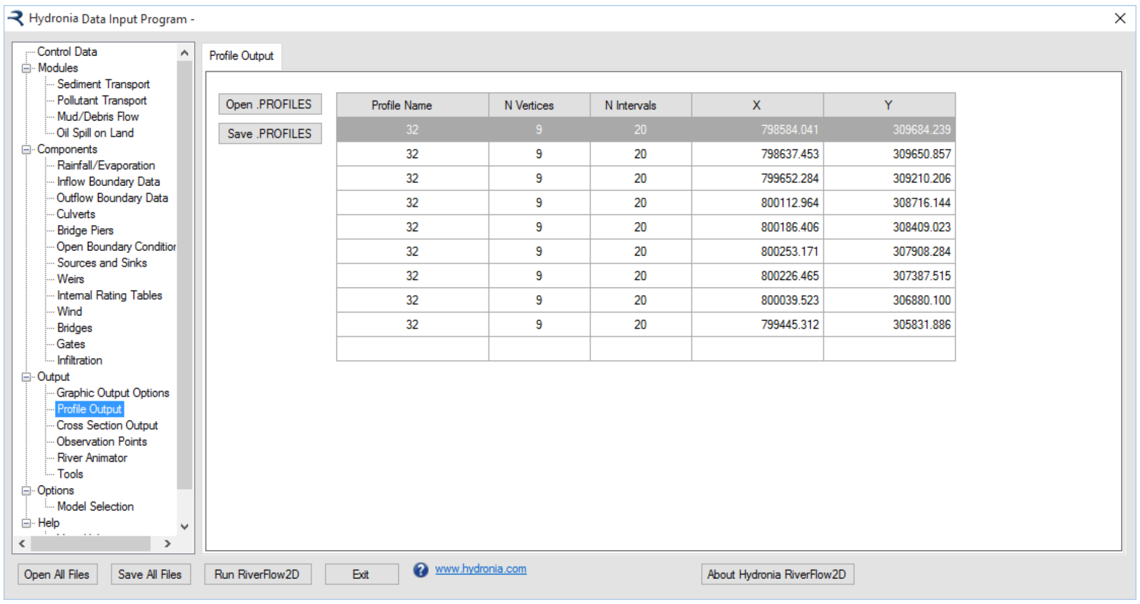


Figure 8.8 – Profile Output File.

Table 8.14 – Parameters on the Profile Output Panel.

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

8.7 Cross Section Output Panel (.XSECS File)

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

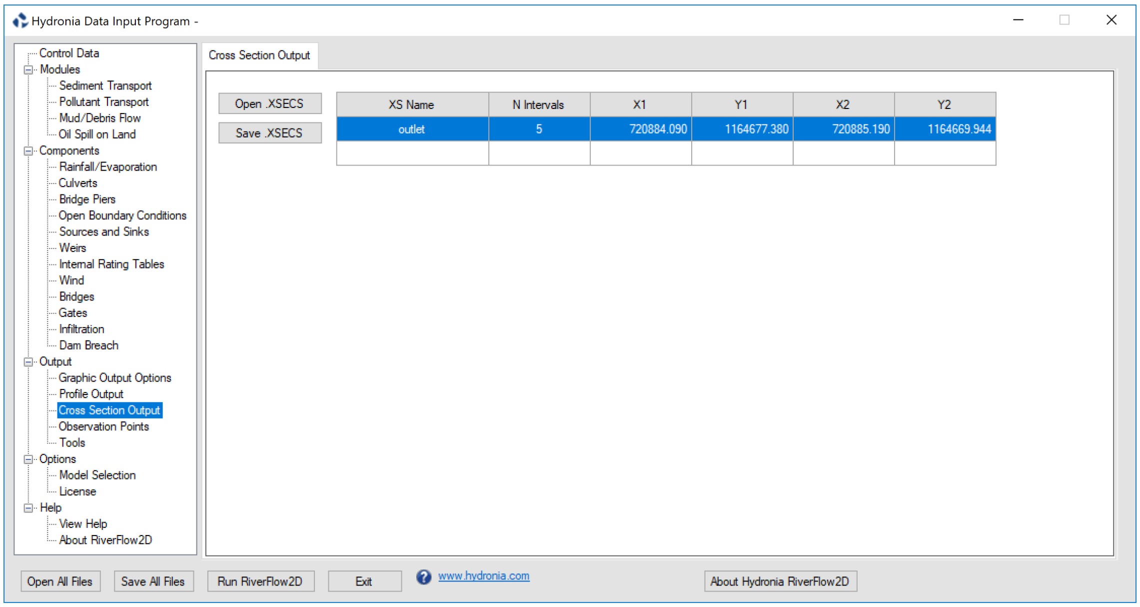


Figure 8.9 – Cross Section Output Panel.

Table 8.15 – Parameters on the Inflow Boundary Data Panel.

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

8.8 Culverts Panel (.CULVERTS File)

This panel is used to display the content of the `.CULVERTS` file and enter data for culverts. Figure 8.10 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 Hydronia Data Input Program.

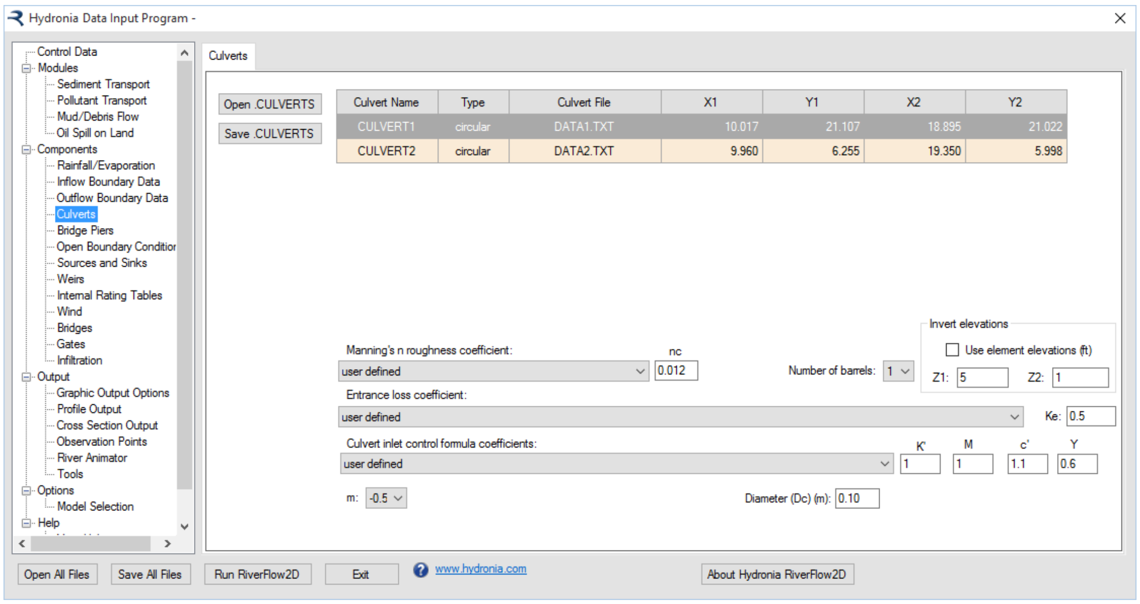


Figure 8.10 – Culverts Panel showing data in rating curve.

Figure 8.10 shows the corresponding data entry controls that appear when selecting the first row for Culvert1 that is a circular culvert.

Table 8.16 – Parameters on the Culverts Panel.

COLUMN NAME	DESCRIPTION
Culvert data	
Culvert Name	Culvert name. Should not contain spaces and must have less than 26 characters.
Type	Type of culvert. For Type = 0, culvert discharge is computed from a user given rating table on the Culvert File. For Types = 1 and 2, discharge is computed using culvert equations based on culvert characteristics provided in the Culvert File.
Culvert File	Culvert rating table file name or culvert characteristic data. Name Should not contain spaces and must have less than 26 characters.
X1, Y1, X2, Y2	Coordinates of vertices defining each culvert line.
Manning's roughness coefficient	Culvert Manning's n Coefficient given by Table 7.2.
Entrance loss coefficient	Culvert entrance loss coefficient given by Table 7.3.

Continued on next page

Table 8.16 – continued from previous page

COLUMN NAME	DESCRIPTION
Culvert inlet control formula coefficients	Culvert inlet control formula coefficients given by Table 7.4.
m	Inlet form coefficient. m=0.7 for mitered inlets, m=-0.5 for all other inlets.
Barrel height (Hb)	Barrel height for box culverts (ft or m). Only for box culverts: CulvertType = 1.
Barrel width (Base)	Barrel width for box culverts (m or ft). Only for box culverts: CulvertType = 1.
Diameter (Dc)	Barrel diameter for circular culverts (m or ft). Only for circular culverts: CulvertType = 2.
Number of barrels	Number of identical barrels.
Use element elevations	When this check box is selected the model will extract the inlet and outlet invert elevations from the element or cell elevations of the culvert ending points. If the check box is not selected, the user can enter the inlet invert elevation (Z1) and outlet invert elevations (Z2) that may be different from the element/cell elevations.
Buttons	
Open `.CULVERTS`	Opens an existing `.CULVERTS` file.
Save `.CULVERTS`	Saves only the culvert data to a `.CULVERTS` file.

8.9 Internal Rating Tables Panel (.IRT File)

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

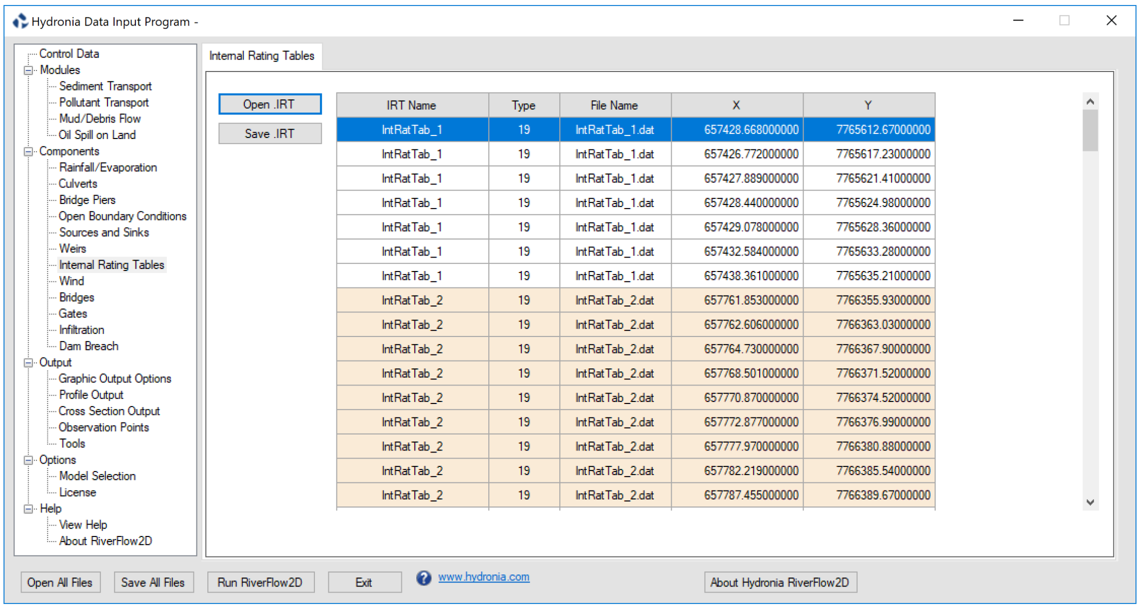


Figure 8.11 – Internal Rating Tables Panel.

Table 8.17 – Parameters on the Internal Rating Tables Panel.

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

8.10 Weirs Panel (.WEIRS File)

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

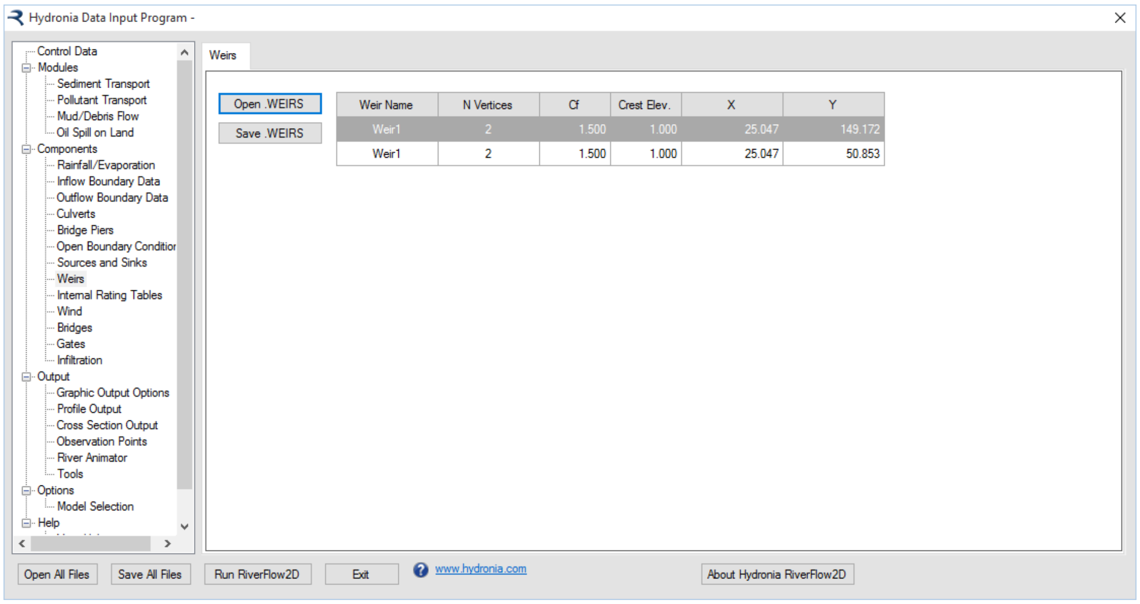


Figure 8.12 – Weirs Panel.

Table 8.18 – Parameters on the Weirs Panel.

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

8.11 Sources/Sinks Panel (.SOURCES File)

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

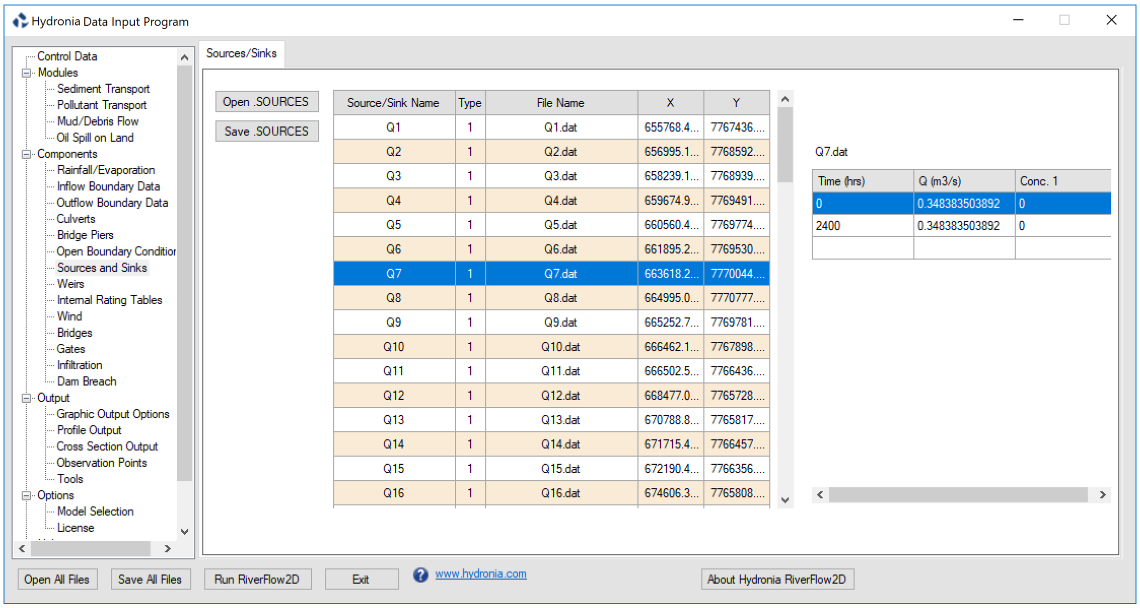


Figure 8.13 – Sources/Sinks Panel.

Table 8.19 – Parameters on the Sources/Sinks Panel.

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

8.12 Open Boundary Conditions Panel (.IFL File)

This panel is used to display the content of the `.IFL` file. Since this file is automatically generated by the RiverFlow2D model based on the user input in QGIS. This file should not be edited since, any change made will be overwritten when the model starts running. To activate this panel, first select the *Open Boundary Conditions* from the Components group on the left panel of Hydronia Data Input Program.

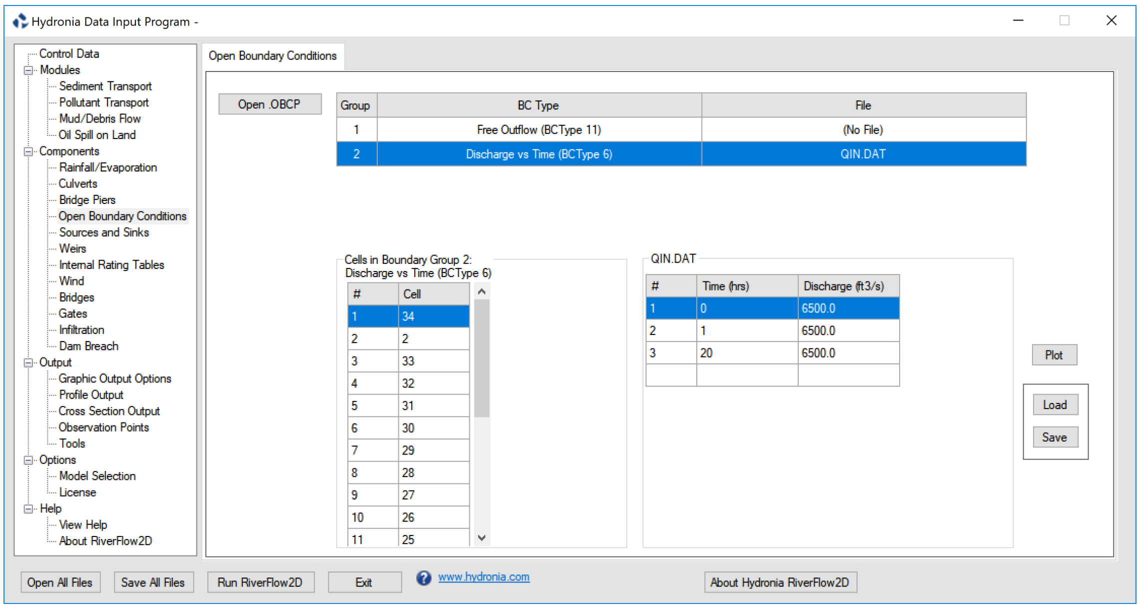


Figure 8.14 – Open Boundary Conditions Panel.

Table 8.20 – Parameters on the Open Boundary Condition Panel.

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

8.13 Bridge Piers Panel (.PIERS File)

This panel is used to display the content of the `.PIERS` 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 Hydronia Data Input Program.

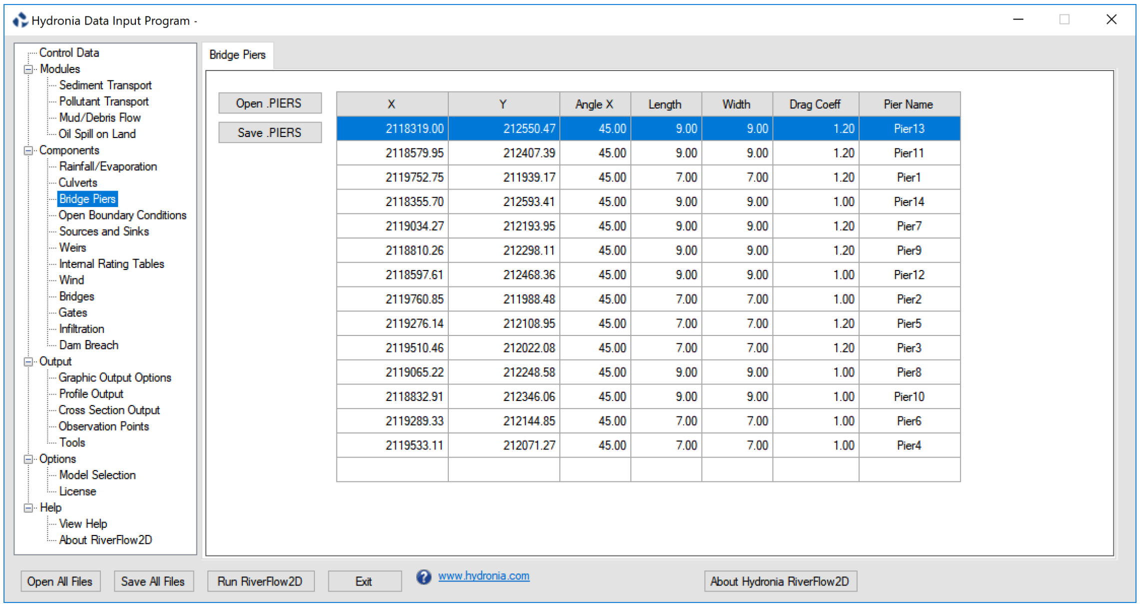


Figure 8.15 – Bridge Piers Panel.

Table 8.21 – Parameters on the Bridge Piers Panel.

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

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

8.14 Observation Points Panel (.OBS File)

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

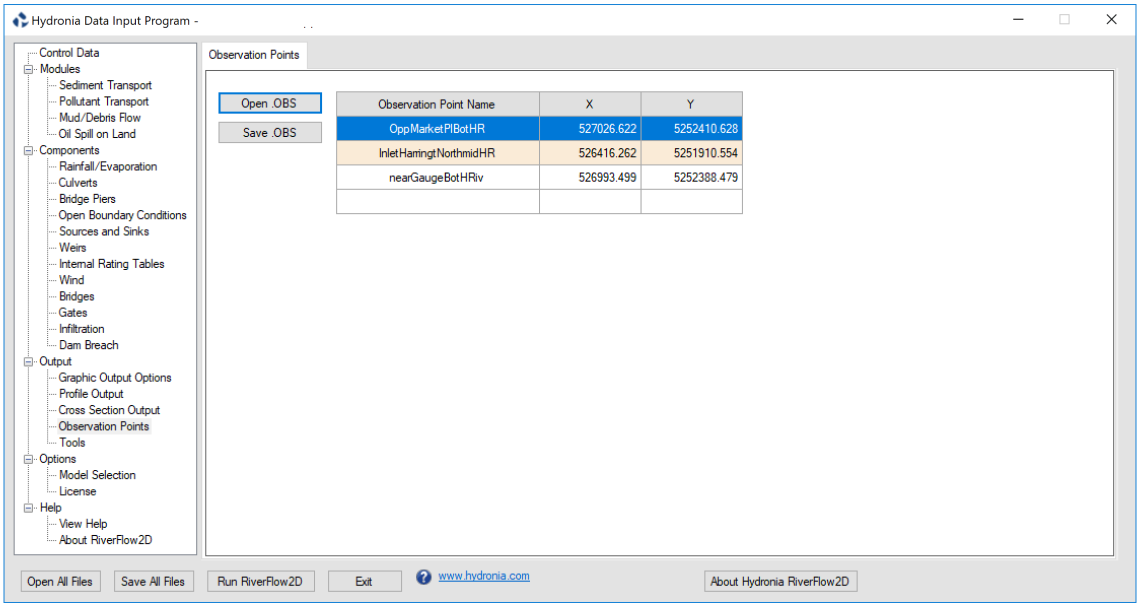


Figure 8.16 – Observation Points Panel.

Table 8.22 – Parameters on the Observation Points Panel.

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

8.15 Tools Panel

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

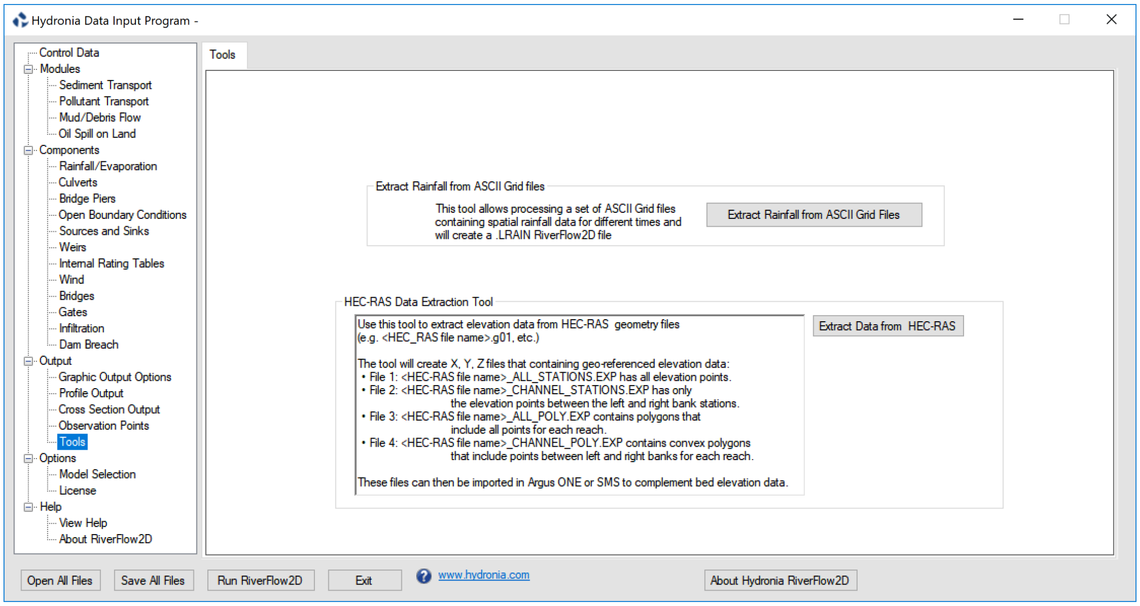


Figure 8.17 – Tools Panel.

8.15.1 Extract Rainfall from ASCII Grid Files Tool

You can use this tool to use ASCII rainfall (e.g. NEXRAD) data files for hydrologic simulations in RiverFlow2D. The program will let you extract data from a set of point rainfall ASCII grid files and to create a `.LRAIN` file in the format readable by RiverFlow2D.

To use the tool, you need to create using a text editor a `.RFC` with 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 `RAINFILEi.ASC` is the ASCII Grid file for the rainfall corresponding to time Ti.

8.15.1.1 Example of a .RFC File

```
-4
0 es_spas1275_001_20040917_0100_utc.asc
1 es_spas1275_002_20040917_0200_utc.asc
2 es_spas1275_003_20040917_0300_utc.asc
3 es_spas1275_004_20040917_0400_utc.asc
```

It is assumed that in the rain ASCII files precipitation values are given in mm or in. Since RiverFlow2D 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 `.RFC` file described above.

If the number of files (NRF) in the first line is positive, the rainfall will be assumed to be given in points, and will be interpolated to each cell. If the number is negative it is assumed that the rain is 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 `.RFC` file created and the `.ASC` files are located in the same folder, use the *Extract Rainfall from ASCII Grid Files Tool* button, select the `.RFC` file and click Open.

Wait for a few moments and enter the name of the `.LRAIN` file. The conversion process will take a few seconds or minutes depending on the number of files and their size.

To use the resulting `.LRAIN` file, you should copy it to the project folder making sure to setting the same file name as that of your project. For instance, if your project files are `Mesh1.dat`, `Mesh1.fed`, then name it as `Mesh1.lrain`.

8.15.2 HEC-RAS Data Extraction Tool

The purpose of this tool is to facilitate migrating existing HEC-RAS projects to RiverFlow2D. The program allows extraction of point elevation data from geo-referenced cross-section from the HEC-RAS one-dimensional model developed by the USACE. The tool reads HEC-RAS geometry files with extension `.g01`, `.g02`, etc., and creates X Y Z files that can be readily imported in 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.

Table 8.23 – Files generated by the HEC-RAS Data Extraction Tool.

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

9 — Input Data File Reference

Data files for non-spatial information required to run the HydroBID Flood. All HydroBID Floodinput 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 HydroBID Floodfile formats. This section explains the input data files, and the parameters included in each file.

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

...\Documents\RiverFlow2D_QGIS\ExampleProjects.

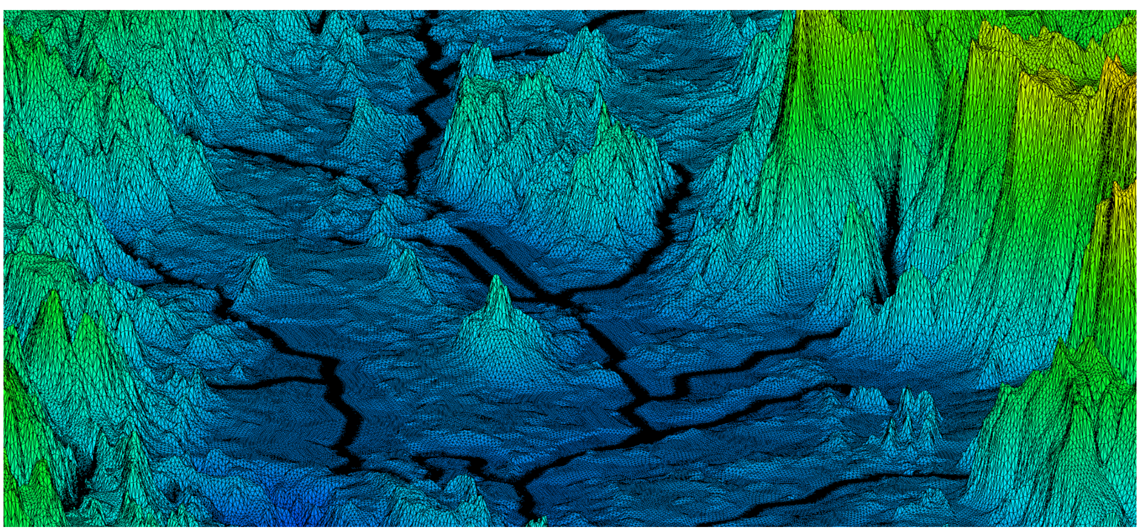


Figure 9.1 – Example of a HydroBID Flood Mesh.

HydroBID Flood data files will share the same name and will use the file extensions listed in the table 9.1 below. For example a run named Run1 will have files as follows: `Run1.DAT`, `Run1.FED`, etc. The following table summarizes the data files used by HydroBID Flood model.

Table 9.1 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 transfered to another computer that will perform the simulations, or to a Virtual Machine on a Cloud Service.

Table 9.1 – List of Input Data Files.

NAME	**FILE EXTENSION**	**DEPENDENCIES**	**CONTENT**
QGIS Plugin			
QGIS project file	`.QSG`	Required when using the HydroBID Flood	This is the project file where QGIS stores all the spatial data used in the project, including the triangular cell mesh.
SPATIAL DATA FILES			
Elevation data	any	Required	Scattered elevation data points.
Triangle-element mesh data	`.FED`	Required	Node coordinates and elevations, triangular mesh topology, boundary condition type and file names, initial water elevations, and Manning's n coefficients.
Mesh boundary nodes	`.TBA`	Internal file	List of external and island boundary nodes. This file is internally generated by RiverFlow2D.
I/O boundary conditions	`.IFL`	Internal file	List of external boundary nodes, inflow and outflow conditions. This file is internally generated by RiverFlow2D.
Boundary condition nodal file	`.OBCP`	Required	List of external boundary conditions. For each boundary, it contains the list of nodes and the associated data file. Note that all files listed within `.OBCP` are required to run the model, and should reside in the same folder. This file is now internally generated by RiverFlow2D, based on the information in the `.FED` file.
CONTROL DATA FILES			
Run control data	`.DAT`, `.DATP`	Required	General run control options, including time step, simulation time, metric or English units, graphical output options, initial conditions, components, etc.
Plot results options	`.PLT`	Optional	Graphical output options.
Observation points data	`.OBS`	Optional	Location of observation points where the model will report time series of results.
Cross section output	`.XSECS`	Optional	List of cross sections where the model will output results. Each cross section is defined by coordinates of its two ending points.
Profile output	`.PROFILES`	Optional	Mesh profile cut where results are desired.
BOUNDARY CONDITIONS DATA FILES			

Continued on next page

Table 9.1 – continued from previous page

NAME	FILE EXTENSION	DEPENDENCIES	CONTENT
Time series or rating table files for inflow or outflow boundary conditions	user defined	Required	Hydrograph, water surface elevations vs. time, etc. The model requires one file for each open boundary condition, except "free" boundary condition types.
Initial concentration of each pollutant	`.CINITIAL`	Required when using the Pollutant Transport module.	Defines the initial concentrations over the mesh.
COMPONENT DATA FILES			
Bridges	`.BRIDGES`	Required when using the Bridges component.	Bridge cross section geometry file is used to compute energy losses.
Culverts	`.CULVERTS`	Required when using the Culvert component.	Culvert location and associated culvert data files.
Dam Breach	`.DAMBREACH`	Required when using the Dam Breach component.	Location and data for the dam breach.
Gates	`.GATES`	Required when using the Gates component.	Gate location and associated gate aperture data files.
Infiltration	`.LINF`	Required when using the Infiltration component.	Infiltration parameters data file.
Internal rating tables	`.IRT`	Required when using the Internal Rating Table component.	Data to impose discharge rating tables along internal boundaries.
Manning's n variable with depth	`.MANNN`	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	`.PIERS`	The Piers component is selected	Bridge pier data used to calculate pier drag forces.
Rainfall/Evaporation	`.LRAIN`	Required when using the Rainfall/Evaporation component.	Time series for rainfall and evaporation.
Sources and sinks	`.SOURCES`	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	`.WEIRS`	Required when using the Weir component.	This file contains weirs polylines and associated weir data.
Wind	`.WIND`	Required when using the Wind component.	This file contains wind specific density and velocity data.
MODULE DATA FILES			
Mud	`.MUD`	Required when using the Mud/Debris Flow module.	Provides the parameters necessary to model mud and debris flow.
Oil Spills	`.OILP`	Required when using the OilFlow2D model.	Provides the parameters necessary to model overland oil spills.

Continued on next page

Table 9.1 – continued from previous page

NAME	FILE EXTENSION	DEPENDENCIES	CONTENT
Pollutant transport	.SOLUTES	Required when using the Pollutant Transport module.	Data for passive or reactive pollutants.
Bed load sediment transport	.SEDB	Required when using the Sediment transport module.	Bed load sediment transport data.
Suspended sediment transport	.SEDS	Required when using the Sediment transport module.	Suspended sediment transport data.

9.1 Run Control Data

9.1.1 Run Control Data File: .DAT

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

Line 1: Internal program version number.
RELEASE

Line 2: Model selector switch.
IMS

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

Line 4: Wet-dry bed method switch.
IWETDRY

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

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

Line 7: Initial conditions and hot start control switches.

IINITIAL IHOTSTART

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

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

Line 10: Mass Balance Reporting Switch.
IMASSBAL

Line 11: Unit system definition switch.
NUNITS

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

Line 13: Initial water surface elevation.
INITIAL_WSE

Line 14: Pollutant transport model switch.
IPOLLUTANT

Line 15: Wind stress switch.
IWIND

Line 16: Mud/Debris flow model switch.
IMUD

Line 17: Number of cores or GPU ID.
IDGPU

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

9.1.1.1 Example of .DAT file

```
201804
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
```

Table 9.2 – Variable Descriptions for the `.DAT` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CFL	R	$(0, 1]$	-	Applies to RiverFlow2D and RiverFlow2D 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 RiverFlow2D.
HMIN	R	-1 or > 0	m/ft	In RiverFlow2D HMIN is the depth limit for dry-wet calculation. If depth is less than HMIN, cell velocity will be set to 0. If HMIN = -1, all cells with depth less than 10^{-6} m will be considered dry.
IADDISP	I	0,1	-	Switch to activate the pollutant transport model. 0: Turn off pollutant transport computations. 1: Apply pollutant transport.
IBRIDGES	I	0,1	-	Switch to activate the Bridges component. 0: Turn off Bridges component. 1: Apply Bridges component. Requires `.BRIDGES` file. See details on the *Bridges* Section of this manual.

Continued on next page

Table 9.2 – continued from previous page

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

Continued on next page

Table 9.2 – continued from previous page

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

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IMASSBAL	I	0,1	-	Mass balance report switch. Used to define when to calculate mass balance and create the `massBalance.out` file. 0: Mass balance IS NOT calculated every time step, AND `massBalance.out` IS NOT created 1: Mass balance IS calculated every time step, AND `massBalance.out` IS created See comment 9.
IMUD	I	0,1	-	Switch to select mud/debris flow model. 0: Do not run mud/debris flow model. 1: Run mud/debris flow model. Requires `.MUD` file. See details on the Mud/Debris Flow Model section of this manual.
IOBS	I	0,1	-	Switch to report time series of results at specified locations defined by coordinates. 0: Do not report on observation points. 1: Report on observation points. Requires `.OBS` file. See details on the *Observation Points* section of this manual.
IPIERS	I	0,1	-	Switch to allow accounting for pier drag force. 0: Do not use pier drag force option. 1: Use pier drag force option. Requires `.PIERS` file. This option may be used if the mesh does not account for the pier geometry. See details on *Bridge Piers Section* of this manual.
IPOLLUTANT	I	0,1	-	Switch to select pollutant transport model. 0: Do not run pollutant transport model. 1: Run pollutant transport model. Requires `.SOLUTES` file. See details on the *Pollutant Transport Model* section of this manual.
IPROFILE	I	0,1	-	Switch to control profile output. 0: No profile results output. 1: Results will be output along a prescribed profile. Requires `.PROFILES` file. See comment 4.

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IRAIN	I	0-4	-	Switch for rainfall and evaporation input. 0: No rainfall modeling. 1: Not used. 2: Rainfall/evaporation. 3: Infiltration. 4: Rainfall/evaporation and Infiltration.
ISED	I	0,1	-	Sediment transport switch. 0: No sediment transport modeling. 1: Sediment transport, mobile bed erosion, and deposition will be simulated. Requires `.SEDS` or `.SEDB` files. See details on the *Sediment Transport* section of this manual.
ISOURCES	I	0,1	-	Switch for sources and sinks. 0: No sources or sinks are present. 1: Sources or sinks are present. Requires `.SOURCES` file. See details on the *Sources* section of this manual.
IWEIRS	I	0,1	-	Weir computation on internal boundary switch. 0: Do not use weir computation on internal boundaries. 1: Use weir computation on internal boundaries. See details on the *Weirs* section of this manual.
IWIND	I	0,1	-	Switch to account for wind stress on the water surface. 0: Do not consider wind stress. 1: Consider wind stress. Requires `.WIND` file. See details on the *Wind Stress* section of this manual.
IXSEC	I	0,1	-	Cross section output switch. 0: No cross section result output. 1: Cross section results will be output to file. Requires `.XSECS` file. See comment 5.

Continued on next page

Table 9.2 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NUNITS	I	0,1	-	Variable to indicate unit system: 0: Metric units. 1: English units.
RELEASE	I	-	-	Release number ID used internally for reference. Should not be modified.
TLIMT	R	> 0	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.

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

9.1.1.2 Comments for the .DAT file

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

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

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

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

3. There are three initial conditions options. If IINITIAL = 0, the initial water elevation will be a constant horizontal surface at the elevation given as INITIAL_WSE. If INITIAL_WSE is = -9999 then the program will assign a constant water elevation equal to the highest bed elevation on the mesh. If IINITIAL = 1, the whole computational mesh will be initially dry,

except at open boundaries where discharge is prescribed and depth $>$ 0 is assumed for the first time step. If IINITIAL = 2, initial water surface elevations are read from the `.FED` data file for each node in the mesh.

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

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

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

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

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

9. 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.

9.2 Mesh Data

9.2.1 Mesh Data File: .FED

This file contains the data that defines the triangular element mesh, and includes node coordinates, connectivity for each triangular element, node elevations, Manning's n coefficients and other parameters. This file is created HydroBID Flood. HydroBID Flood assures that the `.FED` file will be created error free and consistent with the boundary conditions and other mesh parameters. Editing this file outside HydroBID Flood may introduce unexpected errors.

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

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

NELEM lines containing mesh connectivity and element/cell parameters.

IE NODE(IE,1) NODE(IE,2) NODE(IE,3) MANNINGN(IE) ELZB(IE) ELINITWSE(IE) ELMINERODELEV(IE)

9.2.1.1 Example of a .FED file

```
  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.

Table 9.3 – Variable Descriptions for the .FED File.

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

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

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

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

Line 1: Number of nodes on external boundary.
NNODESBOUNDARY

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

9.2.2.1 Example of .IFL file

```
1165
365 1 WSE97out.TXT
367 1 WSE97out.TXT
431 1 WSE97out.TXT
```

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

The format of the `.OBCP` file is as follows

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

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

9.2.2.2 Example of a .OBCP file

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

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

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

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

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCTYPE	I	-	-	Code to indicate type of open boundary. See Table 9.5 and comment 1.
BCFILENAME	S	< 26	-	Boundary condition file name. Should not contain spaces and must have less than 26 characters. See comments 2 and 3.
NOB	I	-	-	Number of open inflow or outflow boundaries.
NODE	I	-	-	Node number.
NNODESBOUNDARYI	I	-	-	Number of nodes on open boundary I.
NNODESBOUNDARY	I	-	-	Total number of nodes on boundary.

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

Table 9.5 – Boundary Condition Types.

BCTYPE	DESCRIPTION
0	Closed impermeable boundary. Slip boundary condition (no normal flow) is imposed. See comment 5.
1	Imposes Water Surface Elevation. An associated boundary condition file must be provided. See comments 2 and 4.
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.
13-18	For future use.
19	Imposes single-valued stage-discharge rating table along an internal polyline. An associated boundary condition file must be provided. See comment 8.
26	Imposes water discharge and sediment discharge time series. An associated boundary condition file must be provided. See comment 9.

9.2.2.3 Comments for the .IFL and .OBCP files

1. HydroBID Flood 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 HydroBID Flood simulation. Theoretically, for subcritical flow it is required to provide at least one condition at inflow boundaries and one for outflow boundaries. For supercritical flow all conditions must be imposed on the inflow boundaries and 'none' on outflow boundaries. Table 9.6 helps determining which conditions to use for most applications.

Table 9.6 – Supercritical Flow Regime Boundary Conditions.

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

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 HydroBID Flood 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.

Loop stage-discharge relationships are not implemented in this HydroBID Flood version.

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

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

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

10. The user must provide a file with the energy slope S_0 for the corresponding boundary. This file will only contain a single value S_0. The model will use S_0, Manning's n, and discharge to create a rating table from which water surface elevations will be imposed as a function of the computed outflow discharge. The rating table is calculated every 0.05 m (0.16 ft.) starting from the lowest bed elevation in the outflow cross section up to 50 m (164 ft.) above the highest bed elevation in the section. If $S_0 = -999$, the model will calculate the average bed

slope perpendicular to the boundary line. Please, note than when letting the model calculate the average bed slope, it uses the elevations on the elements adjacent to the boundary line, which may result in adverse slopes or slopes that do not capture the general trend the reach.

11. This boundary condition is similar to the BCTYPE = 6 for inflow water discharge. However, in this case, instead of converting the discharge into velocities that are imposed on all the inflow nodes; the model creates sources on all the elements adjacent to the boundary line. The condition then can be visualized as if the given discharge enters over the inflow elements. For each time, the model evenly divides the discharge between all the inflow elements. For example if there are Ne inflow elements and the imposed discharge is Qin, each element will receive a discharge equal to Qin/Ne. The water volume will naturally flow away from the inflow depending on the bed slopes, etc. Care must be taken when the inflow boundary elements have lower bed elevations than the surrounding elements. When imposing this condition the user must provide an ASCII file with the discharge time series. See section *Boundary Conditions Data Files* for details on the format for one-variable boundary condition files.

9.2.3 Mesh Boundary Data File: .TBA

`.TBA` file is for internal use by the model and contains the list of boundary nodes in counter-clockwise order for the external boundary polygon and in clockwise order for the internal boundaries. **This file is internally generated by HydroBID Flood** .

Line 1: Start of boundary indicator.
IBOUNDARYID

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

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

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

For each island:

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

Number of nodes in island boundary.
NNODESISLANDBOUNDARY

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

9.2.3.1 Example of a .TBA file

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

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

Table 9.7 – Variable Descriptions for the `.TBA` File.

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

Note: I = Integer variable.

9.2.3.2 Comments for the .TBA file

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

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

9.3 Component data

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

9.3.1 Bridges

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

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

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

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

9.3.2 Bridges Data File: .BRIDGES

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

Line 1: Number of bridges.
NUMBEROFBRIDGES

NUMBEROFBRIDGES lines containing the data for each bridge.

Bridge Id.
BRIDGE_ID

Bridge Cross Section Geometry file name.
BRIDGE_GEOMETRY_FILE

Number of cells pairs along bridge alignment.
NC

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

9.3.2.1 Example of a .BRIDGES file

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

Table 9.8 – Variable Descriptions for the `.BRIDGES` File.

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

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

9.3.3 Bridge Cross Section Geometry Data File

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

Line 1: Number of points defining polylines.
NP

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

The relationship between the four polylines must be as follows:

- For all stations, STATION(I)$\leq$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.

9.3.3.1 Example of the Cross Section Geometry Data File

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

NP ⇨ 23

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

Figure 9.2 – Front view of a bridge cross section.

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

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

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

9.3.4 Bridge Piers Drag Forces File: .PIERS

This option requires the `.PIERS` data file that is internally generated by the model based on the geometrical representation entered in the RiverFlow2D QGIS plugin. The `.PIERS` data file has the following format:

Line 1: Number of piers.
NUMBEROFPIERS

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

9.3.4.1 Example of a .PIERS file

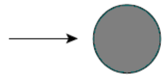
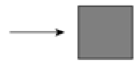
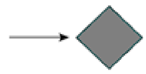
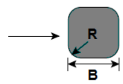
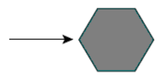
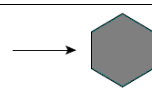
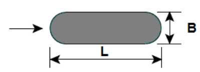
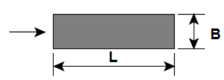
```
124
2042658.82 14214769.48 47.33 19.00 4.00 0.64 P1
2042690.52 14214739.87 46.66 19.00 4.00 0.64 P2
...
2040351.38 14214705.48 0.00 70.00 1.00 0.90 P11
2040375.99 14214622.12 0.00 70.00 1.00 0.90 P12
```

Table 9.10 – Variable Descriptions for the `.PIERS` File.

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

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

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

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

9.3.4.2 Comments for the .PIERS File

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

2. The drag coefficient C_D is related to the drag force though the following formula:

$$F_D = \frac{1}{2} C_D \rho U^2 A_P \tag{9.2}$$

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

$$\tau_{px} = \frac{1}{2} C_D \rho U \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right) \tag{9.3}$$

$$\tau_{py} = \frac{1}{2} C_D \rho V \sqrt{U^2 + V^2} \left(\frac{A_P}{A_e} \right) \tag{9.4}$$

where A_e is the element area.

9.3.5 Culverts Data File: .CULVERTS

The culvert component allows accounting for hydraulic structures that convey flow between two locations. The discharge between the structure inflow and outflow ends will be computed based on a user provided hydraulic structure rating table. The model will determine the flow direction based on the hydraulic conditions on the structure ends.

Line 1: Number of culverts.
NCULVERTS

NCULVERTS groups containing
CulvertID
CulvertType
CulvertFile
X1 Y1 X2 Y2
...

9.3.5.1 Example of a .CULVERTS file

```
2
CulvertA
2
CulvertA.TXT
799550.846 309455.307 799363.544 309031.842
CulvertB
1
CulvertB.TXT
798858.644 309313.609 799153.441 309004.154
```

Table 9.12 – Variable Descriptions for the `.CULVERTS` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CulvertFile	S	< 26	-	Culvert rating table or culvert characteristic file name. See next section for details about the culvert characteristic file. Should not contain spaces and must have less than 26 characters.
CulvertID	S	< 26	-	Culvert name. Should not contain spaces and must have less than 26 characters.
CulvertType	I	$0, 1, 2$	-	Type of culvert. See comments 1 and 2.
NCULVERTS	I	> 0	-	Number of culverts.
X1 Y1 X2 Y2	R	-	m or ft	Coordinates of vertices defining each culvert line.

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

9.3.5.2 Culvert Depth-Discharge Rating table Data Files for CulvertType=0

This format applies to the culvert depth vs. discharge rating table.

Line 1: Number points in data series
NDATA

NDATA lines containing depth and discharge.
DEPTH(I) Q(I)

Where DEPTH(I) is depth corresponding to discharge Q(I).

INVERT_Z1
INVERT_Z2

Where INVERT_Z1 and INVERT_Z2 are the invert elevations for the inlet and outlet respectively.

9.3.5.3 Example of the Culvert Depth-Discharge Rating Table File

The following example shows a depth-discharge rating table for a culvert. NDATA is 7 and there are 7 lines with pairs of depth and corresponding discharge:

```
7
0 0.20
```

```
0.1 1.00
1.00 36.09
2.00 60.00
3.00 84.78
4.00 110.01
100.00 110.02
5.0
1.0
```

Table 9.13 – Variable Descriptions of Culvert Depth-Discharge Data files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
INVERT_Z1	R	> 0	m or ft	Inlet invert elevation. If INVERT_Z1 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet element.
INVERT_Z2	R	> 0	m or ft	Outlet invert elevation. If INVERT_Z2 = -9999, the model makes INVERT_Z2 equal to the average bed elevation of the inlet element.
DEPTH	R	> 0	m or ft	Water depth.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

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

9.3.5.4 Culvert Characteristic Data Files for CulvertType = 1, 2

The culvert characteristic data has the following structure:

Nb
Ke
nc
Kp
M
Cp
Y
m

If CulvertType=1
Hb
Base
Else if CulvertType=2
Dc
INVERT_Z1
INVERT_Z2

9.3.5.5 Example of the culvert characteristic data file

```
 1
 0.5
 0.012
 1
 1
 1.1
 0.6
-0.5
 0.10
 5.0
 1.0
```

This example culvert characteristics data file indicates that the culvert one barrel (Nb =1), Ke=0.4, nc=0.012, Kp=1, cp =1, M =1.1, Y=0.6, m=-0.5, and Dc=0.10, INVERT_Z1=5.0 and INVERT_Z2 = 1.0.

Table 9.14 – Variable Descriptions for the Culvert Characteristic file.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
Nb	I	-	-	Number of identical barrels. The computed discharge for a culvert is multiplied by Nb to obtain the total culvert discharge.
Ke	R	0-1	-	Entrance Loss Coefficient given in Table 9.16.
nc	R	0.01-0.1	-	Culvert Manning's n Coefficient given in Table 9.15.
K'	R	0.1-2.0	-	Inlet Control Coefficient given in Table 9.17.
M	R	0.6-2.0	-	Inlet Control Coefficient given in Table 9.17.
c'	R	0.6-2.0	-	Inlet Control Coefficient given in Table 9.17.
Y	R	0.5-1.0	-	Inlet Control Coefficient given in Table 9.17.

Continued on next page

Table 9.14 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
m	R	0.7,-0.5	-	Inlet form coefficient. m=0.7 for mitered inlets, m=-0.5 for all other inlets.
Hb	R	> 0	m or ft	Barrel Height for box culverts. Only for CulvertType = 1.
Base	R	> 0	m or ft	Barrel Width for box culverts. Only for CulvertType = 1.
Dc	R	> 0	m or ft	Diameter for circular culverts. Only for CulvertType = 2.
INVERT_Z1	R	> 0	m or ft	Inlet invert elevation. If INVERT_Z1 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet.
INVERT_Z2	R	> 0	m or ft	Outlet invert elevation. If INVERT_Z2 = -9999, the model makes INVERT_Z1 equal to the average bed elevation of the inlet element.

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

Table 9.15 – Manning's n roughness coefficients for various culvert materials. Adapted from Froehlich (2003).

Culvert barrel material	Entrance description	Manning's n (n_c)
Concrete	Good joints, smooth walls	0.012
	Projecting from fill, square-cut end	0.015
	Poor joints, rough walls	0.017
Corrugated metal	2-2/3 inch $\times$ 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

Table 9.16 – Entrance loss coefficients K_e. Adapted from Froehlich (2003).

Type of culvert	Entrance description*	Entrance loss coefficient K_e
Concrete pipe	Projecting from fill, grooved end	0.2
	Projecting from fill, square-cut end	0.5
	Headwall or headwall with wingwalls (concrete or cement sandbags)	

Continued on next page

Table 9.16 – continued from previous page

Type of culvert	Entrance description*	Entrance loss coefficient K_e
	Grooved pipe end	0.2
	Square-cut pipe end	0.1
	Rounded pipe end	0.7
	Mitered end that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7
Corrugated metal pipe	Projecting from embankment (no headwall)	0.9
	Headwall with or without wingwalls (concrete or cement sandbags)	0.5
or pipe-arch	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
	With grate	0.7
	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
Reinforced concrete box	Wingwalls at $30°$ to $75°$ to barrel	
	Square-edged at crown	0.4
	Crown edge rounded to radius of 1/12 of barrel dimension	0.2
	Wingwalls at $10°$ to $30°$ to barrel	
	Square-edged at crown	0.5
	Wingwalls parallel to embankment	
	Square-edged at crown	0.7

See Table 9.18 for inlet configurations.

Table 9.17 – Culvert inlet control formula coefficients. Adapted from Froehlich (2003).

Barrel material	Barrel shape	Inlet description*	K’	M	c’	Y
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; grooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45Â° bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7Â° bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30Â° to 75Â° flares; square edge	0.1475	1.0000	1.2385	0.8100

Continued on next page

Table 9.17 – continued from previous page

Barrel material	**Barrel shape**	**Inlet description***	**K’**	**M**	**c’**	**Y**
Concrete	Rectangular	Wingwalls; 90Â° and 15Â° flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0Â° flares ;square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45Â° flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18Â° to 33.7Â° 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; 45Â° bevels	1.5752	0.6670	1.0101	0.8200
Concrete	Rectangular	Headwall; 33.7Â° bevels	1.5466	0.6670	0.8107	0.8650
Concrete	Rectangular	Headwall; 45Â° skew; 3/4 in chamfers	1.6611	0.6670	1.2932	0.7300
Concrete	Rectangular	Headwall; 30Â° skew; 3/4 in chamfers	1.6961	0.6670	1.3672	0.7050
Concrete	Rectangular	Headwall; 15Â° skew; 3/4 in chamfers	1.7343	0.6670	1.4493	0.6800
Concrete	Rectangular	Headwall;10-45Â° skew; 45Â° bevels	1.5848	0.6670	1.0520	0.7500
Concrete	Rectangular	Wingwalls; non-offset 45Â°/flares	1.5816	0.6670	1.0906	0.8030
Concrete	Rectangular	Wingwalls; non-offset 18.4Â°/flares; 3/4 in chamfers	1.5689	0.6670	1.1613	0.8060
Concrete	Rectangular	Wingwalls; non-offset 18.4Â°/flares; 30Â°/skewed barrel	1.5752 0.6670	1.2418	0.7100	
Concrete	Rectangular	Wingwalls; offset 45Â°/flares; beveled top edge	1.5816	0.6670	0.9715	0.8350
Concrete	Rectangular	Wingwalls; offset 33.7Â°/flares; beveled top edge	1.5752	0.6670	0.8107	0.8810
Concrete	Rectangular	Wingwalls; offset 18.4Â°/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

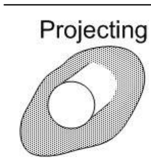
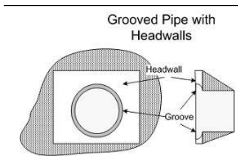
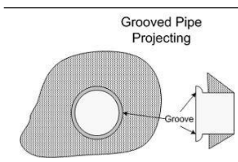
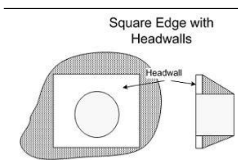
Continued on next page

Table 9.17 – continued from previous page

Barrel material	Barrel shape	Inlet description*	K’	M	c’	Y
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45Â° bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7Â° bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30Â° to75Â° flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90Â° and 15Â° flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0Â° flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45Â° flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18Â° to 33.7Â° flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; Â$\frac{3}{4}$ inch chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45Â° bevels	1.5752	0.6670	1.0101	0.8200

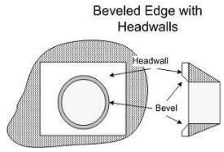
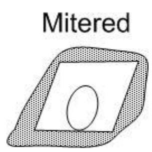
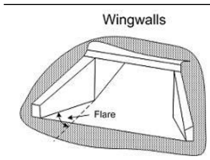
See Table 9.18 for inlet configurations.

Table 9.18 – Culvert inlet configurations. (Adapted from `www.xmswiki.com/xms/`).

Inlet configuration	Description
	End of the culvert barrel projects out of the embankment.
	Grooved pipe for concrete culverts decreases energy losses through the culvert entrance.
	This option is for concrete pipe culverts.
	Square edge with headwall is an entrance condition where the culvert entrance is flush with the headwall.

Continued on next page

Table 9.18 – continued from previous page

Inlet configuration	Description
	'Beveled edges' is a tapered inlet edge that decreases head loss as flow enters the culvert barrel.
	Mitered entrance is when the culvert barrel is cut so it is flush with the embankment slope.
	Wingwalls are used when the culvert is shorter than the embankment and prevents embankment material from falling into the culvert.

9.3.5.6 Comments for the .CULVERTS and culvert characteristics files

1. For CulvertType 0, culvert discharge is computed using a given rating table on the CulvertFile file.

2. For CulvertType 1 and 2, the model will calculate culvert discharge for inlet and outlet control using the FHWA procedures (Norman et al.,1985) that were later restated in dimensionless form by Froehlich (2003).

9.3.6 GATES Data Files: .GATES

This component requires the `.GATES` data file that is internally generated by the model based on the geometrical representation entered in the RiverFlow2D QGIS plugin. The `.GATES` file has the following format:

Line 1: Number of gates.
NUMBEROFGATES

NUMBEROFGATES lines containing the data for each gate.

Gate Id
GATES_ID

Crest elevation height Cd
CRESTELEV GATEHEIGHT CD

Time series of gate aperture
GATE_APERTURES_FILE

Number of cells pairs along gates alignment
NC

NUMBEROFCELLS lines containing pairs of cells numbers along gate alignment

CELL_A[1] CELL B[1]
...
CELL_A[NC] CELL B[NC]

9.3.6.1 Example of 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
```

Table 9.19 – Variable Descriptions for the `.GATES` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CD	R	> 0	-	Gate discharge coefficient.
CRESTELEV	R	> 0	-	Gate crest elevation.
GATE_APERTURES_FILE	S	< 26	-	Gate aperture time series.
GATEHEIGHT	R	> 0	-	Gate height.
GATE_ID	S	< 26	-	Gate ID.
CELL_A[i] CELL B[i]	I	-	-	Cell numbers of cell pairs along gate alignment.
NC	I	> 0	-	Name of pier. Should not contain spaces and must have less than 26 characters.
NUMBEROFGATES	I	> 0	-	Number of cells along the gate alignment.

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

9.3.7 Gate Aperture Time Series File

Line 1: Number of points in time series of gate aperture data.
NPOINTS

NPOINTS lines containing:

Time Aperture.
TIME APERTURE

9.3.7.1 Example of a Gates Aperture Data File

```
3
0 0.0
2 0.5
4 1.0
```

Table 9.20 – Variable Descriptions for the `.GATES` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPOINTS	I	> 1	-	Number of data points in the gate aperture time series.
TIME	R	> 0	h.	Time.
H(I)	R	-	m or ft	Gate aperture for the corresponding time.

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

9.3.8 Internal Rating Table Data File: .IRT

This data file allows modeling complex hydraulic structures inside the modeling domain. The user would enter polylines coincident with mesh nodes and assign a rating table of discharge vs. water surface elevation to the polyline. In other words, the IRT polylines must connect nodes of the triangular element mesh. For each time step, the model will compute the discharge crossing the polyline and find by interpolation the corresponding water surface elevation from the provided rating table. The model will then impose that water surface elevation to all nodes along the polyline. Velocities will be calculated using the standard 2D equations. Therefore, in internal rating table polylines, computed velocities may not necessarily be perpendicular to the IRT polyline.

The file structure is as follows:

Line 1: Number of internal rating table polylines.
IRT_NPL

IRT_NPL line groups containing the IRT polyline ID, the number of vertices defining each polyline, the IRT boundary condition type (always equal to 19 in this version), the Rating Table file name, followed by the list of polyline coordinate vertices as shown:

IRT_ID
IRT_NV IRT_BCTYPE IRT_FILENAME
X_IRT(1) Y_IRT(1)
X_IRT(2) Y_IRT(2)
...
X_IRT(IRT_NV) Y_IRT(IRT_NV)

9.3.8.1 Example of 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_A.DAT`.

Table 9.21 – Variable Descriptions for the `.IRT` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IRT_NPL	I	> 0	-	Number of IRT polylines.
IRT_NV	I	≥ 2	-	Number of points defining each IRT polyline.
IRT_ID	S	< 26	-	Name of IRT. Should not contain spaces and must have less than 26 characters.
IRT_BCTYPE	I	19	-	Boundary condition always equals to 19 in this version corresponding to discharge vs. water surface elevation tables. Future versions will include further options.
X_IRT Y_IRT	R	-	m or ft	Coordinates of vertices defining each IRT polyline. See comment 1.
IRT_FILENAME	S	< 26	-	File name containing internal rating table in the format described as a stage-discharge data file. Should not contain spaces and must have less than 26 characters.

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

9.3.8.2 Comments for the .IRT file

1. IRT polylines should be defined avoiding abrupt direction changes (e.g. 90 degree turns). Polyline alignments as such may create errors in the model algorithm that identifies the nodes that lie over the polyline. Therefore, it is recommended that the IRT follow a more or

less smooth path.

9.3.9 Mud and Debris Flow Data File: .MUD

This file provides the parameters necessary to model mud and debris flow using the RiverFlow2D model.

Line 1: Flow resistance relation.
MF_FRR

Line 2: Yield stress.
MF_YS

Line 3: Bingham viscosity.
MF_BVIS

Line 4: Internal friction angle.
MF_THETA

Line 5: Material density.
MF_DENS

9.3.9.1 Example of a .MUD file

```
1
300.
0.00899
3.5
2200.
```

Table 9.22 – Variable Descriptions for the `.MUD` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
MF_BVIS	I	> 0	Poise or lb/in^2	Bingham viscosity.
MF_DENS	R	> 0	kg/m^3 or lb/ft^3	Material density.
MF_FRR	I	1-8	-	Flow resistance relation (see Table 9.23).
MF_THETA	R	> 0	Degrees	Internal friction angle.
MF_YS	R	> 0	Pa or lb/in^2	Yield stress.

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

9.3.9.2 Comments for the .MUD file

1. The equations that describe the tangential forces generated by the stresses can be lumped into the same mathematical formula despite of having a different nature. In this way several shear stresses formulations can be considered: turbulent stress τ_t, dispersive stress τ_d, Coulomb-type frictional stress τ_f, yield stress τ_y and viscous stress τ_μ. Table 9.23 includes turbulent-collisional, yield stress and laminar flow resistance terms in a quadratic rheological law, where a standard value of $\kappa = 24$ is assumed. In this table is also included the input parameters which are required by its computation.

Table 9.23 – Flow resistance relation.

MF_FFR	Flow Resistance Relation	Flow resistance term	Input parameters
1	Turbulent (default)	$\tau_b = \tau_t$	Manning's n, density
2	Full Bingham	$\tau_b = \tau_0$ with $f_1(\tau_b, \tau_0) = 0$	Bingham viscosity, yield stress, density
3	Simplified Bingham	$\tau_b = 1.5\tau_y + 3\tau_\mu$	Bingham viscosity, yield stress, density
4	Turbulent and Coulomb	$\tau_b = \tau_t + \tau_f$	Manning's n, density, Friction angle
5	Turbulent and Yield	$\tau_b = \tau_t + \tau_y$	Manning's n, density, yield stress
6	Turbulent, Coulomb and Yield	$\tau_b = \tau_t + \min\left(\tau_y, \tau_f\right)$	Manning's n, density, yield stress, friction angle
7	Quadratic	$\tau_b = \tau_t + \tau_y + \frac{\kappa}{8}\tau_\mu$	Manning's n, density, yield stress, viscosity
8	Granular flow	$\tau_b = \tau_f$	Friction angle, density. For this formula, the model internally sets Manning's n = 0 for all cells.

9.3.10 Oil Properties File: .OILP

This file only applies to the OilFlow2D model. This file provides the parameters necessary to model flow of viscous fluids including oil over complex terrain using the OilFlow2D model.

Line 1: Flow resistance relation.
OL_FRR

Line 2: Yield stress. Not used in this release.
OL_YS

Line 3: Fluid viscosity.
OL_BVIS

Line 4: Internal friction angle. Not used in this release.
OL_THETA

Line 5: Oil density.
OL_DENS

Line 6: Temperature time series file.
OL_TEMPTSERIES

Line 7: Temperature - Viscosity - Density table file.
OL_TEMPVISCDENS

9.3.10.1 Example of a .OILP file

```
3
1.
0.00899
1
2200.
Temptseries.TXT
Tempviscdensetable.TXT
```

Table 9.24 – Variable Descriptions for the .OILP File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
OL_BVIS	I	> 0	Pa . s or lb-s/in^2	Fluid viscosity that will be used if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
OL_DENS	R	> 0	kg/m^3 or lb/ft^3	Fluid density that will be used if the lines corresponding to the Temperature-viscosity-density-table file and temperature data file are not provided.
OL_FRR	I	3	-	Flow resistance relation. (see Table 9.25)
OL_TEMPTSERIES	S	≤ 26	-	Temperature time series file.
OL_TEMPVISCDENS	S	≤ 26	-	Temperature-viscosity-density table file. The model will use the temperature for a given time to interpolate in the table the viscosity and density.

Continued on next page

Table 9.24 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
OL_THETA	R	> 0	Degrees	Not used in this release.
OL_YS	R	> 0	Pa or lb/in^2	Yield stress.

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

Table 9.25 – Flow resistance relations for the OilFlow2D model.

OL_FFR	Oil Flow Resistance Relation	Oil Flow resistance term	Input parameters
2	Full Bingham	$\tau_b = \tau_0$ with $f_1(\tau_b, \tau_0) = 0$	Bingham viscosity, yield stress, density
3	Simplified Bingham	$\tau_b = 1.5\tau_y + 3\tau_\mu$	Bingham viscosity, yield stress, density
7	Quadratic	$\tau_b = \tau_t + \tau_y + \frac{\kappa}{8}\tau_\mu$	Manning's n, density, yield stress, viscosity

9.3.10.2 Example of the Temperature Time Series file

```
7
0.0 60
2.0 65
4.0 70
6.0 75
8.0 77
10.0 79
12.0 65
```

In this example there are 7 entries in the Temperature Time series. For the initial time 0.0 hours the temperature is $60°F$, and so on.

9.3.10.3 Example of the Temperature-Viscosity-Density Table file

```
2
50 1.38E-06 58.00
120 6.94E-07 56.00
```

In this example there are 2 lines in the Temperature-Viscosity-Density table file. For $50°F$ the viscosity is 1.38E-06 lb-s/in^2 and the density 50 lb/ft^3.

9.3.11 Pollutant Transport Module Data File: .SOLUTES

The pollutant transport module calculates concentration of passive or reactive pollutants (solutes) based on advection and dispersion. The required data is included in the `.SOLUTES` data file that has the following format:

Line 1: Number of pollutants (solutes)
NPOLLUTANTS_MAX

Line 2: Number of pollutants used in this run
NPOLLUTANTS_USED

Line 3: List of solutes used (all in one line)
I_1 I_2 ... I_NPOLLUTANTS_USED

Line 4: Longitudinal and transversal dispersion coefficients
DISPL DISPT

Line 5...: List of names of pollutants (one in each line)
NAME_I1
NAME_I1
...
NAME_NPOLLUTANT_MAX

Line 6 to NPOLLUTANT_MAX+6: First order reaction rate coefficient matrix K(I,J)
K(1,1) K(1,2) ... K(1,NPOLLUTANT_MAX)
K(2,1) K(2,2) ... K(2,NPOLLUTANT_MAX)
...
K(NPOLLUTANT_MAX,1) K(NPOLLUTANT_MAX,2) ... K(NPOLLUTANT_MAX,NPOLLUTANT_MAX)

9.3.11.1 Example of a .SOLUTES file

```
5
5
1 2 3 4 5
0.1 0.02
NITRATE
AMMONIUM
TEMPERATURE
CHLORINE
```

```
PHOSPHATE
0.2 0.0 0.0 0.0 0.0
0.0 0.1 0.0 0.0 0.0
0.0 0.0 1.1 0.0 0.0
0.0 0.0 0.0 0.2 0.0
0.0 0.0 0.0 0.0 0.5
```

Table 9.26 – Explanation of the example .SOLUTES file.

DATA	DESCRIPTION
5	There are five pollutants
5	Five pollutants will be used in this run
1 2 3 4 5	The active pollutants will be 1 through 5
0.1 0.02	Longitudinal dispersion coefficient is 0.1 and transversal = 0.02.
D	Pollutant No. 1 name is: NITRATE
AMMONIUM	Pollutant No. 2 name is: AMMONIUM
TEMPERATURE	Pollutant No. 3 name is: TEMPERATURE
CHLORINE	Pollutant No. 4 name is: CHLORINE
PHOSPHATE	Pollutant No. 5 name is: PHOSPHATE
0.2 0.0 0.0 0.0 0.0	First order reaction constant for pollutant No. 1 is 0.2
0.0 0.1 0.0 0.0 0.0	First order reaction constant for pollutant No. 2 is 0.1
0.0 0.0 1.1 0.0 0.0	First order reaction constant for pollutant No. 3 is 1.1
0.0 0.0 0.0 0.2 0.0	First order reaction constant for pollutant No. 4 is 0.2
0.0 0.0 0.0 0.0 0.5	First order reaction constant for pollutant No. 5 is 0.2

Table 9.27 – Variable Descriptions for the .SOLUTES File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
DISPL	R	> 0	m^2/s or ft^2/s	Longitudinal dispersion coefficient.
DISPT	R	> 0	m^2/s or ft^2/s	Transversal dispersion coefficient.
K(I,J)	R	-	s^{-1}	First-order reaction constants, where K(I,I) is the reaction constant for pollutant I and K(I,J) the reaction constant of pollutant I with pollutant J.
NPOLLUTANTS_MAX	I	$0, 1, 2$	-	Maximum number of pollutants.

Continued on next page

Table 9.27 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPOLLUTANTS_USED	I	> 0	-	Number of pollutants used in the present run. Should be equal to NPOLLUTANTS_MAX

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

9.3.12 Pollutant Transport Module Initial Concentration Data File: .CINITIAL

The initial concentration data file is created by when the dataset with the initial concentrations is exported from the RiverFlow2D QGIS plugin based on the data entered as Initial Concentrations.

Each column in this file corresponds to each of the pollutants indicated in the corresponding `.SOLUTES` file. Each line corresponds to the initial concentrations for each cell. Therefore the `.CINITIAL` file must have NPOLLUTANTS_MAX columns and NELEM lines.

9.3.13 Manning's n Variable with Depth Data File: .MANNN

This file is created by the RiverFlow2D 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])

9.3.13.1 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 `Manning1.TXT` and the polygon is defined by four vertices.

Table 9.28 – Variable Descriptions for the `.MANNN` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NNZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NNZONES	I	-	-	Number of zones.
MANNNFILE	S	$\leq$ 26	-	Manning's n file. See comment 1.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

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

9.3.13.2 Comments for the .MANNN file

1. The spatial distribution of Manning's n variable with depth is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygon (complementary area) would be assigned the default Manning's n file.

9.3.13.3 Manning's n variable with depth data file

Line 1: Number of points in Manning's n file.

NPRE

ND lines containing:

DEPTH MANNINGS N

9.3.13.4 Example of a Manning's variable with depth data file

```
3
0. 0.1
0.3 0.1
1.0 0.03
```

Table 9.29 – Variable Descriptions for the Manning's n variable with Depth Data File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
DEPTH	R	≥ 0	m or ft	Flow depth. See comment 1.
MANNINGS N	R	≥ 0	-	Manning's n corresponding to DEPTH. See comment 1.
ND	I	-	-	Number values in file.

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

9.3.13.5 Comments for the Mannign's n variable with depth data file

1. To calculate the Manning's n over the mesh, the model will first identify the polygon over each cell and then will use the interpolated n value for cell depth from the table corresponding to the polygon. In the example above, for all depth between 0.3 and 1, Manning's n will be obtained by linear interpolation between 0.1 and 0.03.

2. The user should provide a `DefaultManningsn.DAT` file in the project folder and the program will apply the data contained in that file to the complementary area to the polygons provided. If the `DefaultManningsn.DAT` does not exist, the model will apply a default value of 0.035 to the areas not covered by Manning's n polygons.

9.3.14 Rainfall And Evaporation Data File: .LRAIN

Use this file to enter spatially distributed and time varying rainfall and evaporation data. The model assumes that the rainfall and evaporation can vary over the modeling area.

Line 1: Number of zones defined by polygons where rainfall time series are defined.
NRZONES

NRZONES group of lines containing hyetograph and evaporation data file for each zone
RAINEVFILE[i]

Number of vertices of polygon i
NPZONE[i]

List of NPZONE[i] vertex coordinates
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.14.1 Example of 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 `hyeto1.TXT` and the polygon is defined by four vertices.

Table 9.30 – Variable Descriptions for the `.LRAIN` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NRZONES	I	-	-	Number of zones.
RAINEVFILE	S	$\leq$ 26	-	Rainfall intensity. See comment 1.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

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

9.3.14.2 Comments for the .LRAIN file

1. The spatial distribution of rainfall and evaporation is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons would have no rainfall or evaporation imposed onto the mesh.

9.3.14.3 Hyetograph and Evaporation data file

Line 1: Number of points in time series of rainfall and evaporation.
NPRE

NPRE lines containing:

Time Daily Rainfall, Daily Evaporation.
TIME RAINFALL EVAPORATION

9.3.14.4 Example of a Hyetograph and Evaporation data file

```
 0.  0.0 1.0
24   4.0 2.0
48  12.0 3.0
```

Table 9.31 – Variable Descriptions for the Hyetograph and Evaporation Data File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
EVAPORATION	R	≥ 0	mm/hr or in/hr	Evaporation intensity. See comment 1.
NPRE	I	-	-	Number of times in rainfall and evaporation time series.
RAINFALL	R	≥ 0	mm/hr or in/hr	Rainfall intensity. See comment 1.
TIME	R	> 0	hours	Time interval

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

9.3.14.5 Comments for the Hyetograph and Evaporation data file

1. To calculate the rainfall/evaporation over the mesh, the model will use the interpolated rainfall and evaporation intensities between each time interval. For instance in the example above, for all times between 0 and 24 hours the rainfall will be determined by linear interpo-

lation between 0 mm/hr and 4 mm/hr, and so on for evaporation.

2. If the user has a `DefaultRainEvap.DAT` file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.15 Infiltration Data File: .LINF

Use this file to enter spatially distributed infiltration parameters.

Line 1: Number of zones defined by polygons where infiltration parameters are defined.
NIZONES

NIZONES group of lines containing:

Infiltration data file for each zone
INFILFILE

Number of vertices of polygon i
NPZONE[i]

List of NPZONE[i] vertex coordinates
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.15.1 Example of 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 `inf1.inf` and the polygon is defined by four vertices.

Table 9.32 – Variable Descriptions for the `.LINF` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NIZONES	I	-	-	Number of zones. See Comments 1 and 2.
INFILFILE	S	≤ 26	-	Infiltration parameter file.
X(i) Y(i)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

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

9.3.15.2 Comments for the .LINF file

1. The spatial distribution of infiltration parameters is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons would have no infiltration loss calculated.
2. Each polygon can have a different infiltration method assigned.
3. If the user has a `DefaultInfiltration.DAT` file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.15.3 Infiltration parameters data file

Line 1: Model to calculate infiltration.
INFILMODEL

Line 2: Number of infiltration parameters.
NIPARAM

If INFILMODEL = 1: Horton method then:
Line 3: **K** $\mathbf{f_c}$ $\mathbf{f_0}$

If INFILMODEL = 2: Green and Ampt method then:
Line 3: **KH PSI DELTATHETA**

If INFILMODEL = 3: SCS-CN method then:
Line 3: **CN POTRETCONST AMC**

9.3.15.4 Example of a Infiltration parameter data file

```
1
```

```
3
8.3E-04 3.47E-06 2.22E-5
```

In this example the infiltration loss method is set to 1 corresponding to the Horton model. There are 3 parameters as follows: K = 8.3E-04, f_c = 3.47E-06 and f_0 = 2.22E-5.

Table 9.33 – Variable Descriptions for the Infiltration Parameter File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
AMC	I	>0	1, 2, 3	Antecedent Moisture Content (AMC). Represents the preceding relative moisture of the soil prior to the storm event (Chow et al. 1988) . Allows accounting for variation of CN for different storm events, or initial soil moisture for a given event using Eqs. 7.41 and 7.42. See possible AMC values in Table 9.34.
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.

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

Table 9.34 – Antecedent Moisture Content groups (adapted from Mishra et al. (2003)

SOIL AMC	Total 5-day rainfall (dormant season)	Total 5-day rainfall (growing season)
1	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

9.3.16 RiverFlow2D Sediment Transport Data Files: .SEDS and .SEDB

This file provides the parameters necessary to model sediment transport using the RiverFlow2D Plus model.

9.3.16.1 .SEDS file for suspended sediment data

The `.SEDS` file provides the data for the suspended sediment transport model.

Line 1: Suspended sediment activation switch.
ISSACT

Line 2: Number of suspended sediment fractions.
NSSNFRAC

Line 3: Equilibrium concentration formula.
ISSTF

Line 4: Sediment density for each fraction.
SSDEN(1) ... SSDEN(NSSNFRAC)

Line 5: Initial suspended sediment concentration for each fraction.
INICON(1) ... INICON(NSSNFRAC)

Line 6: For future use.
DUMMY1(1) ... DUMMY1(NSSNFRAC)

Line 7: Suspended sediment D50 size for each fraction.
D50(1) ... D50(NSSNFRAC)

Line 8: For future use.
DUMMY2(1) ... DUMMY2(NSSNFRAC)

Line 9: Porosity for each fraction.
SSPOR(1) ... SSPOR(NSSNFRAC)

Line 10: Critical Shield Stress for each fraction.
THETAC(1) ... THETAC(NSSNFRAC)

Line 11: Friction angle for each fraction.
FRICANG(1) ... FRICANG(NSSNFRAC)

Line 12: Equilibrium concentration formula factor for each fraction.
SSTFACT(1) ... SSTFACT(NSSNFRAC)

Line 13: Settling velocity formula.
ISETFOR

Line 14: Settling velocity formula factor.
SETFORFACT

9.3.16.2 Example of a .SEDS file

```
1
2
1
165.00 165.00
0.01 0.01
0.003937008 0.003937008
0.0039370 0.0039370
0.003937008 0.003937008
0.40 0.40
0.047 0.047
35.00 35.00
1.00 1.00
1
1
```

Table 9.35 – Variable Descriptions for the .SEDS File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
INICON	R	0,0.3	-	Initial volumetric sediment concentration. See comment 1.
ISSACT	I	0,1	-	Suspended sediment activation switch. If ISSACT = 1, the model will compute suspended sediment transport. If ISSACT = 0 suspended sediment transport will not be calculated.
NSSNFRAC	I	1-10	-	Number of suspended sediment fractions.

Continued on next page

Table 9.35 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ISSTF	I	1,2	-	Equilibrium concentration formula. This code indicates the formula according to this list: 1: Bagnold (1966) 2: Van Rijn (1984a)
DUMMY1	R	> 0	-	Not used in this release, but must be present.
D50	R	> 0	mm or in	Sediment median size. 50% of the sediment is finer than D50. See comment 1.
DUMMY2	R	> 0	-	Not used in this release, but must be present.
SSPOR	R	0.3-0.6	-	Porosity. See comment 1.
THETAC	R	0.03-0.06	-	Critical Shield Stress. See comment 1.
FRICANG	R	5-45	-	Friction angle. See comment 1.
ISETFOR	I	0-9	-	Settling velocity formula. It is a unique formula for all fractions. This code indicates the formula according to this list: 1: Rubey (1983) 2: Zhang (1961) 3: Zanke (1977) 4: Van Rijn (1984a) 5: Raudkivi (1990) 6: Julien (1998) 7: Cheng (1997) 8: Jimenez-Madsen (2003) 9: Wu-Wong (2006)
SETFORFACT	R	-	-	Settling velocity formula factor. This factor multiplies the settling velocity calculated by the formula selected in ISETFOR. It's a factor that may be used for calibrating the model.
SSDEN	R	-	kg/m^3 or lb/ft^3	Suspended sediment density. See comment 1.

Continued on next page

Table 9.35 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
SSTFACT	R	-	-	Equilibrium concentration formula factor for each fraction. This factor multiplies the equilibrium concentration formula ISSTF. It's a factor that may be used for calibrating the model. See comment 1.

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

9.3.16.3 Comments for the .SEDS file

1. There should be one value for each sediment fraction up to NSSNFRAC.

9.3.16.4 .SEDB file for bed load transport data

The `.SEDB` file provides the data for the bed load transport model.

Line 1: Bed load sediment transport activation switch.
IBLACT

Line 2: Number of bed load sediment fractions.
NBLNFRAC

Line 3: Sediment transport formula.
IBLTF

Line 4: Sediment density for each fraction.
BLDEN(1) ... BLDEN (NBLNFRAC)

Line 5: Sediment D30 size for each fraction.
D30(1) ... D30(NBLNFRAC)

Line 6: Sediment D50 size for each fraction.
D50(1) ... D50(NBLNFRAC)

Line 7: Sediment D90 size for each fraction.
D90(1) ... D90(NBLNFRAC)

Line 8: Porosity for each fraction.
BLPOR(1) ... BLPOR(NSSNFRAC)

Line 9: Critical Shield Stress for each fraction.
THETAC (1) ... THETAC (NBLNFRAC)

Line 10: Friction angle for each fraction.
FRICANG (1) ... FRICANG (NBLNFRAC)

Line 11: Fractions on the bed.
BEDFRACT (1) ... BEDFRACT (NBLNFRAC)

Line 12: Transport formula factor for each fraction.
BLFORFACT(1) ... BLFORFACT (NBLNFRAC)

Line 13: Coupled or uncoupled computation switch.
ICOUPLED

9.3.16.5 Example of a .SEDB file

```
1
2
2
165.0 165.0
0.0039370 0.0039370
0.0039000 0.0039370
0.0039370 0.0039370
0.40 0.40
0.047 0.047
35.00 35.00
0.50 0.50
1.00 1.00
0
```

Table 9.36 – Variable Descriptions for the .SEDB File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IBLACT	I	0,1	-	Bed load transport activation switch. If IBLACT = 1, the model will compute bed load sediment transport. If IBLACT = 0 bed load transport will not be calculated.

Continued on next page

Table 9.36 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ICOUPLED	I	0,1	-	Bed load transport activation switch. If ICOUPLED = 0 the model will run in coupled mode, where the bed load transport will be computed together with the hydrodynamic model for each time step. If ICOUPLED = 1 the model will run the bed load transport uncoupled from the hydrodynamic model.
NBLNFRAC	I	1–10	-	Number of sediment fractions.
IBLTF	I	1–10	-	Coupled or uncoupled computation switch. This code indicates the formula according to this list: 1: Meyer-Peter & Muller (1948) 2: Ashida (1972) 3: Engelund (1976) 4: Fernandez (1976) 5: Parker fit to Einstein (1979) 6: Smart (1984) 7: Nielsen (1992) 8: Wong 1 (2003) 9: Wong 2 (2003) 10: Camenen-Larson (2005)
D30	R	> 0	m or in	Sediment D30 size. 30% of the sediment is finer than D30. Only used for Smart Formula. See comment 1.
D50	R	> 0	m or in	Sediment median size. 50% of the sediment is finer than D50. See comment 2.
D90	R	> 0	m or in	Sediment D90 size. 90% of the sediment is finer than D90. Only used for Smart Formula. See comment 1.
BLPOR	R	0.3–0.6	-	Porosity. See comment 1.
THETAC	R	0.03–0.06	-	Critical Shields Stress. See comment 3.
FRICANG	R	5–45	-	Friction angle. See comment 1.
BEDFRACT	R	-	-	Sediment fraction. The sum of all fractions should add to 1. See comment 1.

Continued on next page

Table 9.36 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BLDEN	R	-	kg/m^3 or lb/ft^3	Sediment density. See comment 1.
BLFORFACT	R	-	-	Transport formula factor for each fraction. This factor multiplies the result of the transport formula selected (IBLTF). It's a factor that may be used for calibrating the model. See comment 1.

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

9.3.16.6 Comments for the .SEDB file

1. There should be one value for each sediment fraction up to NBLNFRAC.
2. Characteristic diameter for all sediment transport formulas.
3. The default critical Shields stress is 0.047.

9.3.17 Sources and Sinks Data File: .SOURCES

Use this file to enter data to simulate point inflows or outflows at any location. This feature is typically used when modeling intakes (outflow) or point inflows. The user may provide time varying hydrographs that will be applied to each point.

Line 1: Number of source and sink points.
NSOURCES

NSOURCES groups of lines containing source/sink point identification text, name of the file containing the discharge time series or rating table, and the coordinates of the point as follows:

SOURCEID
SOURCETYPE
ISFILENAME
X_S(I) Y_S(I)
...

9.3.17.1 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 `Drain.TXT` 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 `Discharge.TXT`.

Table 9.37 – Variable Descriptions for the `.SOURCES` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NSOURCES	I	> 0	-	Number of source or sink points.
ISFILENAME	S	-	-	Name of file containing the time series of each point source or sink. Must not contain blank spaces. See comments 1 and 2.
SOURCEID	S	< 26	-	Name of point source or sink. Should have less than 26 characters and must not contain blank spaces.
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.

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

9.3.17.2 Comments for the .SOURCES File

1. The file name is arbitrary but must not contain blank spaces. The file format is the same as the *one variable boundary condition* file described in Section 9.6.1.
2. To model inflows use positive discharge values, and to model outflows use negative values.

9.3.18 Weirs Data File: .WEIRS

This data file allows using weir calculations along user defined polylines representing road or weir overtopping. The user selects the weir coefficient associated with each weir and the model will determine the nodes on each polyline and the discharge across each pair of nodes based on the weir formulae and methods described on Hydraulics of Bridge Waterways FHWA, 1978 (see comment 1). The present version allows defining a variable crest elevation along the weir polyline.

Line 1: Number of weir polylines.
NWEIRS

NWEIRS group of lines including weir ID, number of vertices defining each weir polyline, the weir coefficient followed by the coordinates each vertex as shown:

WEIR_ID
NV CF WRCRESTELEV
X_W(1) Y_W(1) WRCREST(1)
X_W(2) Y_W(2) WRCREST(2)
...
X_W(NV) Y_W(NV) WRCREST(NV)

9.3.18.1 Example of a `.WEIRS` file

```
2
WEIR_A
4 3.0 -999
799429.362 308905.287 200.
799833.895 308354.857 201.
799986.424 307738.111 202.
799847.158 307141.259 203.
WEIR_B
4 3.04 -999
799482.440 309453.678 203.5
799135.525 309118.164 204.0
798914.020 309269.634 204.9
798787.701 309467.583 205.0
```

This file indicates that there are 2 weirs. The first one is named WEIR_A and is defined by a polyline with 4 vertices. Weir discharge coefficient is equal to 3.0.

Table 9.38 – Variable Descriptions for the `.WEIRS` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
CF	R	> 0	-	Weir coefficient. See comment 1.
NWEIRS	I	> 0	-	Number of weir polylines.
NV	I	≥ 2	-	Number of points defining each weir polyline.
WEIR_ID	S	< 26	-	Name of weir. Should have less than 26 characters and must not contain blank spaces.
WRCRESTELEV	R	-	m of ft	Weir crest elevation for all the weir. If WRCRESTELEV = -999 a weir elevation is provided for each weir polyline vertex.
WRCREST(I)	R	-	m of ft	Weir crest elevation for vertex I.
X_W(I) Y_W(I)	R	-	m of ft	Coordinates of vertices defining each weir polyline. See comment 2.

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

9.3.18.2 Comments for the `.WEIRS` File

1. Weir discharge is computed between pairs of nodes along the polyline based on the following formula:

$$Q = C_f L H^{3/2} \tag{9.5}$$

   where L is the distance between nodes, H is the total head upstream of the polyline segment and C_f is the discharge coefficient. The model checks for submergence and it occurs C_f will be corrected according to the correction factor defined by (FHWA, 2001).

2. Weir polylines should be defined avoiding abrupt direction changes (e.g. $\geq$ 90 degree turns), because such angles may create errors in the algorithm that identifies the nodes that lie over the polyline.

9.3.19 Wind Data File: .WEIRS

Use this file to enter spatially distributed and time varying wind velocity data. The model assumes that the wind velocity can vary over the modeling area. The user should provide a set of polygons and a time series of velocities for each polygon.

Line 1: Number of zones defined by polygons where wind velocity time series are defined.
NWZONES

Line 2: Wind stress coefficient.
CD

Line 3: Air density.
AIRDENSITY

NWZONES group of lines containing hyetograph and evaporation data file for each zone.
WINDFILE

Number of vertices of polygon i.
NPZONE[i]

List of NPZONE[i] vertex coordinates.
X(1) Y(1)
...
X(NPZONE[i]) Y(NPZONE[i])

9.3.19.1 Example of a `.WEIRS` file

```
2
0.009
1.225
Wind1.TXT
4
25.0 25.0
25.0 75.0
75.0 75.0
75.0 25.0
Wind2.TXT
4
25.0 125.0
```

```
25.0 175.0
75.0 175.0
75.0 125.0
```

In this example, there are two polygons. The C_d coefficient is set to 0.009 and the wind density to 1.225 kg/m^3. The wind velocity file for the first polygon is `Wind1.TXT` and the polygon is defined by four vertices.

Table 9.39 – Variable Descriptions for the `.WIND` File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
AIRDENSITY	R	≥ 0	-	Air density. Always given in metric units. Default = 1.225.
CD	R	≥ 0	-	Wind stress coefficient. Always given in metric units. Default = 0.009.
NPZONE[i]	I	≥ 1	-	Number of vertices defining zone i.
NWZONES	I	-	-	Number of zones.
WINDFILE	S	$\leq$ 26	-	Wind velocity vector time series file. See Comment 1.
X(I) Y(I)	R	> 0	m or ft	Vertex coordinates of the polygon defining Zone i.

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

9.3.19.2 Comments for the `.WEIRS` File

1. The spatial distribution of wind is given as a number of non-overlapping polygons that would cover or not the mesh area. Zones not covered by any polygons will be considered as having no wind stress.

2. If the user has a `DefaultWind.DAT` file in the project folder, the program will apply the data contained in that file to the complementary area to the polygons provided.

9.3.19.3 Wind Velocity Data File

Line 1: Number of points in time series of wind velocity data.
NPOINTS

NPOINTS lines containing:

Time Wind velocity component in X and Y directions.
TIME UX UY

9.3.19.4 Example of a Wind Velocity and Data File

```
3
0. 0.0 0.0
24 4.0 -3.0
48 4.0 -3.0
```

Table 9.40 – Variable Descriptions for the Wind Velocity File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NPOINTS	I	> 1	-	Number of data points in the wind velocity time series.
TIME	R	> 0	h	Time.
UX(I) UY(I)	R	-	m/s or ft/s	Wind velocity components in x and y directions.

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

9.4 Output control data

9.4.1 Observation Points Data File: .OBS

This file contains data to allow the model reporting time series of results at user specified locations based on point coordinates. The user would indicate the number of observation points and then the list of point coordinates.

Line 1: Number of observation points.
NOBSPOINTS

NOBSPOINTS groups of lines containing the observation point ID, and coordinate of each point:
ObsID
X_OP(I) Y_OP(I)
...

9.4.1.1 Example of a .OBS file

```
3
PointA
798798.380 309627.950
PointB
799146.926 309430.876
PointC
799721.8608 309041.615
```

This .OBS file has three points. The first point is named PointA and has coordinates: X=798798.380 Y=309627.950.

Table 9.41 – Variable Descriptions for the .OBS File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ObsID	S	< 26	-	Name of observation point. Should have less than 26 characters and must not contain blank spaces.
NOBSPOINTS	I	> 0	-	Number of observation points.
X_OP Y_OP	R	-	m or ft	Coordinates of point.

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

9.4.2 Graphical Output Control Data File: .PLT

This file provides parameters to control graphic output options for plots generated while the model is running.

Line 1: Plot control variables.
IGRAPHCODE COLORSCHEME IAXES IDXF IGRAPHFILES IVSF

Line 2: Velocity vector scale multiplier.
SF_MULT

Line 3: Coordinates for plot window.
XMING XMAXG YMING YMAXG

Line 4: Limits of plotted variable.
MINVARG MAXVARG

Line 5: Maximum velocity to plot.
MAXVELOC

Line 6: Transparency.
USEBACKIMAGE

Line 7: Transparency.
TRANSP

Line 8: Background aerial image.
IMAGEFILE

Line 9: Background aerial image world file.
IMAGEWF

9.4.2.1 Example of the .PLT file

```
100 5 1 0 1 0
5
0 0 0 0
0 0
7
1
0.6
C:\Projects\Example\Aerial.gif
C:\Projects\Example\Aerial.gwf
```

Table 9.42 – Variable Descriptions for the `.PLT` File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
COLORSCHEME	I	1-6	-	Variable to select plot option. See comment 1.
IAXES	I	0,1	-	Switch to control weather to plot axes. 0: Do not plot X and Y axes. 1: Plot X and Y axes.
IDXF	I	0,1	-	Switch to control velocity field output in DXF CAD format. 0: Do not output DXF mesh and velocity field. 1: Create mesh and velocity field DXF files for each output time.

Continued on next page

Table 9.42 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IGRAPHCODE	I	100, 101, 102, 103, 110, 201, 202, 203, 204, 600-610	-	Parameter to indicate the plot type to display while the program is running. 100: Plot velocity field using vectors in black. 101: Plot velocity field using vectors in black over depths. 102: Plot velocity field over bed elevations. 103: Plot water elevations. 110: Plot velocity field using vectors in color. 201: Plot depths. 202: Plot bed elevations. 203: Plot velocity field over water elevations. 204: Plot bed elevation changes. 600-610: Plot suspended sediment or pollutant concentrations.
IGRAPHFILES	I	0,1	-	Variable to control whether to output graphic files. 0: Do not output graphic files. 1: Output graphic files.
IMAGEFILE	S	-	-	Name of aerial image file including path and extension. Supported formats include `.BMP`, `.GIF`, `.PNG` and `.TIFF`. Other graphic file formats will be included in forthcoming releases.
IMAGEWF	S	-	-	Name of aerial image world file including path and extension. World file format should follow ESRI specifications. See comment 2.
IVSF	I	0,1	-	Switch to control velocity field output in a file that allows creating shapefiles. 0: Do not output SF velocity field. 1: Create velocity field SF files for each output time. See comment 3.
MAXVELOC	R	-	m/s or ft/s	Use this variable to control the maximum velocity displayed in vector plots. If MAXVELOC = 0, the whole velocity range will be plotted. If MAXVELOC $>$ 0, it will define the maximum velocity to be displayed.

Continued on next page

Table 9.42 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
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.

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

9.4.2.2 Comments for the .PLT file

1. COLORSCHEME defines the color palette that will be used for all plots. The available palettes are shown in this figure:

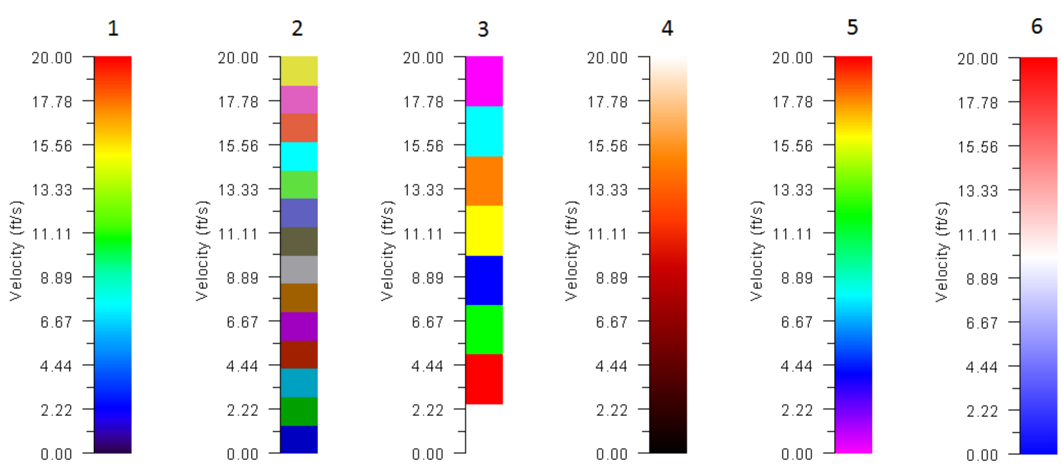


Figure 9.3 – Color palettes.

2. The generic meaning of world file parameters are:

Line 1: A: pixel size in the x-direction in map units/pixel

Line 2: D: rotation about y-axis (ignored in this version)

Line 3: B: rotation about x-axis (ignored in this version)

Line 4: E: pixel size in the y-direction in map units, almost always negative

Line 5: C: x-coordinate of the center of the upper left pixel

Line 6: F: y-coordinate of the center of the upper left pixel.

Example:

```
2.05
0.00
0.00
-2.05
795944.99
310049.73
```

In this example, 2.05 is the pixel size in x-direction, rotation in x and y axes is 0.00, pixel size in y direction is 2.05 (shown in negative), x-coordinate of upper left pixel is 795944.99 and y-coordinate of upper left pixel is 310049.73.

The following table indicates the supported image formats and their corresponding world file extensions.

Table 9.43 – Supported image formats and their corresponding world file extensions.

IMAGE FILE FORMAT	WORLD FILE EXTENSION
.BMP	.BMPW, .BPW
.GIF	.GFW, .GIFW, .WLD
.PNG	.PGW, .PNGW, .WLD
.TIF, .TIFF	.TFW, .WLD

9.4.3 Data for Profile Result Output: .PROFILES

Use this file to provide profiles (polylines) along which results will be generated.

Line 1: Number of profiles.
NPROFILES

NPROFILES group of files including: Profile ID, number of vertices in profile I, the number of intervals to divide each profile, and coordinates for each vertex in polyline.
PROFILEID
NVERTICES_PR(I) ND_PR
X_PRF(I), Y_PRF(I)
...

9.4.3.1 Example of 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.

Table 9.44 – Variable Descriptions for the .PROFILES File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ND_PR	I	> 2	-	Intervals to divide each profile sub-segment between vertices. Results will be reported at each interval.
NPROFILES	I	> 0	-	Number of profiles.
NVERTICES_PR(I)	I	> 1	-	Number of vertices in each profile.
PROFILEID	S	< 26	-	Profile name. Should have less than 26 characters and must not contain blank spaces.

Continued on next page

Table 9.44 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
X_PRF(I,J), Y_PRF(I,J)	R	-	m or ft	Coordinates of each vertex J in profile I.

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

9.4.4 Cross Section Data for Result Output File: .XSECS

Cross sections are used to output numeric results at user defined lines on the mesh.

Line 1: Number of cross sections.
NCROSS_SECTIONS

NCROSS_SECTIONS groups of lines containing the cross section ID, the number of vertices defining the cross section (always equal to 2), the number of intervals to divide the cross section and the list of coordinates of initial and final point in cross section:

XSECID
NPXSEC ND_CS
X1_CS(I) Y1_CS(I)
X2_CS(I) Y2_CS(I)

9.4.4.1 Example of 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 `.XSECS` file indicates there are 3 cross sections. The first one has ID = CrossSectionA and will be divided in 40 segments.

Table 9.45 – Variable Descriptions for the .XSECS File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NCROSS_SECTIONS	I	> 0	-	Number of cross sections.
ND_CS	I	> 2	-	Cross section will be divided in ND_CS segments. Results will be reported at each segment. See comment 1.
NPXSEC	I	2	-	Number of points defining cross section. In the present version only the two extreme points are allowed to define the cross section, therefore this value should always be 2.
X1_CS, Y1_CS, X2_CS, Y2_CS	R	-	m or ft	Coordinates of initial and ending point of each cross section.
XSECID	S	< 26	-	Cross section name. Should have less than 26 characters and must not contain blank spaces.

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

9.4.4.2 Comments for the .XSECS File

1. The model will cut the mesh using the cross section line and extract results at the division points. If ND_CS is too small, the program may not capture anything in between the divisions, and the computed cross section discharges may have big errors.

9.5 Elevation data

9.5.1 X Y Z data with header

These files contain scattered data in the format suitable to import it in a text editor or spreadsheet program. For example the *BedElevations* data layer. It usually has .EXP extension, but can have any other file extension provided that the format is as described herein. Each point is identified by its X and Y coordinates and the elevation value for that coordinate.

Line 1: Number of points and number of parameters per point (header)
NUMBER_OF_DATA_POINTS NUMBER_OF_PARAMETERS

NUMBER_OF_DATA_POINTS lines with X, Y and parameters data.
X(POINT) Y(POINT) P1(POINT) P2(POINT) ... PN(POINT)

9.5.1.1 Example of an .EXP File

```
11086 1
798439.73 306063.87 160.00
798477.04 309506.95 201.10
798489.45 309522.30 200.93
798498.09 306222.29 162.00
798504.45 305915.63 160.00
798511.71 306075.55 161.00
798516.09 309412.73 201.74
798517.37 309592.42 163.14
...
```

In this example .EXP file, there are 11086 elevation data points, one parameter per point (the elevation for each point).

Table 9.46 – Variable Descriptions for the .EXP File.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NUMBER_OF_DATA_POINTS	I	> 0	-	Number of data points in the file.
NUMBER_OF_PARAMETERS	I	> 0	-	Number of parameters for each point. In the case of the elevation data file this value is equal to 1.
X	I	-	m or ft	X Coordinate of each elevation point. See comment 1.
Y	R	-	m or ft	Y Coordinate of each elevation point. See comment 1.
P	R	-	m or ft	Parameter value. See comment 2.

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

9.5.1.2 Comments for the .EXP Data File

1. X and Y coordinates may be given in either meters or feet, depending on the units being used in the project. Coordinate system should always correspond to plane projection. RiverFlow2D does not support geographical coordinates in Latitude/Longitude format.

2. Elevation values should be given in the same units as the corresponding coordinates.

9.6 Boundary conditions data

9.6.1 One Variable Boundary Condition Files

This format applies to the following data files:

- Time vs. Water Surface Elevation (BCTYPE = 1)
- Time vs. Discharge (BCTYPE = 6)

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**.

9.6.1.1 Example of the Boundary Condition File for One Variable Time Series

The following example shows an inflow hydrograph where NDATA is 7 and there are 7 lines with pairs of time and discharge:

```
7
0. 20.
1. 30.
1.3 50.
2. 90.
4. 120.
5. 200.
7. 250.
```

Table 9.47 – Variable Descriptions of Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	h	Time in hours. The time interval is arbitrary.

Continued on next page

Table 9.47 – continued from previous page

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
VARIABLE	R	-	-	Represents Water Surface Elevation, Discharge, U or V velocity components depending on the boundary condition.

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

9.6.2 Two Variables Boundary Condition Files

This format applies to the following data files:

- Time vs. Discharge Q and Water Surface Elevation (BCTYPE = 5)
- Time vs. Q water discharge and Qs sediment discharge (BCTYPE = 26)

Line 1: Number points in data series.
NDATA

NDATA lines containing time and two values.
TIME(I) VARIABLE1(I) VARIABLE2(I)

Where VARIABLE1(I) and VARIABLE2(I) depend on the boundary condition type as follows:

BCTYPE	VARIABLE1	VARIABLE2
5	Q	WSE

9.6.2.1 Example of the Two-Variable Boundary Condition File

The following example shows a file for BCTYPE=5 where discharge and WSE are given, NDATA is 10 and there are 10 lines with pairs of time, discharge and WSE:

```
10
 0.   20. 1420.
 1.   30. 1421.5
 1.3 50. 1423.
...
 7.   250. 1420.
 8.1 110. 1426.
10.    60. 1423.5
```

```
20.   20. 1421.
```

Table 9.49 – Variable Descriptions of Two-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	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.

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

9.6.3 Multiple-Variable Boundary Condition Files

This format applies to the following data file:

- Time vs. Q water discharge and Qs sediment discharge (BCTYPE = 26)

Line 1: Number points in data series.
NDATA

NDATA lines containing time and two values.
TIME(I) VARIABLE1(I) VARIABLE2(I) ... VARIABLEN(I)

Where VARIABLE1(I) ... VARIABLEN(I) depend on the boundary condition type as follows:

BCTYPE	VARIABLE1	VARIABLE2
26	Q	Qs

9.6.3.1 Example of the Multiple-Variable Boundary Condition File

The following example shows a file for BCTYPE=26 where water discharge and sediment discharge for two fractions are given, NDATA is 10 and there are 10 lines with pairs of time, discharge and WSE:

```
10
 0.   20. 0.001 0.002
 1.   30. 0.002 0.005
 1.3  50. 0.003 0.010
...
 7.  250. 0.01  0.015
 8.1 110. 0.005 0.009
10.   60. 0.004 0.007
20.   20. 0.003 0.005.
```

Table 9.51 – Variable Descriptions of Multiple-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of points in data series.
TIME	R	> 0	h	Time in hours. The time interval is arbitrary.
VARIABLE1	R	-	-	Represents Water Discharge.
VARIABLE2..N	R	-	-	Represents Sediment Discharge for the given fraction.

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

9.6.4 Stage-Discharge Data Files

This format applies to the stage (water surface elevation) vs. discharge table used for BC-TYPE = 9 and 19.

Line 1: Number points in data series.
NDATA

NDATA lines containing stage and discharge.
STAGE(I) Q(I)

Where STAGE(I) is water surface elevation and Q(I) is the corresponding discharge.

9.6.4.1 Example of the Stage-Discharge Boundary Condition File

The following example shows a stage-discharge rating table where NDATA is 21 and there are 21 lines with pairs of stage and corresponding discharge:

```
21
-1.00 0.00
-0.75 1.79
-0.50 5.20
-0.25 9.45
0.00 14.23
0.25 19.37
0.50 24.76
0.75 30.36
1.00 36.09
1.25 41.95
1.50 47.89
1.75 53.92
2.00 60.00
2.25 66.14
2.50 72.31
2.75 78.53
3.00 84.78
3.25 91.05
3.50 97.35
3.75 103.67
4.00 110.01
```

Table 9.52 – Variable Descriptions of Two-Variable Boundary Condition Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
STAGE	R	> 0	m or ft	Water surface elevation.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

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

9.6.5 Culvert Depth-Discharge Data Files

This format applies to the culvert depth vs. discharge table.

Line 1: Number points in data series.
NDATA

NDATA lines containing depth and discharge.
DEPTH(I) Q(I)

Where DEPTH(I) is depth corresponding to discharge Q(I).

9.6.5.1 Example of the Culvert Depth-Discharge File

The following example shows a depth-discharge rating table for a culvert. NDATA is 7 and there are 7 lines with pairs of depth and corresponding discharge:

```
7
0         0.20
0.1       1.00
1.00     36.09
2.00     60.00
3.00     84.78
4.00    110.01
100.00 110.02
```

Table 9.53 – Variable Descriptions of Culvert Depth-Discharge Data Files.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	I	> 0	-	Number of lines in data file.
DEPTH	R	> 0	m or ft	Water depth.
Q	R	> 0	m^3/s or ft^3/s	Water discharge.

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

10 — Output File Reference

RiverFlow2D generates results in many output ASCII text files. These files can be easily accessed with any text editor and they can be imported into QGIS o other GIS software for visualization and analysis. HydroBID Flood always creates output ASCII files in both English and metric units depending on the units provided in the data files.

10.1 Output File Overview

The following tables summarizes the output files generated by RiverFlow2D:

Table 10.1 – List of Output Data Files.

RESULTS	FILE EXTENSION	DESCRIPTION
GENERAL OUTPUT		
List of output times	`.outfiles`	Reports output times for result files.
Run control parameters, components used, etc.	`.outi`, `.oute`	Echoes input data read from files including modeling control parameters, mesh data, boundary conditions, and for each report time interval inflow and outflow discharges and velocities `.outi` file is in metric units and `.oute` file in English units.
Triangular element mesh information	`.meshouti`, `.meshoute`	These files provide comprehensive information about the triangular element mesh. `.meshouti` is in metric units and `.meshoute` file in English units.
Run progress results	`.rout`	This file report for each output interval the computer time, average time step, inflow and outflow water and sediment transport discharge at open boundaries and volume and mass conservation errors.
Maximum values tabular output	`.maxi`, `.maxe`	For each output interval maximum nodal velocity modules, depths, and are written to file `.maxi` is in metric units and `.maxe` in English units.
Time series at observation points	`.outi`, `.oute`	These files report time series of results at observation points. The model finds the cell closest to the provided coordinates and the files report time series of velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge. File name format is as follows: `RESvsT_cellnumber.outi` for metric units and `RESvsT_cellnumber.oute` for English units.

Continued on next page

Table 10.1 – continued from previous page

RESULTS	FILE EXTENSION	DESCRIPTION
Mass balance	massBalance.out	Report total inflow and outflow discharges and volumes for each output interval.
Hot start	2binitialized.hotstart	Generated in the RiverFlow2D model to restart a simulation from previously computed results. The file contains the time in seconds and the corresponding stateN.out file name from which the model will restart when using the hot start option.
PROFILE AND CROSS SECTION OUTPUT		
Cross section output	.xseci, .xsece	For all output intervals, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number, water and sediment discharge. .xseci is in metric and .xsece in English units.
Cross section hydrographs	.xsech, .xsecsed	Report a hydrograph table for each cross section. .xsech is the water hydrograph and .xsecsed the sediment flux hydrograph for each cross section. See Comment 1.
Profile output	.prfi, .prfe	For each output interval and for a number of points along user defined polylines, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number are written to file .prfi is in metric and .prfe in English units.
COMPONENT OUTPUT		
Culverts	.out	Output discharge at every culvert for each report interval. File name format is as follows: CULVERT_culvertID.out where culvertID is the user provided name.
Internal Rating Tables	.out	Output discharge at every IRT for each report interval.File name format is as follows: IRT_IrtID.out, where IrtID is the text provided by the user to identify the Internal Rating Table.
Weirs	.weiri, .weire	Report results for weirs. .weiri is in metric units and WEIRE in English units.
GRAPHIC OUTPUT		
Visualization Toolkit Files		
General results	.vtk	When the Create Graphic Output Files check box is selected in the Graphic Output Panel, RiverFlow2D model will output .vtk files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of RiverFlow2D results. ParaView (www.paraview.org) is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D o programmatically using ParaView's batch processing capabilities.

Table 10.2 – List of Output Data Files for HydroBID Flood.

RESULTS	FILE EXTENSION	DESCRIPTION
CELL BASED RESULTS		
Spatial distribution of results for each report interval	`.textout`	For each output interval cell velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number, and sediment transport fluxes, etc., are written to file. These files are used by HydroBID Floodto prepare Results vs Time maps. The file names are `cell_time_metric_dddd_hh_mm_ss.textout` for metric units and `cell_time_eng_dddd_hh_mm_ss.textout` for English units.
Spatial distribution of pollutant concentrations for each report interval	`.textout`	For each output interval cell concentrations are written to file. These files are used by HydroBID Floodto prepare Pollutant Concentration vs Time maps. The file name is `cell_conc_dddd_hh_mm_ss.textout`.
Spatial distribution of sediment concentrations for each report interval	`.textout`	For each output interval cell sediment concentrations are written to file. These files are used by HydroBID Floodto prepare sediment concentration vs Time maps. The file name is `cell_st_dddd_hh_mm_ss.textout`.
Result files to generate animations		The following files share the `<ProjectName>.*` name and are used to generate animations. They can also be used to create plots for specific times. The files for each variable the corresponding value for each cell and for all times.
	`.DepthAllTimes`	Depths.
	`.WSEAllTimes`	Water surface elevations.
	`.UAllTimes`	Velocity component in x direction.
	`.VAllTimes`	Velocity component in y direction.
	`.VelAllTimes`	Velocity magnitud.
	`.FroudeAllTimes`	Froude number.
	`.ShearAllTimes`	Shear stress.
	`.RainAllTimes`	Rainfall volume. This file is only created when using the Rainfall option.
	`.InfilAllTimes`	Infiltration volume. This file is only created when using the Infiltration option.
	`.ZbedAllTimes`	Bed elevation. This file is only created when using the ST module.
	`.DeltaZAllTimes`	Bed elevation change. This file is only created when using the ST module.
	`.qsxAllTimes`	Volumetric sediment discharge per unit width in x direction. This file is only created when using the ST module.
	`.qsyAllTimes`	Volumetric sediment discharge per unit width in y direction. This file is only created when using the ST module.
	`.SSConcAllTimes`	Suspended sediment volumetric concentration. These files are only created when using the ST module with Suspended Sediment. The files are named as follows: `<ProjectName>_conc_xx.SSConcAllTimes`, where `_xx` is the sediment fraction number (e.g. _01, _02, etc.)
	`.ConcAllTimes`	Pollutant concentration. These files are only created when using the pollutant transport module. The files are named as follows: `<ProjectName>_conc_xx.SSConcAllTimes`, where `_xx` is the pullutant number (e.g. _01, _02, etc.)

Continued on next page

Table 10.2 – continued from previous page

RESULTS	FILE EXTENSION	DESCRIPTION
	.ManNAllTimes	Manning n coefficient. This file is only created when using the Manning n variable with depth option.
Maximum Values at Cells Files	.textout	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: For English units: <ProjectName>_cells_eng_max.textout. For metric units: <ProjectName>_cells_metric_max.textout. See Comment 1.
Time-to-Depth at Cells File	.textout	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: <ProjectName>_time2depths_cells.textout. See Comment 1.
Hazard Intensity Values at Cells File	.textout	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: <ProjectName>_cells_hazard.textout. See Comment 1.

10.1.0.1 Comments for Output Files

In the RiverFlow2D model these files are generated during the final step after the model completes the run, and when post processing results using the Plot RiverFlow2D results on the *Data Input Program Graphic Output Options* panel.

10.1.1 Essential files required to generate maps, graphics and animations

As it is clear from the list of files given above, HydroBID Flood 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 HydroBID Flood. 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 HydroBID Flood project. This assumes the existing project has the layers created to generate the RiverFlow2D files such as *Trimesh*, etc.

The following table presents the various graphics and animations that can be created with HydroBID Flood and the output files necessary for each graph.

Table 10.3 – Essential output files to create graphs with HydroBID Flood

GRAPHIC OUTPUT	REQUIRED FILES
Results vs Time Maps	1 `<ProjectName>.outfiles` 2 `cell_time_metric_dddd_hh_mm_ss.textout` for metric units. 3 `cell_time_eng_dddd_hh_mm_ss.textout` for English units.
Pollutant Concentration vs Time Maps	1 `<ProjectName>.outfiles` 2 `cell_conc_dddd_hh_mm_ss.textout`
Sediment Concentration vs Time Maps	1 `<ProjectName>.outfiles` 2 `cell_st_dddd_hh_mm_ss.textout`
Maximum Result Maps	1 `<ProjectName>_cells_eng_max.textout` for metric units. 2 `<ProjectName>_cells_metric_max.textout` for English units.
Time-to-Depth Maps	1 `<ProjectName>_time2depths_cells.textout`
Hazard Intensity Maps	1 `<ProjectName>_cells_hazard.textout`
Animations	1 `<ProjectName>.outfiles` 2 `<ProjectName>.*AllTimes`
CrossSections - Profiles	1 `<ProjectName>.outfiles` 2 `cell_time_metric_dddd_hh_mm_ss.textout` for metric units. 3 `cell_time_eng_dddd_hh_mm_ss.textout` for English units. 4 `<ProjectName>.*AllTimes`

10.2 General Output Files

This section describes content of each output file.

10.2.1 Output times .outfiles file

This file report the time corresponding to each output interval.

The following is an example of the content of a typical `.outfiles` file:

```
time_metric_0000_00_00_00.exp
time_metric_0000_00_06_00.exp
time_metric_0000_00_12_00.exp
time_metric_0000_00_18_00.exp
time_metric_0000_00_24_00.exp
time_metric_0000_00_30_00.exp
time_metric_0000_00_36_00.exp
```

10.2.2 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 `.outi` file is in metric units and `.oute` 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.
================================================================================
            OUTPUT OPTIONS
================================================================================
  Output for cross sections : ON.
  Output for profile        : ON.
  Text Output               : ON.
  Graphical output          : OFF.
================================================================================
            INITIAL CONDITIONS
================================================================================
  Initial dry bed.
================================================================================
            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

    Calculated Rating Table for boundary No.    1 using Slope = .300E-01 is:
         WSE               Q
           m.             m3/s
       153.839           0.104
       154.339          62.526
       154.839         363.995
       155.339         934.946
       155.839        1774.062
       156.339        2915.863
       156.839        4396.274
       157.339        6281.424
       157.839        8580.529
       158.339       11284.717
       158.839       14415.009
       159.339       18178.709
       159.839       22196.233
```

10.2.3 Mesh Data and Mesh Metrics .meshouti and .meshoute files

Mesh data is written to files with extensions: `.meshouti` (metric units) and `.meshoute` (English units). These files provide comprehensive information about the triangular element mesh. The following table summarizes the available output.

Table 10.4 – Variables reported in `.meshouti` and `.meshoute` Files.

VARIABLE	DESCRIPTION
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	Element/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
Element 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 element/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.

Part 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
```

10.2.4 Run Summary .rout file

Run summary report is written to file with extension: `.rout`. These files report for each output interval the computer time, average time step, and for each open boundary inflow (positive) o outflow (negative) discharge (m^2/s o ft^3/s), volume conservation error (%), volumetric sediment discharge (m^2/s o ft^3/s) and sediment mass conservation error (%)

10.2.4.1 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.
```

10.2.5 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. `.maxi` is in metric units and `.maxe` in English units. The reported variables are described in the following tables:

Table 10.5 – Variables Reported on the Maximum Value Tabular Files when not using the Sediment Transport Model.

COLUMN	VARIABLE	DESCRIPTION	ENGLISH UNITS	METRIC UNITS
1	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

Table 10.6 – Variables Reported on the Maximum Value Tabular Files when using the Sediment Transport Model.

COLUMN	VARIABLE	DESCRIPTION	ENGLISH UNITS	METRIC UNITS
1	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	lb/ft	N/m

A typical output `.maxi` 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
```

10.2.6 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:

Table 10.7 – Variables Reported on the Observation Point Files.

COLUMN	VARIABLE	DESCRIPTION	ENGLISH UNITS	METRIC UNITS
1	Time	Time in hours	-	-
2	U	Velocity component in x direction	ft	m
3	V	Velocity component in y direction	ft	m
4	VELOCITY	Maximum velocity magnitude $\sqrt{U^2+V^2}$	ft/s	m/s
5	DEPTH	Maximum water depth	ft	m
6	WSEL	Maximum water elevation	ft	m
7	BEDEL_ORI	Maximum bed elevation*	ft	m
8	BEDEL	Maximum bed elevation*	ft	m
9	DELTA_BED	Minimum erosion depth*	ft	m
10	Froude	Maximum deposition depth*	ft	m
11	QSX	Volumetric sediment discharge per unit width in x direction	ft^2/s	m^2/s
12	QSY	Volumetric sediment discharge per unit width in y direction	ft^2/s	m^2/s
13	QS	Volumetric sediment discharge magnitude $Q_s = \sqrt{Q_{sx}^2 + Q_{sy}^2}$	ft^2/s	m^2/s

The file name for each cell is:

`RESvsT_cellnumber.outi` for metric units and
`RESvsT_cellnumber.oute` for English units.

For example: `RESvsT_0000010.outi` is the file name for time series results of cell 10. 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
```

10.2.7 Hot Start 2binitialized.hotstart File

The hot start `2binitialized.hotstart` file is used to restart a simulation from previously computed results and when hot start option is selected. By default the file contains the name of the last report time in seconds and the corresponding state file. Those results will be used as initial condition to restart the simulation when the hot start option is activated. For example, if the user stops the simulation at 5 hours to review results o for any other reason the, `2binitialized.hotstart` file would have the following text:

```
18000 state5.out
```

Note that the `stateN.out` files are named sequentially. For instance, `stateN.out` corresponds to the Nth report interval.

HydroBID Flood can be restarted from the any existing report time by reading the initial conditions from the `stateN.out` file indicated in the `2binitialized.hotstart` file. To restart from a time different from the last one calculated, just edit the `2binitialized.hotstart` file and enter the desired time in seconds and corresponding `stateN.out` file name that is to be used as initial conditions. For example, to hot start from hour 3 (10800 seconds) and assuming that the report interval is 0.5 hours, the `2binitialized.hotstart` file should contain the following entry:

```
10800 state6.out
```

The hot start option is often useful to establishing initial conditions common to a series of simulations for various return periods. For instance, to generate your initial state, you could run the model with a constant discharge inflow until the model converges to a steady state. Assuming that the final report time corresponds to the `state20.out` file, you can edit the `2binitialized.hotstart` file as shown:

```
0 state20.out
```

Then when you run the HydroBID Flood model using the hot start option, the model will start assuming that the data in the `state20.out` file will define the initial conditions. You may want to keep the `2binitialized.hotstart` and `state20.out` files in a separate directory and copy them to the project folder for each desired scenario.

Please, keep in mind that the `stateN.out` files are tied to the mesh you use, so if you modify the mesh in any way, you will need to use the `stateN.out` corresponding to that mesh.

10.3 Component Output Files

10.3.1 Culvert CULVERT_culvertID.out Output Files

For each culvert, HydroBID Flood creates an output file named: `CULVERT_culvertID.out`, where `culvertID` is the text provided by the user to identify the culvert. Report includes discharge for each report interval as shown:

```
=====================================================================================
=====================================================================================
                          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
```

10.3.2 Internal Rating Table IRT_irtID.out Files

For each Internal Rating Table, HydroBID Flood creates an output file named: `IRT_irtID.out`, where `irtID` is the text provided by the user to identify the Internal Rating Table. Report includes discharge for each report interval as shown:

```
================================================================================
Results for Internal Rating Table no.:          1 ID: IIRT1
================================================================================
 Time              WSE         Qc
 hrs.               m.         m3/s
0.10000           0.712       4.110
0.20000           1.080       7.888
0.30000           1.290       10.376
0.40000           1.291       10.381
0.50000           1.290       10.375
|
```

10.3.3 Weir Output .weiri and .weire Files

These files report results for each weir and for each output interval. File extension is `.weiri` for metric units and `.weire` for English units. Output includes the following information:

Table 10.8 – Variables Reported on the Weir Point Files.

VARIABLE	DESCRIPTION
EDGE	Edge number
N1	Cell at side 1 of the edge
N2	Cell at side 2 of the edge
WSE1	Water surface elevation at cell N1
WSE2	Water surface elevation at cell N2
D1	Depth at cell N1
D2	Depth at cell N2
Distance	Edge length
Q	Edge discharge

A typical weir output file format is shown below:

```
=====================================================================================

 WEIR RESULTS IN SI UNITS

| TIME: 0000 days,07 hours,30 min.,00 secs.

 WEIR NO.:   1 WEIR ID: Weir6

 EDGE       N1         N2        WSE1       WSE2        D1        D2      Distance      Q
                                  (m)        (m)        (m)       (m)       (m)      (m3/s)
   1     208257      24110     527.13     527.14      0.63      3.14      6.00      0.20
   2     208259      24105     527.13     527.14      0.63      3.14      6.00      0.24
   3     208261      24101     527.13     527.14      0.63      3.14      6.00      0.25
   4     208263      24085     527.13     527.14      0.63      3.14      6.00      0.25
   5     208265       1800     527.13     527.14      0.63      3.14      6.00      0.25
   6     208267        909     527.13     527.14      0.63      3.14      6.00      0.25
   7     208269        111     527.14     527.14      0.64      3.14      6.00      0.15
Total discharge over weir Q =     1.597 m3/s
```

10.4 Cross Section and Profile Output Files

10.4.1 General Cross Section .xseci and .xsece Files

When using the *Output results* for *cross sections* option, the model will generate files with extensions `.xseci` and `.xsece`, that report results along user provided cross sections. For each output interval and for each user defined cross sections the bed elevation, depth, water surface elevation, depth average velocity, Froude number and volumetric sediment discharge per unit width is written to file `.xseci` is in metric and `.xsece` in English units. A typical `.xseci` file is as follows:

```
=====================================================================================
=====================================================================================
                           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 `.xseci` and `.xsece` files will display the cross section water discharge. When running sediment transport, in addition to the water discharge these files will report the total sediment discharge in ft^3/s o m^3/s.

10.4.2 Cross Section Hydrograph .xsech and .xsecsed Files

These files will only be generated using the post processing Plot RiverFlow2D results button on the *Graphic Output Options* panel. When using the *Output results for cross sections* option, the model will generate files with extension `.xsech` and `.xsecsed` (if using sediment transport component), that report a hydrograph table for each cross section. A typical path.xsech file is as follows:

```
=====================================================================================
=====================================================================================
                           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
```

10.4.3 Profile .prfi and .prfe Files

When using the *Output results for profiles* option, the model will generate files with extensions `.prfi` and `.prfe`, that report results along user provided polylines. For each output interval and for the number of points along user defined polylines these files list bed elevation, depth, water surface elevation, depth average velocity, and Froude number. `.prfi` is in metric and `.prfe` in English units. An example output is shown below:

```
===================================================================================
===================================================================================
                          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
```

10.5 Output Files for QGIS Post-processing

10.5.1 General Results at Cells

These ASCII files allow seamless transfer to QGIS Geographic Information System software. These files use the `.textout` extension and are named as follows:

For English units: `cell_time_eng_dddd_hh_mm_ss.textout`
For Metric units: `cell_time_metric_dddd_hh_mm_ss.textout`

Where dddd is days, hh is hours, mm is minutes and ss seconds. For example

`cell_time_eng_0001_12_01_34.textout`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line contains the number if cells (NELEM) and the number of cell parameters which is 16. Then it follows NELEM lines with results for each cell in the triangular element mesh as shown:

Table 10.9 – Variables Reported on cell_time_*.textout Output Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Velocity component in x direction U	ft/s	m/s
2	Velocity component in y direction V	ft/s	m/s
3	Velocity magnitude $\left\|\vec{U}\right\| = \sqrt{U^2+V^2}$	ft/s	m/s
4	Water surface elevation	ft	m
5	Depth H	ft	m
6	Initial bed elevation	ft	m
7	Bed elevation	ft	m
8	Bed elevation change since time = 0	ft	m
9	Froude number	-	-
10	Volumetric sediment discharge per unit width in x direction: Q_{sx}	ft^2/s	m^2/s
11	Volumetric sediment discharge per unit width in y direction: Q_{sy}	ft^2/s	m^2/s
12	Volumetric sediment discharge magnitude: $Q_s \sqrt{Q_{sx}^2 + Q_{sy}^2}$	ft^2/s	m^2/s
13	Bed shear stress*: $\tau = \gamma H S_f = \gamma \left(Un/k\right)^2 / H^{1/3}$	lb/ft^2	Pa
14	Accumulated rainfall volume	ft^3	m^3
15	Accumulated infiltration volume	ft^3	m^3
16	Manning's n	-	-

**English units:* $= 62.4 lb/ft^3$, $k = 1.49$.
**Metric units:* $= 9810 N/m^3$; $k = 1$, n = *Manning's coefficient.*

10.5.2 Pollutant Concentration Files

These ASCII files contains the pollutant. These files use the `.textout` extension and are named as follows:

`cell_conc_dddd_hh_mm_ss.textout`

Where dddd is days, hh is hours, mm is minutes and ss seconds. For example

`cell_conc_0001_12_01_34.textout`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of solutes used in the PL run (NPOLLUTANTS_MAX). Then follows NELEM lines with results for each cell in the triangular element mesh as shown:

Table 10.10 – Variables Reported on `cell_conc*.textout` Output Files.

COLUMN	VALUE	UNITS
1	Concentration for solute 1	Same as in BC's
2	Concentration for solute 2	Same as in BC's
...	...	...
NPOLLUTANTS_MAX	Concentration for solute NPOLLUTANTS_MAX	Same as in BC's

The following file is an example of a typical `cell_conc*.textout` 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`cell_conc*.textout` has 3 pollutants.

10.5.3 Suspended Sediment Concentration Files

These ASCII files contain suspended sediment concentrations. These files use the `.textout` extension and are named as follows:

`cell_st_dddd_hh_mm_ss.textout`

Where dddd is days, hh is hours, mm is minutes and ss seconds. For example

`cell_st_0001_12_01_34.textout`

corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of suspended sediment fractions used in the ST run (NSSNFRAC). Then follows NELEM lines with results for each cell in the triangular element mesh as shown:

Table 10.11 – Variables Reported on `cell_st*.textout` Output Files.

COLUMN	VALUE	UNITS
1	Concentration by volume for fraction 1	Fraction of 1
2	Concentration by volume for fraction 2	Fraction of 1
...	...	...
NSSNFRAC	Concentration by volume for fraction NSSNFRAC	Fraction of 1

The following file is an example of a typical `cell_st*.textout` file:

```
2
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000000 0.000000
0.000287 0.000287
0.000456 0.000456
0.000356 0.000356
0.000334 0.000334
0.000000 0.000000
...
```

In this example, the `cell_st*.textout` has 2 suspended sediment fractions.

10.5.4 *AllTimes Output Files to Generate Animations, Cross Sections and Profiles

These result files to generate animations, cross sections and profiles in HydroBID Flood, and can also be used to create plots for specific times. The file name is as follows: `<ProjectName>.<VAR>AllTimes`, were `<VAR>` is the corresponding output variable. For instance Depths for all times is written to `<ProjectName>.DepthsAllTimes`. Other files names are summarized in table 10.2.

The file format is as follows:

Line 1:
FILEID
Line 2:
NELEM NTIMES TIME_INTERVAL GLOBALMIN GLOBALMAX

then NELEM lines containing for each cell
VAR[T0] VAR[T1] VAR[T2]...VAR[NTIMES]
...

Table 10.12 – Variable Descriptions for `<ProjectName>.<VAR>AllTimes` Files

VARIABLE	DESCRIPTION
FILEID	File ID for internal use.
GLOBALMAX	Maximum of the variable for all report times.
GLOBALMIN	Maximum of the variable for all report times
NELEM	Number of cells.
NTIMES	Number of times. Is also the number of columns in the file. Each column corresponds to a time.
TIME_INTERVAL	Time interval between columns. Column 1 corresponds to the initial time, Column 2 to the first report interval, and so forth.
VAR[Ti]	Variable reported in the file for each time Ti.

10.5.4.1 Example of a <ProjectName>.<VAR>AllFiles file

```
2
1803    10      0.100   0.000   30.117
0.000   0.000   0.000   0.259   6.152   7.591   8.448   8.932   9.000   9.016
0.000   0.000   0.000   0.000   1.628   2.618   3.765   4.160   4.244   4.265
0.000   1.637   8.971  13.016  15.385  17.298  17.814  17.905  17.938  17.943
0.000   0.000   0.000   1.901   5.009   7.095   7.928   8.035   8.071   8.079
0.000   0.000   0.000   0.000   1.527   3.970   5.193   5.528   5.607   5.624
0.000   0.000   0.000   0.000   1.076   3.682   4.921   5.255   5.334   5.350
0.000   0.000   6.999  10.958  13.243  15.117  15.633  15.727  15.760  15.766
0.000   0.000   0.000   7.812  10.678  12.561  13.227  13.315  13.349  13.354
0.000   0.000   0.000   5.180   8.140  10.062  10.746  10.844  10.880  10.886
0.000   0.000   0.000   3.715   7.009   9.121   9.971  10.081  10.117  10.125
0.000   0.000   0.000   5.048   8.852  11.344  12.713  13.162  13.263  13.286
0.000   0.000   0.000   0.000   2.925   4.873   5.660   5.769   5.801   5.808
0.000   0.000   0.000   0.000   2.785   4.799   5.609   5.722   5.755   5.762
0.000   0.000   0.000   6.348   9.814  12.177  13.320  13.612  13.688  13.705
0.000   0.000   0.000   6.415  10.956  12.621  13.941  14.179  14.235  14.249
0.000   0.000   5.403  11.126  15.477  18.012  19.044  19.173  19.215  19.225
```

This `<ProjectName>.<VAR>AllTimes` file has results for 1803 cells. There are 10 times and the time interval between times is 0.1 h. The GLOBALMAX is 30.117 and GLOBALMIN is 0.000.

10.5.5 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 `.textout` extension and are named as follows:

For English units: `<ProjectName>_cells_eng_max.textout`
For Metric units: `<ProjectName>_cells_metric_max.textout`

The format for these files is as follows. The first line contains the number of cells (NELEM), and the number of cell parameters which is 6 by default, o 11 if the run was made with the Sediment Transport Module. There follows NELEM lines with velocity module, depth and water surface elevation for each cell as shown:

Table 10.13 – Variables Reported on the <ProjectName>_cells_eng_max.textout o <ProjectName>_cells_metric_max.textout Files.

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Cell number	-	-
2	Maximum velocity magnitude $\sqrt{U_x^2 + U_y^2}$	ft/s	m/s
3	Maximum depth	ft	m
4	Maximum water surface elevation	ft	m
5	Maximum depth x velocity	ft^2/s	m^2/s
6	Maximum bed elevation*	ft	m
7	Minimum bed elevation*	ft	m
8	Maximum erosion depth*	ft	m
9	Maximum deposition depth*	ft	m
10	Maximum bed shear stress	lb/ft^2	Pa
11	Maximum impact force	lb/ft	N/m

**Written only when using the sediment transport component.*

10.5.6 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:

`<ProjectName>_time2depths_cells.textout`

The format for these files is as follows. The first line indicates the number of cells (NELEM) and the number of cell parameters (5 by default). For the file in Metric Units there follows NELEM lines with time to 0.30 m, time to 0.5 m, time to 1 m, time to maximum depth, and total inundated time for each cell as shown in Table 10.14. 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 10.14. When the cell remains dry o depth is below 1ft the reported value is -1.

Table 10.14 – Variables Reported on the <ProjectName>_time2depths_cells.textout File.

COLUMN	VALUE	UNITS
1	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.

**Time = -1 when cell is dry o depth does not reach 0.3 m (1ft).*

10.5.7 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 `.textout` extension and are named as follows:

`<ProjectName>_cells_hazard.textout`

The format for these files is as follows. The first line indicates the number of cells (NELEM), and the number of cell parameters (11). There follows NELEM lines with the hazard intensities for each cell as shown:

Table 10.15 – Variables Reported on the <ProjectName>_cells_hazard.textout File.

COLUMN	VALUE	HAZARD ZONES
1	USBR Homes	$0, 1, 2,$ and 3
2	USBR Passenger Vehicles	$0, 1, 2,$ and 3
3	USBR Mobile Homes	$0, 1, 2,$ and 3
4	USBR Adults	$0, 1, 2,$ and 3
5	USBR Children	$0, 1, 2,$ and 3
6	Swiss Method for Water Flooding	$0, 1, 2,$ and 3
7	Swiss Method for Debris Flow	$0, 1, 2,$ and 3
8	Austrian Method for River Flooding	$0, 1,$ and 2
9	Austrian Method for Torrents Tr=100 yrs.	$0, 1,$ and 2
10	Austrian Method for Torrents Tr=10 yrs.	$0, 1,$ and 2
11	UK Method	$0, 1, 2,$ and 3
12	Australia Flood Hazard	$0, 1, 2,$ and 3

.

10.5.7.1 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:

Table 10.16 – USBR Hazard Classification.

ATTRIBUTE VALUE	HAZARD CLASSIFICATION
1	Low-danger zone
2	Judgment zone
3	High-danger zone

For further details about the USBR Hazard classification, consult USBR (1988).

10.6 VTK Output Files for Paraview

RiverFlow2D model will output `.vtk` files, that report velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number and sediment transport discharge for each output time interval. These files can be used by third party software including Paraview to generate high quality graphs of RiverFlow2D results. ParaView (`www.paraview.org`) is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D o programmatically using ParaView's batch processing capabilities.

11 — References

Abderrezzak, K. & Paquier, A. (2011), 'Applicability of sediment transport capacity formulas to dam-break flows over movable beds', *Journal of Hydraulic Engineering, ASCE* **137**, 209–221.

Ackers, P. & White, W. (1973), 'Sediment transport, new approach and analysis', *Journal of Hydraulic Div. ASCE* **99**(11), 2041–2060.

Akan, A. O. (1993), *Urban Stormwater Hydrology: A Guide to Engineering Calculations*, CRC Press.

ASCE (1996), *Hydrology Handbook*, 2nd edn, American Society of Civil Engineers (ASCE).

Ashida, K. & Michiue, M. (1972), 'Study on hydraulic resistance and bed-load transport rate in alluvial streams', *Transactions Japan Society of Civil Engineering* **206**, 59–69.

Bagnold, R. (1973), 'The nature of saltation of bed load transport in water', *Proc. of Royal Society. Ser. A* **332**, 473–504.

Bai, L. & Jin, S. (2009), 'A conservative coupled flow/transport model with zero mass error', *J. of Hydrodynamics* **21**, 166 – 175.

Baldock, T., Tomkins, M., Nielsen, P. & Hughes, M. (2004), 'Settling velocity of sediments at high concentrations', *Coastal Engineering* **51**, 91–100.

Belleundy, P. (2000), 'Restoring flow capacity in the loire river bed', *Hydrological Processes* **14**, 2331–2344.

Brufau, P., García-Navarro, P. & Vázquez-Cendón, M. (2004), 'Zero mass error using unsteady wetting-drying conditions in shallow flows over dry irregular topography', *Int. Journal for Numerical Methods in Fluids* **45**, 1047–1082.

Camenen, B. & Larson, M. (2005), 'A bedload sediment transport formula for the nearshore', *Estuarine, Coastal and Shelf Science* **63**, 249–260.

Caviedes-Voullième, D., García-Navarro, P. & Murillo, J. (2012), 'Influence of mesh structure on 2D full shallow water equations and SCS Curve Number simulation of rainfall/runoff events', *Journal of Hydrology* **448-449**(0), 39 – 59.

Cheng, N. (1997), 'Simplified settling velocity formula for sediment particle', *J. Hydraulic Eng., ASCE.* **123**(2), 149–152.

Chien, N. & Ma, H. (1958), 'Properties of slurry flow', *J. Sediment Res.* **3**(3).

Chow, V. T. (1959), *Open Channel Hydraulics*, McGraw-Hill, New York.

Chow, V. T., Maidment, D. & Mays, L. (1988), *Applied Hydrology*, McGraw-Hill Civil Engineering Series, MCGRAW-HILL Higher Education.

Cunge, J., Holly, F. & Vervey, A. (1980), *Practical Aspects of Computational River Hydraulics*, Pitman: London.

Dai, J. (1980), An experimental study of slurry transport in pipes, *in* 'Proc, Int. Symposium on River Sedimentation', China., pp. 195–204.

De Vriend, H., Zyserman, J., Nicholson, J., Roelvink, J., Péchon, P. & Southgate, H. (1993), 'Medium-term 2DH coastal area modelling', *Coastal Engineering* **21**(1-3), 193–224.

Engelund, F. & Fredsoe, J. (1976), 'A sediment transport model for straight alluvial channels.', *Nordic Hydrology* **7**, 293–306.

Engelund, F. & Hensen, E. (1967), A monograph on sediment transport to alluvial streams, Report, Copenhagen: Teknique Vorlag.

Fei, X. J. (1981), 'Bingham yield stress of sediment water mixtures with hyperconcentration', *J. Sediment Res.* **3**.

Fernandez-Luque, R. & van Beek, R. (1976), 'Erosion and transport of bed sediment', *Journal of Hydraulic Research, IAHR* **14**(2), 127–144.

FHWA (1978), Hydraulics of bridge waterways, Report EPD-86-101, FHWA (Federal Highway Administration).

Froehlich, D. (2003), *User's Manual for FESWMS FST2DH Two-dimensional Depth-averaged Flow and Sediment Transport Model*, Report No. FHWA-RD-03-053, Federal Highway Administration, Washington, DC.

García, R., Espinoza, R., Valera, E. & González, M. (2006), An explicit two-dimensional finite element model to simulate short- and long-term bed evolution in alluvial rivers, *Journal of Hydraulic Research* **44**(6), 755–766.

García, R., González, N. & O'Brien, J. (2009), *Dam-break flood routing. Chapter 4 in: Dam-Break Problems, Solutions and Case Studies*, WIT Press, Southampton-Boston.

Garde, R. & Ranga Raju, K. (1985), *Mechanics of Sediment Transportation and Alluvial Stream Problems*, John Wiley & Sons, New York.

Godunov, S. (1959), ‘A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics’, *Mat. Sb.* **47**, 271–306.

Gupta, R. S. (1995), *Hydrology and Hydraulic Systems*, Waveland Press.

Henderson, F. (1966), *Open Channel Flow*, MacMillan series in civil Engineering.

Horton, R. (1933), ‘The role of infiltration in the hydrologic cycle’, *Trans. Am. Geophys. Union* **14**, 446 – 460.

Iida, K. (n.d.), ‘The mud flow that occurred near the explosion crater of mt. bandai on may 9 and 15, 1938, and some physical properties of volcanic mud’, *Tokyo Imperial University Earthquake Research Institute Bulletin* **16**, 1938.

Jimenez, J. & O.S., M. (2003), ‘A simple formula to estimate settling velocity of natural sediments’, *J. Waterway, Port, Coastal, Ocean Eng* **129**(2), 70–78.

Juez, C., Murillo, J. & P., G.-N. (2014), ‘Numerical assessment of bed-load discharge formulations for transient flow in 1d and 2d situations’, *Journal of Hydroinformatics* **15**(4), 1234–1257.

Julien, P. (1998*a*), *Erosion and Sedimentation*, Cambridge University Press.

Julien, P. (1998*b*), *Erosion and Sedimentation*, Cambridge.

Jury, W. & Horton, R. (2004), *Soil Physics*, John Wiley and Sons.

Kang, Z. & Zhang, S. (1980), A preliminary analysis of the characteristics of debris flow., *in* ‘Proc, Int. Symposium on River Sedimentation’, China., pp. 213–220.

Karim, F. (1988), ‘Bed material discharge prediction for non-uniform bed sediments’, *Journal of Hydraulic Engineering, ASCE* **124**(6), 597–604.

LeVeque, R. (2002), *Finite-Volume Methods for Hyperbolic problems*, Cambridge University Press, New York.

Mein, R. G. & Larson, C. L. (1973), ‘Modeling infiltration during a steady rain’, *Water Resources Research* **9**(2), 384–394.

Meyer-Peter, E. & Muller, R. (1948), ‘Formulas for bed-load transport’, *Proc. of the Second Meeting. IAHR, Stockholm, Sweden* pp. 39–64.

Mishra, S. & Singh, V. (2003), *Soil Conservation Service Curve Number (SCS-CN) Methodology*, Kluwer Academic Publishers.

Morvan, H., Knight, D., Wright, N., Tang, X. & Crossley, A. (2008), ‘The concept of roughness in fluvial hydraulics and its formulation in 1-d, 2-d and 3-d numerical simulation models’, *Journal of Hydraulic Research* **46**(2), 191–208.

Murillo, J., Burguete, J., Brufau, P. & García-Navarro, P. (2005), 'Coupling between shallow water and solute flow equations: analysis and management of source terms in 2D', *International Journal for Numerical Methods in Fluids* **49**(3), 267–299.

Murillo, J., Burguete, J., Brufau, P. & García-Navarro, P. (2007), 'The influence of source terms on stability, accuracy and conservation in two-dimensional shallow flow simulation using triangular finite volumes', *International Journal of Numerical Methods in Fluids* **54**, 543–590.

Murillo, J. & García-Navarro, P. (2010*a*), 'An exner-based coupled model for two-dimensional transient flow over erodible bed', *Journal of Computational Physics* **229**(23), 8704–8732.

Murillo, J. & García-Navarro, P. (2010*b*), 'Weak solutions for partial differential equations with source terms: Application to the shallow water equations', *Journal of Computational Physics* **229**(11), 4327–4368.

Murillo, J., García-Navarro, P. & Burguete, J. (2008), 'Analysis of a second-order upwind method for the simulation of solute transport in 2d shallow water flow', *International Journal for Numerical Methods in Fluids* **56**(6), 661–686.

Murillo, J., García-Navarro, P. & Burguete, J. (2012), 'Wave riemann description of friction terms in unsteady shallow flows: Application to water and mud/debris floods', *Journal of Computational Physics* **231**, 1963–2001.

Naef, D., Rickenmann, D., Rutschmann, P., & McArdell, B. (2006), 'Comparison of flow resistance relations for debris flows using a one-dimensional finite element simulation model', *Nat. Hazards Earth Syst. Sci.* **6**, 155–156.

Nielsen, P. (1992), *Coastal Bottom Boundary Layers and Sediment Transport*, Advanced Series on Ocean Engineering. World Scientific Publishing.

Norman, J. M., Houghtalen, R. J. & Johnston, W. J. (1985), Hydraulic design of highway culverts, Hydraulic Design Series No. 5 FHWA-IP-85-15, Federal Highway Administration, Washington, D.C.

O'Brien, J. & Julien, P. (1988), 'Laboratory analysis of mudflow properties', *J. Hydraul. Eng.* **114**(8), 877–887.

Parker, G. (1979), 'Hydraulic geometry of active gravel rivers', *Journal of Hydraulic Engineering, ASCE* **105**(9), 1185–1201.

Parker, G., Klingeman, P. & McLean, D. (1982), 'Bed load and size distribution in paved gravel bed streams', *Journal of Hydraulic Engineering* **108**(4), 544–571.

Qian, N. (1980), Basic characteristics of flow with hyperconcentration of sediment, *in* 'Proc, Int. Symposium on River Sedimentation', China., pp. 175–184.

Qian, N. & Wan, Z. (1986.), A critical review of the research on the hyperconcentrated flow in china., Report, International Research and Training Center on Erosion and Sedimentation, China.

Ratia, H., Murillo, J. & García-Navarro, P. (2014), ‘Numerical modelling of bridges in 2D shallow water flow simulations’, *International Journal for Numerical Methods in Fluids* **75**(4), 250–272.

Raudkivi, A. (1990), *Loose boundary hydraulics*, Pergamon Press, Inc., Tarrytown, N.Y.

Rawls, W. & Brakensiek, D. (1983), A procedure to predict green and ampt infiltration parameters, *in* ‘Proceeding of ASAE Conferences on Advances in Infiltration’, Chicago, Illinois, pp. 102–112.

Rawls, W., Yates, P. & Asmussen, L. (1976), Calibration of selected infiltration equation for the georgia coastal plain, Report ARS-S-113, Agriculture Research Service.

Roe, P. (1981), ‘Approximate riemann solvers, parameter vectors, and difference schemes’, *Journal of Computational Physics* **43**(2), 357 – 372.

Rogers, B. & Borthwick, A.G.L. Taylor, P. (2003), ‘Mathematical balancing of flux gradient and source terms prior to using roe’s approximate riemann solver’, *J. Comput. Phys.* **192**(2), 422–451.

Rosatti, G., Murillo, J. & Fraccarollo, L. (2008), ‘Generalized roe schemes for 1D two-phase, free-surface flows over a mobile bed’, *J. Comput. Phys.* **227**(24), 10058–10077.

Rubey, M. (1933), ‘Settling velocities of gravel, sand and silt particles’, *Amer. J. Soc.* **225**, 325–338.

Sha, Y. (1965), *Introduction to Sediment Dynamics*, Industry Press, 302-310.

Smart, G. (1984), ‘Sediment transport formula for steep channels’, *J. Hydr. Eng., ASCE.* **111**(3), 267–276.

Toro, E. (2001), *Shock-Capturing Methods for Free-Surface Shallow Flows*, Wiley, New York, p. 109.

USBR (1988), Downstream hazard classification guidelines, ACER Technical Memorandum 5, United States Bureau of Reclamation (USBR).

USDA (1986), Urban hydrology for small watersheds, Environment agency report, United States Department of Agriculture (USDA).

USDA (2004), National engineering handbook part 630 hydrology, Technical report, United States Department of Agriculture (USDA).

Van Rijn, L. (1984*a*), ‘Sediment pick-up functions’, *J. Hydr. Eng., ASCE.* **110**(10), 1494–1502.

Van Rijn, L. (1984*b*), 'Sediment transport, part i: bed load transport', *J. Hydr. Eng., ASCE.* **110**(10), 1431–1456.

Van Rijn, L. (1984*c*), 'Sediment transport, part ii: suspended load transport', *J. Hydr. Eng., ASCE.* **110**(11), 1613–1641.

Vázquez-Cendón, M. (1999), 'Improved treatment of source terms in upwind schemes for the shallow water equations in channels with irregular geometry', *Journal of Computational Physics* **148**(2), 497–526.

Vreugdenhil, C. (1994), *Numerical methods for shallow water flow*, Water Science and Technology Library. Kluwer Academic Publishers.

Wan, Z. & Wang, Z. (1994), *Hyperconcentrated Flow*, Balkema, Rotterdam.

Wilcock, P. & Crowe, J. (2003), 'Surface-based transport model for mixed-sized sediment', *Journal of Hydraulic Engineering* **129**(2), 120–128.

Wong, M. (2003), 'Does the bedload transport relation of meyer-peter and muller fits its own data?', *Proc. 30th IAHR-Congress, Thessaloniki, Greece, 8 pp.* .

Wong, M. & Parker, G. (2006), 'Re-analysis and correction of bed load relation of meyer-peter and mÃ$\frac{1}{4}$ller using their own database', *Journal of Hydraulic Engineering* **132**(11), 1159–1168.

Wu, W. (2008), *Computational River Dynamics*, Taylor and Francis.

Wu, W. & Wang, S. (2006), 'Formulas for sediment porosity and settling velocity', *J. Hydraulic Eng., ASCE.* **132**(8), 858–862.

Yang, C. (1996), *Sediment Transport Theory and Practice*, McGraw Hill, New York.

Yarnell, D. (1934*a*), 'Bridge piers as channel obstructions', *U.S. Department of Agriculture, Tech. Bull.* **442**.

Yarnell, D. (1934*b*), 'Pile trestles as channel obstructions', *U.S. Department of Agriculture, Tech. Bull.* **429**.

Zanke, U. (n.d.), Berechnung der sinkgeschwindigkeiten von sedimenten, Technical report, Technischen Universitat Hannover.

Zhang, R. (1961), *River Dynamics (in Chinese)*, Industry Press.

IDB

Inter-American
Development Bank