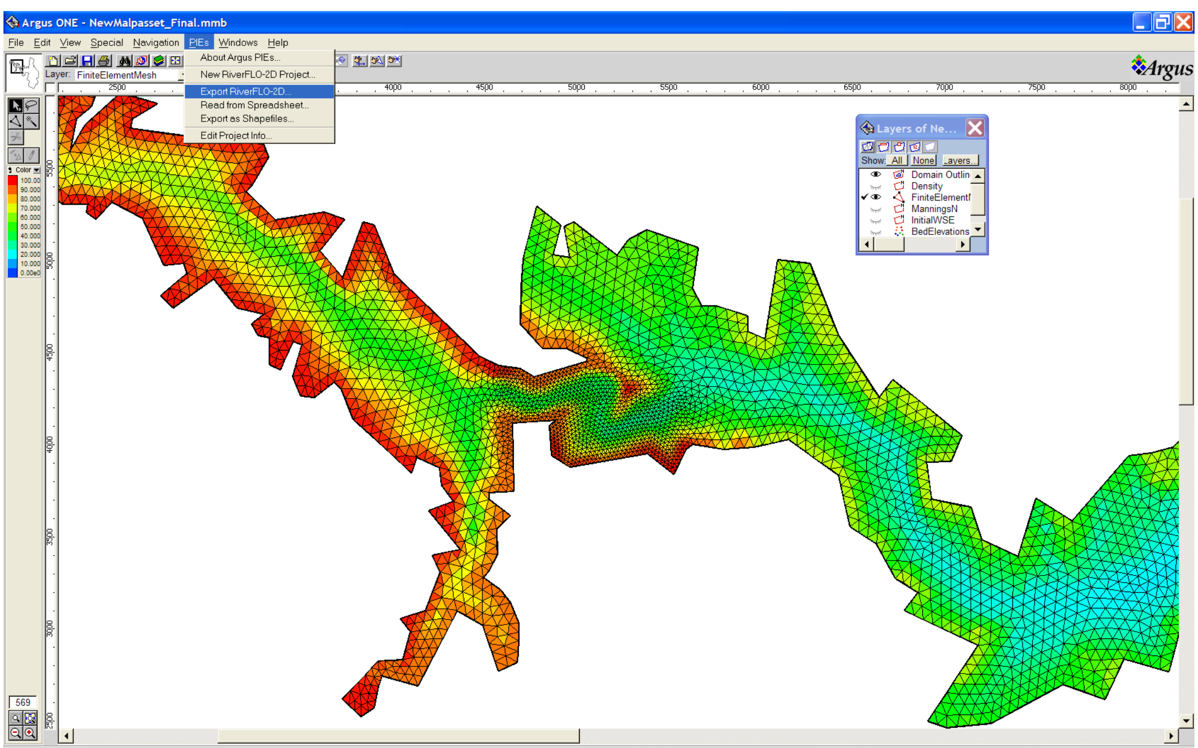
RIVERFLO-2D©

TWO-DIMENSIONAL FINITE-ELEMENT RIVER DYNAMICS MODEL

INPUT AND OUTPUT FILES REFERENCE MANUAL

RELEASE 2009



Hydronia LLC

RiverFLO-2D© model and documentation produced by Hydronia LLC.
Pembroke Pines, FL. USA.

Last source code and document modification date: March, 2011

Technical Support: E-mail: support@hydronia.com

Contents

Introduction to RiverFLO-2D input and output files

RiverFLO-2D is a hydrodynamic and mobile bed finite element model for rivers developed by Hydronia LLC. This document describes the content and format of the data input files as well as that of the output files generated by the model. The data input files are ASCII free format and can be opened by any text editor or spreadsheet program.

RiverFLO-2D uses the Argus ONE graphical software as a pre-processor to facilitate data input and as a post-processor to display results (www.argusone.com). The Argus ONE software provides a user friendly GIS environment for RiverFLO-2D, automatically linking the physical data in GIS layers to mesh nodes and elements. It provides a variety of triangular mesh generation options to optimize the mesh creation around complex river features enabling the model to resolve difficult flow issues in a rapidly varied flow field. Regenerating the computational mesh does not require re-entering bed elevations, roughness coefficients, nor boundary conditions.

To conduct a basic RiverFLO-2D simulation, four data files must be created. These are identified by their name extension: *.DAT, *.FED, *.TBA, and *.IFL. In addition, there are boundary condition files that can have any extension. These files contain run control and numerical stability data, bed elevations, roughness coefficients, finite element mesh system data, and inflow and outflow conditions. The *.DAT and boundary condition files can be interactively created with RiverFLO-2D tabbed user interface and the remaining files can be generated with Argus ONE program. To use some of the RiverFLO-2D components, additional data files need to be prepared using the RiverFLO-2D interface.

When the model is running, the user has the option of displaying the flow as velocity vector plots, flow depths, etc. Upon completion of the RiverFLO-2D simulation, results can be visualized in a number of graphic formats.

This document includes descriptions of all input and output files. Each description contains a list of variables, their definitions and instructional comments and guidelines for data organization range of data values and data limitations.

Input data files

Data files required to run the RiverFLO-2D model are automatically created by the RiverFLO-2D software system through interactive dialog boxes and the Argus ONE program. In some instances, it may be convenient to use a text editor program to quickly edit the data. Manually editing the files may be occasionally expeditious, but it may also result in data errors. This section explains the input data file content, format, and provides instructional comments on how to set key parameters for each file.

In the *C:\Program Files (x86)\RiverFLO-2D\ExampleProjects* folder there are several example projects that can be used to review the data file format. The following table summarizes the data files used by RiverFLO-2D model. For any RiverFLO-2D simulation, all files will share the same name (e.g. Case1) and use the file extensions listed in the table. For example: Case1.DAT, Case1.FED, Case1.TBA, Case1.IFL, etc.

NAME	DATA FILE EXTENSION	DEPENDENCIES	CONTENT
Elevations data file	EXP	Required	File containing scattered elevation data points.
Run control data	DAT	Required	General options such as time step parameters, metric or English units, graphical output, and initial conditions.
Finite element mesh data	FED	Required	Node coordinates and elevations, triangular mesh topology, Manning's n coefficients, and other mesh related parameters.
Mesh boundary nodes	TBA	Required	List of external and island boundary nodes.
I/O boundary conditions	IFL	Required	List of nodes where inflow or outflow conditions are imposed.
Sediment data	SED	Optional	Sediment transport formula and data.
Rainfall/Evaporation	RET	Optional	Time series for rainfall and evaporation.
Cross section coordinates for output results	REP	Optional	List of cross sections where output is required. Each cross section is defined by coordinates of its two extreme points.
Plot results options	PLT	Optional	Graphical output options.
Profile output	PRF	Optional	Mesh profile cut where results are desired.
Time series files for inflow or outflow boundary conditions	User defined	Required	Input hydrograph, outflow water surface elevations vs. time, etc. There will be a file for each open boundary condition.
Argus ONE project file	MMB	Required	This is the file where Argus ONE stores the finite element mesh and related data.

*FILE: *.EXP: Elevations Data*

This file contains scattered elevation data and is imported in an ArgusONE *BedElevations* Data layer. Each point is identified by its X and Y coordinates and the elevation value for that coordinate.

Line 1 contains

1 **NUMBER OF DATA POINTS, NUMBER OF PARAMETERS**

Following "NUMBER OF DATA POINTS" lines with X, Y and Elevation data.

2 **X(POINT), Y(POINT), ZB(POINT)**

*Example of a *.EXP Elevations data 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, there are 11086 elevation data points, one parameter per point (the elevation for each point).

Variable Descriptions for the *.EXP Elevations Data File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NUMBER OF DATA POINTS	I	> 0	-	Number of elevation 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 normally equal to 1.
X	R	-	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.
ZB	R	-	m. or ft.	Point elevation . See comments 2 and 3.

Note: I = Integer variable. R = Real variable

*Instructional Comments for the *.EXP Elevations Data File*

The following comments will assist in the developing of the *.EXP Elevations 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. RiverFLO-2D does not support geographical coordinates in Latitude/Longitude format. If the available data is in Lat/Lon format, it will have to be converted to UTM or plane Cartesian coordinates before importing them into Argus ONE. Presently RiverFLO-2D does not provide tools to make this conversion.

2. Elevation values should be given in the same units as the corresponding coordinates.

NOTE: *Each line must be delimited by the TAB or SPACE characters and should terminate with a Carriage Return. Empty fields and sequential delimiters are not allowed. If the file does not fully adhere to the above format, Argus ONE will fail to read the file and will notify you by presenting a message box indicating that it encountered a problem trying to read the file.*

*FILE: *.DAT: SYSTEM CONTROL VARIABLES*

This file contains control options such as timestep, metric or English units, physical process switches, and graphical output and initial conditions options.

Line 1 contains program version number

1 **RELEASE**

Line 2 has system component switches.

2 **IRAIN, ISED, IXSEC, IPROFILE, IVARDT, IINITIAL, IHOTSTART**

Line 3 contains time control related data.

3 **DT, DTMULT, TOUT, TLIMT**

Line 4 contains the run time plot control variable.

4 **NOGRAPH**

Line 5 contains the Manning's n value global multiplication factor.

5 **XNMAN**

Line 6 contains the finite element selective lumping parameter.

6 **EPSILON**

Line 7 is unit definition switch.

7 **NUNITS**

Line 8 is surface detention or minimum value of flow depth for dry areas.

8 **HMIN**

Line 8 is the number of nodes for time series output.

7 **NN_OUTPUT**

Line 9 contains the list of NN_OUTPUT node numbers for time series output.

9 **INTS (1:NN_OUTPUT), TIMTEP**

*Example of the *.DAT file*

```
200901
0 0 0 0 0 1 0
0.1 1 0.1 1
1
1
0.90
0
0.01
6
20
230
238
250
429
432
```

*Variable Descriptions for the *.DAT File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
DT	R	> 0	sec.	Timestep. See comment 1.
DTMULT	R	>0	-	Timestep multiplier for variable time step option. See comment 2.
EPSILON	R	0.80-0.99	-	Selective lumping parameter. See comment 3.
HMIN	R	>0	m. or ft.	Depth limit for dry-wet calculation. If depth is less than HMIN, node will be considered dry.
IHOTSTART	I	0,1	-	Switch to start run from scratch or continue a previous simulation. 0: start simulation from time=0. 1: start simulation from previous run.
IINITIAL	I	0,1,2	-	Switch to control initial conditions for water surface elevations. 0: for flat horizontal water surface elevations. 1: for initial dry bed on whole mesh. 2: initial water surface elevations read from *.FED file. See comment 4.
INTS	I	1 - NNODES	-	List of node numbers where time series of results is required.
IPROFILE	I	0,1	-	Switch to control profile section output. 0: No profile section results output. 1: Results will be output along a prescribed profile. *.PRF file needs to be created. See comment 5.
IRAIN	I	0,1		Switch to control rainfall and evaporation input. 0: no rainfall modeling. 1: rainfall/evaporation will be modeled. File *.RET with time series of rainfall and evaporation data needs to be created.
ISED	I	0,1	-	Sediment transport switch. 0: no sediment transport modeling. 1: sediment transport and mobile bed scour and deposition will be simulated. File *.SED with time sediment transport data needs to be created.

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IVARDT	I	0,1	-	Switch to control variable timestep option. See comment 6. 0: DT constant time step will be used through the simulation. 1: Variable time step will be used.
IXSEC	I	0,1	-	Cross section output switch. 0: No cross section result output. 1: Cross section results will be output to file. <NAME.REP> needs to be created. See comment 7.
NN_OUTPUT	I	1-100	-	Number of nodes where time series of results will be output.
NOGRAPH	I	0,1	-	Variable to control screen output during simulation. 0: only screen text output 1: text and graphic output during simulation. If NOGRAPH =1, the *.PLT file needs to be created.
RELEASE	I	-	-	Release number used internally for reference. Should not be changed.
TLIMT	R	>0	hours	Total simulation time.
TOUT	R	<TLIMT	hours	Output time interval for reporting results.
XNMAN	R	[0.1-2]	-	Manning's n coefficient multiplier. See comment 8.

Note: I = Integer variable. R = Real variable

*Instructional Comments for the *.DAT File*

The following comments will assist in the developing of the *.DAT file:

3. Setting the timestep DT is a critical issue for adequate stability and mass conservation. RiverFLO-2D explicit time scheme is conditionally stable, meaning that there is a maximum DT above which the simulations will become unstable. This threshold can be theoretically approximated by the Courant-Frederick-Lewy condition defined as follows:

$$\mu = \frac{\Delta t}{\Delta x} \leq \frac{4}{3}\left(\frac{3}{gh}\right)^{1/2}(1-e)^{1/2}$$

   where Δt = DT is the timestep, Δx is a measure of the minimum triangle element size, *g* is the acceleration of gravity, *h* is the flow depth and *e* is the selective lumping parameter (EPSILON). It may occur that during the initial stages of a hydrograph, velocities small and the selected the 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 timestep. Alternatively, the variable timestep option may be used (see below).

4. For variable timestep simulations, RiverFLO-2D computes an estimate of the maximum DT using the theoretical Courant-Frederick-Lewy (CFL) condition. Often this estimation is high, leading to instabilities and you may use the DTMULT variable to adjust it. The estimated CFL timestep can be multiple by the DTMULT factor to adjust the DT. Typical DTMULT values range from 0.3 to 0.7, but may vary with the simulation.

5. The selective lumping parameter EPSILON is used to improve the model capabilities to simulate abrupt change in depth or rapid flows. Lower values tend to smooth out depth and velocity gradients and higher values steepen frontal waves and may lead to instabilities. It is recommended to use an EPSILON = 0.9 for most applications.

6. There are three initial conditions options. If IINITIAL = 0, the initial water surface elevation will be constant and arbitrarily set to an elevation equal to 0.5 m (1.5 ft) higher than the highest bed elevation on the mesh. If IINITIAL = 1, the whole computational mesh will be initially dry, except at open boundaries where discharge is prescribed and depth > 0 is assumed for the first time step. If IINITIAL = 2, initial water surface elevations are read from the *.FED data file for each node in the mesh. These initial elevations can be set in the Argus ONE InitialConditions Layer, and can be arbitrary. This last option allows modeling cases where part of the domain is flooded (e.g. a reservoir) and other parts of the mesh are dry. The initial velocities are always assumed equal to 0.

7. Use the IPROFILE option to allow RiverFLO-2D to generate results along a polyline. The polyline and other required data should be given in file *.PRF which is defined later in this document.

8. If IVARDT is = 0, a constant timestep equal to DT will be used through the computation. This DT may potentially lead to instabilities if at some point during the simulation the CFL condition is exceeded. In that case, it is necessary to reduce DT and restart the simulation. If IVARDT = 1, for each time, RiverFLO-2D computes an estimate of the maximum DT using the theoretical Courant-Frederick-Lewy (CFL) condition multiplied by DTMULT. As the flow accelerates, the timestep will automatically decrease and when the flow slows down the timestep may increase.

9. Use this option to allow RiverFLO-2D to generate results along prescribed cross sections. The cross sections and other required data should be given in file *.REP which is defined later in this document.

10. 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. By performing several simulations with various XNMAN values, the model calibration can be improved.

*FILE: *.FED: FINITE ELEMENT MESH DATA*

NOTE*: It is recommended to use the Argus ONE program to create this file. The templates for RiverFLO-2D provided with Argus ONE assures that the file will be created error free and consistent with the boundary conditions and other mesh parameters. Manually editing this file may introduce errors.*

Line 1 contains general mesh parameters

1 **NELEM, NNODES, NPARAMEL, NPARAM**

Following NNODES lines contain the node coordinates and node parameters.

2 **IN, X(IN), Y(IN), ZB(IN), ETA(IN), (ARN(J), J=1, NPARAM-1)**

Following NELEM lines contain element connectivity.

3 **IE, NODE(IE,1), NODE(IE,2), NODE(IE,3), ZBED, MANNING(IE)**

*Example of a * .FED file*

```
1231   682   2   5
1  799001.27  305583.22   162.12   0.   0.04  0 0
2  798948.74  305505.30   159.74   0.   0.04  0 0
3  799037.14  305459.39   156.96   0.   0.04  0 0
.
.
.
1   1   2   3  160.88  0.04
2   4   5   6  173.78  0.04
3   7   8   9  164.90  0.04
.
.
.
```

*Variable Descriptions for the * .FED File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ARN	R	-	-	Not used in this release.
ETA(IN)	R	-	m. or ft.	Initial water surface elevation for node IN.
IE	I	>0	-	Element index. Consecutively from 1 to NELEM.
IN	I	>0	m/s or ft/s	Node number. Consecutively from 1 to NNODES.
MANNING(IE)	R	>0	-	Manning n value for element IE.
NELEM	I	1-5	-	Number of triangular finite elements.
NNODES	I	>0	-	Number of finite element nodes.
NODE(IE,1), NODE(IE,2), NODE(IE,3)	I	>0	-	Node numbers for element IE given in counter clockwise direction.
NPARAMEL	I	>0	-	Number of element parameters.
NPARAM	I	>0	-	Number of node parameters.
X(IN)	R	-	m. 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.
ZBED(IE)	R	-	m. or ft.	Initial bed elevation for element IE.

Note: I = Integer variable. R = Real variable. All variables are separated by one or more spaces.

*FILE: *.IFL: OPEN BOUNDARY CONDITIONS DATA*

***NOTE**: It is recommended to use Argus ONE program to create this file. The templates for RiverFLO-2D provided with Argus ONE assures that the file will be created error free and consistent with the boundary conditions and other mesh parameters. Manually editing this file may introduce errors.*

Line 1 contains the number of parameters and number of nodes on external boundary

1 **NPARAMB, NNODESBOUNDARY**

The next **NNODESBOUNDARY** lines contain the boundary conditions data

IN, DUMMY1, DUMMY2, DUMMY3, BCTYPE, BCFILENAME

*Example of the *.IFL file*

```
5   131
1   162.22   0.   0.04    0 0
2   159.74   0.   0.04    0 0
3   156.96   0.   0.04    0 0
4   163.57   0.   0.04    6 qsec94.qvt
5   198.72   0.   0.04    6 qsec94.qvt
6   190.19   0.   0.04    0 0
18  171.54   0.   0.04    0 0
20  170.77   0.   0.04    6 qsec94.qvt
22  163.56   0.   0.04    6 qsec94.qvt
24  159.88   0.   0.04    6 qsec94.qvt
26  160.91   0.   0.04    6 qsec94.qvt
29  179.82   0.   0.04    0 0
33  168.77   0.   0.04    0 0
99  170.03   0.   0.04    0 0
.
.
.
167 157.14   0.   0.04    1 hsec52.hvt
169 155.55   0.   0.04    1 hsec52.hvt
171 153.06   0.   0.04    1 hsec52.hvt
173 154.98   0.   0.04    1 hsec52.hvt
```

*Variable Descriptions for the *.IFL File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
BCTYPE	I	0-10	-	Code to indicate type of open boundary. See table below. See Comment 1.
BCFILENAME	S	-	-	Boundary condition file name. Can be any valid file name. See comments 2 and 3.
NNODESBOUNDARY	I	1-5	-	Total number of nodes on boundary.
NPARAMB	I	1-5	-	Number of node parameters.

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

BCTYPE	DESCRIPTION
0	Closed impermeable boundary. Slip boundary condition (no normal flow) is imposed. See comment 5.
1	Imposes Water Surface Elevation. An associated boundary condition file must be provided. See comments 2 and 4.
2	Imposes U velocity component. An associated boundary condition file must be provided. See comment 2.
3	Imposes V velocity component. An associated boundary condition file must be provided. See comment 2.
4	Imposes U and V velocity components. A two-variable boundarcondition file must be provided. See comment 3.
5	Imposes water discharge and water surface elevation. A two-variable boundary condition file must be provided. See comment 3.
6	Imposes water discharge. An associated boundary condition file must be provided. See comment 2.
7	Imposes U velocity and water surface elevation. A two-variable boundary condition file must be provided. See comment 3.
8	Imposes V velocity and water surface elevation. An associated boundary condition file must be provided. See comment 3.
9	Imposes single-valued stage-discharge rating table. An associated boundary condition file must be provided. See comment 6.
10	“Free” inflow or outflow condition. Velocities and water surface elevations are calculated by the model. See comment 7.

*Instructional Comments for the *.IFL File*

The following comments will assist in the developing of the *.IFL file:

1. RiverFLO-2D 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 RiverFLO-2D 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. The table below helps determining which conditions to use for most applications.

Flow regime at boundary	Inflow boundary condition	Outflow boundary condition
Subcritical	Q or Velocity	Water Surface Elevation
Supercritical	Q and WSE	"Free"

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

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

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

4. When imposing water surface elevation it is important to check that the imposed value is higher than the bed elevation. Even though the RiverFLO-2D model may run with that condition, it may lead to volume conservation errors.

5. When Argus ONE is used to create RiverFLO-2D data files 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

and then imposes zero-flow across the boundary. Tangential flow is free corresponding to a slip condition. This condition can be overridden using any of the other open inflow or outflow conditions described herein.

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. Most small slope rivers, the stage-discharge relationship is affected by hysteresis. In other words, the stage-discharge curve is looped with higher discharges occurring on the rising limb than on the rescission limb of the hydrograph. This is mainly caused by the depth gradient in the flow direction that changes in sign throughout the hydrograph. In practice, this implies that there can be two possible stages for the same discharge. If the stage-discharge relationship is not well known or if it just computed assuming steady state uniform flow, it may lead to considerable errors when used as downstream boundary condition. That it is why it is often preferred to use the stage hydrograph for that purpose. However, such hydrograph may not be available to study changes in the river and evaluating proposed conditions. For those cases, it is useful to use a stage-discharge relationship, preferably measured over an extensive range of discharges.
   When this relationship is not available, one option would be to assume steady state flow to determine a single-value rating curve. Since this condition may generate wave reflection that can propagate upstream, it is important to locate the downstream boundary on a reach sufficiently far from the area of interest, therefore minimizing artificial backwater effects. Unfortunately, there is no general way to select such place, but numerical experimenting with the actual model will be necessary to achieve a reasonable location.
   NOTE: *loop stage-discharge relationships are not allowed in this version.*

7. On free condition boundary nodes, the model calculates velocities and water surface elevations applying the full equations from the internal elements. No condition is imposed *per se* on these nodes, which on the finite element context, is equivalent to apply natural boundary conditions. In practice this should be equivalent to assume that derivatives of water surface elevations and velocities are 0. Use advisable to use this condition when there is at least another open boundary where WSE or stage-discharge is imposed.

*FILE: *.PLT: GRAPHICAL OUTPUT CONTROL DATA*

Line 1 contains plot control variables

1 **IGRAPHCODE, COLORSCHEME, IAXES, IDXF, IGRAPHFILES**

Line 2 contains velocity vector scale multiplier.

2 **SF_MULT**

Line 3 contains coordinates for plot window.

3 **XMING, XMAXG, YMING, YMAXG**

Line 4 contains limits of plotted variable.

4 **MINVARG, MAXVARG**

Line 5 contains maximum velocity to plot.

5 **MAXVELOC**

*Example of the *.PLT file*

```
100 5 1 0 1
5
0 0 0 0
0 0
7
```

*Variable Descriptions for the *.PLT File*

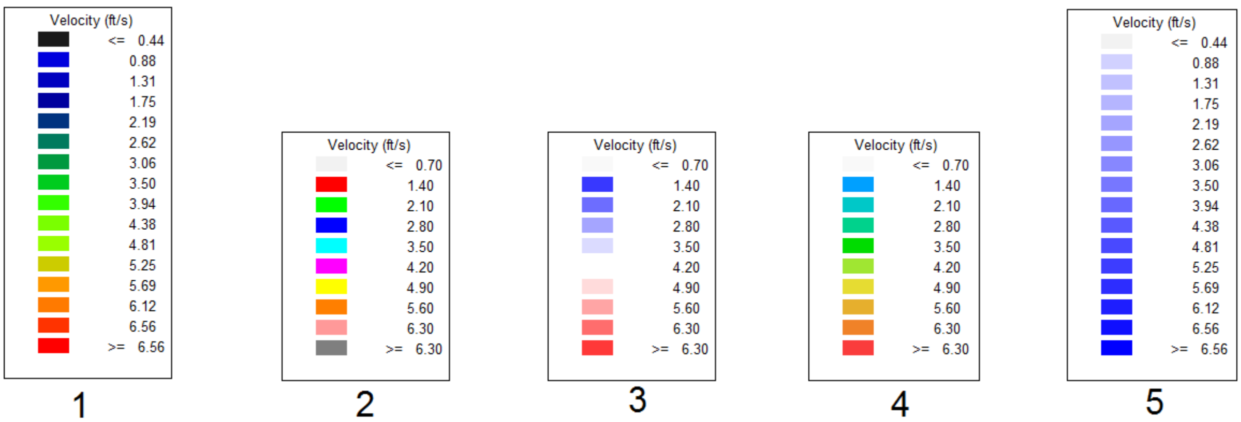
VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
COLORSCHEME	I	1-5	-	Variable to select plot option. See comment 1.
IAXES	I	0,1	-	Switch to control weather to plot axes. 0: Do not plot X and Y axes. 1: Plot X and Y axes.
IDXF	I	0,1	-	Switch to control velocity field output in DXF CAD format. 0: Do not output DXF velocity field. 1: Create velocity field DXF files for each output time.
IGRAPHFILES	I	0,1	-	Variable to control weather to output graphic files. 0: Do not output graphic files. 1: Output graphic files.
MAXVELOC	R	-	m/s or ft/s	Use this variable to control the maximum velocity displayed in vector plots. If MAXVELOC = 0, the whole velocity range will be plotted. If MAXVELOC > 0, it will define the maximum velocity to be displayed.
MINVARG, MAXVARG	R	-	-	These variables define the minimum and maximum values to be displayed of the selected variable. If equal to 0, the maximum range will be displayed.
SF_MULT	R	>0	-	Variable to control velocity vector scale. Use this variable to adjust velocity vectors. Velocities will be scaled according to SF_MULT.
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 are will be displayed.

Note: I = Integer variable. R = Real variable

*Instructional Comments for the *.PLT File*

1. COLORSCHEME defines the color palette that will be used for all plots. The available palettes are shown in this figure.

.



Color palettes.

*FILE: *.PRF: PROFILE CUT DATA FOR RESULT OUTPUT*

Use this file to provide profiles (polylines) along which results will be provided.

Line 1 Number of profiles and number of intervals to divide each profile.

1 **NPROFILES, ND_PR**

For each profile I, write number of vertices in profile I and list coordinates for each vertex in polyline. There should be NPROFILES groups like this one:

NVERTICES_PR(I)
X_PRF(I), Y_PRF(I)
.
.

*Example of the *.PRF file*

2 200
2
800500.45 306895.63
799095.07 307457.34
3
800503.45 306896.63
799500.00 306900.00
799095.07 307457.34

This file indicates there is data for 2 profiles, each will be divided in 200 parts. The first profile is defined by 2 vertices and the second profile is defined by 3 vertices.

*Variable Descriptions for the *.PRF File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
ND_PR	I	>2	-	Intervals to divide each profile. Results will be reported at each interval.
NPROFILES	I	>0	-	Number of cross sections.
NVERTICES_PR(I)	I	>1	-	Number of vertices in each profile.
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

FILE: *.REP: CROSS SECTION DATA FOR RESULT OUTPUT

Line 1 Number of cross sections and number of intervals to divide each section.

1 **NCROSS_SECTIONS, ND_CS**

For each cross section I, coordinates of initial and final point in cross section. There should be NCROSS_SECTIONS lines like this one:

X1_CS(I), Y1_CS(I), X2_CS(I), Y2_CS(I)

*Example of the * .REP file*

```
12 40
800500.45 306895.63  799095.07 307457.34
800492.17 307163.36  799171.99 307594.56
800449.99 307404.31  799223.97 307690.20
800456.79 307736.80 799226.05 307729.70
800467.07 308012.96  799325.84 307856.52
800463.75 308460.92 799325.84 307896.03
800256.02 309010.15  799244.76 308051.96
800441.71 309664.04 7 99113.78 308284.81
800175.75 309705.15  799038.93 308528.06
800016.40 309821.89  798968.24 308918.92
799633.54 309854.50  798828.95 309112.27
800631.40 305589.90  798349.30 307285.18
.
.
```

*Variable Descriptions for the *.REP File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NCROSS_SECTIONS	I	>0	-	Number of cross sections.
ND_CS	I	>2	-	Intervals to divide each section. Results will be reported at each interval.
X1_CS, Y1_CS, X2_CS, Y2_CS	R	-	m or ft	Coordinates of initial and ending point of each cross section.

Note: I = Integer variable. R = Real variable

*FILE: *.RET: RAINFALL AND EVAPORATION DATA*

Line 1 Number of points in time series of rainfall and evaporation

1 **NPRE**

Line 2 Time, Daily Rainfall, Daily Evaporation

2 **TIME, RAINFALL, EVAP**

*Example of the *.RET file*

```
10
0.   0.0   0.0
24.  4.0  0.0
48   12.0 0.0
.
.
.
```

*Variable Descriptions for the *.RET File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
EVAPORATION	R	>=0	mm/day or in/day	Daily evaporation. Should be given for each 24 hour interval
NPRE	I	-	-	Number of times in rainfall and evaporation time series.
RAINFALL	R	>=0	mm/day or in/day	Daily rainfall. Should be given for each 24 hour interval.
TIME	R	>=0	hours	Time should be given for each 24 hour interval

Note: I = Integer variable. R = Real variable

*FILE: *.SED: SEDIMENT TRANSPORT DATA*

Line 1 Sediment discharge formula.

1 **ST_FORMULA**

Line 2 Median sediment size in mm, sediment specific gravity and porosity

2 **D50, GR, POROS**

*Example of the *.SED file*

1
0.01 2.65 0.4

*Variable Descriptions for the *.SED File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
D50	R	>0	mm	Sediment median size. 50% of the sediment is finer than D50.
GR	R	2-2.7	-	Sediment particles specific gravity.
POROS	R	0.3-0.6	-	Sediment porosity.
ST_FORMULA	I	1-3	-	Code indicating the sediment transport rate to use. The options available in this version are: 1: Meyer-Peter & Muller (1948), 2: Karim-Kennedy (1998), 3: Ackers-White (1975). 4: Yang (Sand), 5: Yang (Gravel), 6: Parker-Klingeman-Mclean (1982), 7: Van Rijn (1984a-c), 8: Engelund Hansen (1967).

Note: I = Integer variable. R = Real variable

*FILE: *.TBA: MESH BOUNDARY DATA*

***NOTE**: It is recommended to use Argus ONE program to create this file. The templates for RiverFLO-2D provided with the Argus ONE program assures that the file will be created error free and consistent with the boundary conditions and other mesh parameters. Manually editing this file may introduce errors.*

Line 1 contains new boundary parameter.

1 **IBOUNDARYID**

Line 2 contains the number of nodes in external boundary of mesh.

2 **NNODESBOUNDARY**

Next NNODESBOUNDARY lines contain the list of boundary nodes in counter clockwise direction.

3 **BOUNDARYNODE (1:NNODESBOUNDARY)**

The next lines are only used if there are islands in the mesh. Repeat for each island.

Next line contains the new boundary parameter indicator for each island or internal closed contour.

4 **IBOUNDARYID**

Line 2 contains the number of nodes in island external boundary.

2 **NNODESISLANDBOUNDARY**

Next NNODESISLANDBOUNDARY lines contain the list of boundary nodes in clockwise direction.

3 **ISLANDBOUNDARYNODE (1: NNODESISLANDBOUNDARY)**

*Example of the *.TBA file*

```
-9999
132
1
2
3
173
171
.
.
.
244
240
236
232
228
224
175
1
-9999
34
5
7
45
66
.
.
.
5
```

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

*Variable Descriptions for the *.TBA File*

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
IBOUNDARYID	I	-99999	-	Always = -99999. This is to indicate the program that the following lines contain a boundary.
NNODESBOUNDARY	I	>0	-	Number of nodes on external boundary.
BOUNDARYNODE	I	>0	-	Node number on external boundary. See comment 1.
NNODESISLANDBOUNDARY	I	>0	-	Number of nodes on island boundary.
ISLANDBOUNDARYNODE	I	>0	-	Node number on island boundary.

Note: I = Integer variable. R = Real variable

*Instructional Comments for the *.TBA File*

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

BOUNDARY CONDITIONS DATA FILES

One variable boundary condition.

This format applies to the following data files:

- Time vs Water Surface Elevation (BCTYPE = 1)
- Time vs Discharge (BCTYPE = 6)
- Time vs U velocity component in x direction (BCTYPE = 2)
- Time vs V velocity component in y direction (BCTYPE = 3)

Line 1 Number points in data series.

1 **NDATA**

For each time I

TIME(I) VARIABLE(I)

Where VARIABLE(I) is WSE, Q, U or V depending on the boundary condition

Example of a Boundary Condition file for one variable time series

The following example shows an inflow hydrograph where NDATA is 10 and there are 10 lines with pairs of time and discharge:

```
10
  0.      20.
  1.      30.
  1.3     50.
  2 .     90.
  4 .    120.
  5.     200.
  7.     250.
  8.1    110.
  10.     60.
  20.     20.
```

Variable Descriptions of a Boundary Condition File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	R	> 0	-	Number points in data series.
VARIABLE	R	-	-	Represents Water Surface Elevation, Discharge, U or V velocity components depending on the boundary condition.

Note: I = Integer variable. R = Real variable

Two variables boundary condition.

This format applies to the following data files:

- Time vs U and V velocity components (BCTYPE = 4)
- Time vs Discharge and Water Surface Elevation (BCTYPE = 5)
- Time vs U velocity component and Water Surface Elevation (BCTYPE = 7)
- Time vs V velocity component and Water Surface Elevation (BCTYPE = 8)

Line 1 Number points in data series.
2 **NDATA**

For each time I
TIME(I) VARIABLE1(I) VARIABLE2(I)

Where VARIABLE1(I) and VARIABLE(2) depend on the boundary condition type as follows:

BCTYPE	VARIABLE1	VARIABLE2
4	U	V
5	Q	WSE
7	U	WSE
8	V	WSE

Example of a 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.
    2 .     90.     1425.
    4 .    120.     1427.
    5.     200.     1428.
    7.     250.     1420.
    8.1    110.     1426.
    10.     60.     1423.5
    20.     20.     1421.
```

Variable Descriptions of a Two-Variable Boundary Condition File

VARIABLE	TYPE	RANGE	UNITS	DESCRIPTION
NDATA	R	>0	-	Number points in data series.
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: R = Real variable

Stage-discharge data file.

This format applies to the stage (water surface elevation) vs. discharge table used in BCTYPE = 9.

Line 1 Number points in data series.

NDATA

For each table entry

STAGE(I) Q(I)

Where STAGE(I) is Water Surface Elevation and Q(I) is the corresponding discharge.

Example of a Stage-Discharge data file

The following example shows an inflow hydrograph where NDATA is 10 and there are 10 lines with pairs of time and 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
```

Output Files

RiverFLO-2D generates extensive output data into a series of ASCII files. These ASCII files can be easily accessed with various text editor software and they can be import into the Argus ONE program and other GIS software for visualization and analysis. RiverFLO-2D always creates output ASCII files in both English and Metric units. The following sections describe the content of each output file. The following table summarizes the output files generated by RiverFLO-2D:

Data File Extension	Content
OUT 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. OUT file is in metric units and OUTE file in English units.
UVH UVHE	For each output interval nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number are written to file. UVH is in metric units and UVHE in English units.
Time Series OUT FILES	For user selected nodes a time series of velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number is written to file. File name format is: RESvsT_NODE_NODENUMBER.OUT for metric units and RESvsT_NODE_ENG_NODENUMBER.OUT for English units.
EXP	For each output interval nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number are written to file. These files can be directly imported into an Argus ONE data layer for post processing. The file names are: TIME_METRIC_DDDD_HH_MM_SS.EXP. for metric units and TIME_ENG_DDDD_HH_MM_SS.EXP for English units.
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.
UVHB VTP	Binary files that allow hot starts of previous simulations.
XSEC XSECE	For output interval and user defined cross section, these files provide bed elevation, depth, water surface elevation, depth average velocity, and Froude number. XSEC is in metric and XSECE in English units.
DXF	Velocity field using colored vectors in AutoCAD DXF format. File name is in this format: VelField_TIME_ DDDD_HH_MM_SS.DXF.
GIF	Raster file plot for each report time interval.

UVH and UVHE Output Files

These files report the output time interval nodal velocities, depths, water surface and bed elevations, bed elevation changes, wet-dry condition, Froude number. UVH is in metric units and UVHE in English units. A typical output UVHE file is shown below:

RESULTS FOR TIME (DDDD:HH:MM:SS): 0000:00:15:00

NODE	Ux (ft/s)	Uy (ft/s)	VELOCITY (ft)	WSURFELEV (ft)	DEPTH (ft)	ZBED_ORI (ft)	ZBED (ft)	DELTA_ZBED (ft)	DRY-WET	FROUDE
1	-2.092	-3.104	3.743	167.730	5.730	162.000	162.000	0.000	WET	0.28
2	0.418	-1.442	1.502	167.125	7.376	159.750	159.750	0.000	WET	0.10
3	3.523	-1.830	3.970	168.113	11.153	156.960	156.960	0.000	WET	0.21
4	0.000	0.000	0.000	187.361	23.784	163.576	163.576	0.000	WET	0.00
5	0.000	0.000	0.000	198.724	0.000	198.724	198.724	0.000	DRY	0.00
6	0.000	0.000	0.000	190.195	0.000	190.195	190.195	0.000	DRY	0.00
7	0.000	0.000	0.000	165.595	0.864	164.731	164.731	0.000	WET	0.00
8	0.000	0.000	0.000	165.495	1.574	163.921	163.921	0.000	WET	0.00
9	0.000	0.000	0.000	167.444	0.000	167.444	167.444	0.000	DRY	0.00
10	-0.625	0.325	0.705	165.377	3.533	161.844	161.844	0.000	WET	0.07
11	1.032	-0.097	1.036	181.547	4.246	177.301	177.301	0.000	WET	0.09
12	0.589	-0.145	0.607	181.519	4.547	176.972	176.972	0.000	WET	0.05
13	0.561	0.229	0.606	181.373	6.553	174.820	174.820	0.000	WET	0.04
14	0.842	-0.937	1.259	181.482	4.098	177.384	177.384	0.000	WET	0.11
15	1.221	-0.090	1.225	181.443	8.717	172.726	172.726	0.000	WET	0.07
16	0.272	0.111	0.294	181.489	8.939	172.551	172.551	0.000	WET	0.02
17	2.325	-0.156	2.330	181.307	10.146	171.160	171.160	0.000	WET	0.13
18	2.582	1.056	2.790	180.273	8.726	171.548	171.548	0.000	WET	0.17
19	6.491	-4.355	7.817	181.127	11.692	169.435	169.435	0.000	WET	0.40
20	3.586	-8.767	9.472	182.657	11.887	170.771	170.771	0.000	WET	0.48
21	8.212	-8.648	11.926	182.395	15.041	167.354	167.354	0.000	WET	0.54
22	6.305	-7.069	9.472	184.681	21.112	163.570	163.570	0.000	WET	0.36
23	6.656	-7.878	10.313	182.726	21.168	161.558	161.558	0.000	WET	0.40
24	6.305	-7.069	9.472	182.736	22.855	159.881	159.881	0.000	WET	0.35

OUT and OUTE Output 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. OUT is in metric units and OUTE in English units. Part of a typical OUT file format is shown below:

```
  MODELING OPTIONS

  Model Rainfall/Evaporation: NO.
  Model Sediment Transport:   NO.
  Output for cross sections:  ACTIVATED
  Output for profile:         ACTIVATED.
  Use variable time-steps:    NO.
  Use constant initial water surface elevation.
  Start simulation from time = 0.

  Time-step = 0.50 s.
  Report Interval =  0.00 hr.
  Simulation Time =  0.00 hr.
  Text Output:    ACTIVATED.
  Graphical Output:    ACTIVATED.
  Manning n Multiplier:  1.00
  Selective Lumping Parameter:  0.90
  Input units:    ENGLISH.
  Depth tolerance for dry bed: 0.300 m.

   LIST OF NODES FOR TIME SERIES OUTPUT

          1
         23
         34

  MESH DATA

 NUMBER OF ELEMENTS:      1231
 NUMBER OF NODES:          682

    NODE        X           Y         ZBED    Initial WSE
               (m)         (m)         (m)        (m)
     1   243695.39    93202.88     49.41      61.11
     2   243679.37    93179.12     48.72      61.11
     3   243706.33    93165.12     47.87      61.11
     4   243559.79    94418.96     49.89      61.11
     5   243541.88    94402.99     60.61      61.11
     6   243560.86    94382.46     58.01      61.11
```

PRFI and PRFE Output Files

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:

```
PROFILE RESULTS IN ENGLISH UNITS
TIME: 0000 days, 00 hours, 15 min.  ,00 secs.
 ELEM   DISTANCE   BEDELEV     DEPTH   SURFELEV     VELP     FROUDE
  258     37.84    167.94       4.13   172.06       2.82       0.25
  258     75.67    166.42       5.79   172.21       2.82       0.21
  261    113.51    162.53       9.95   172.48       4.73       0.26
  532    151.35    160.21      12.49   172.70       6.95       0.35
  535    189.18    158.15      14.66   172.81       8.22       0.38
  684    227.02    156.89      15.91   172.80       8.42       0.37
  684    264.86    156.67      16.17   172.85       8.42       0.37
  687    302.70    156.26      16.47   172.73       8.44       0.37
  839    340.53    155.60      17.08   172.68       8.38       0.36
  842    378.37    154.78      17.91   172.69       7.94       0.33
  974    416.21    153.90      18.84   172.74       7.56       0.31
  974    454.04    153.57      19.17   172.74       7.56       0.30
  977    491.88    153.79      18.95   172.74       6.33       0.26
  978    529.72    156.40      16.38   172.77       6.33       0.28
 1223    567.55    160.09      12.65   172.74       5.67       0.28
 1225    605.39    163.53       9.01   172.54       5.38       0.32
 1225    643.23    165.19       7.28   172.47       5.38       0.35
 1224    681.06    167.26       5.24   172.50       5.38       0.41
 1231    718.90    168.29       4.42   172.72       6.10       0.51
 1230    756.74    167.94       5.03   172.96       5.44       0.43
 1230    794.58    167.34       5.75   173.10       5.44       0.40
 1070    832.41    166.58       6.55   173.13       5.44       0.38
  610    870.25    166.42       6.84   173.26       5.74       0.39
  607    908.09    166.74       6.69   173.44       5.02       0.34
  607    945.92    167.14       6.38   173.53       5.02       0.35
```

XSEC and XSECE Output Files

For output interval and for each user defined cross sections the bed elevation, depth, water surface elevation, depth average velocity, and Froude number is written to file. XSEC is in metric and XSECE in English units. A portion of a typical XSEC file is as follows:

```
 CROSS SECTION RESULTS IN SI UNITS
 TIME: 0000 days, 00 hours, 15 min. ,00 secs.

 CROSS SECTION :        1
( 244152.64,   93603.17), ( 243724.00,   93774.49)
  ELEM    DISTANCE    BEDELEV      DEPTH    SURFELEV      VELP     FROUDE
   258       11.54      51.22       1.26       52.48      0.86       0.25
   258       23.08      50.76       1.76       52.52      0.86       0.21
   261       34.62      49.57       3.04       52.61      1.44       0.26
   532       46.16      48.86       3.81       52.67      2.12       0.35
   535       57.70      48.24       4.47       52.71      2.51       0.38
   684       69.24      47.85       4.85       52.70      2.57       0.37
   684       80.78      47.79       4.93       52.72      2.57       0.37
   687       92.32      47.66       5.02       52.68      2.58       0.37
   839      103.86      47.46       5.21       52.67      2.56       0.36
   842      115.40      47.21       5.46       52.67      2.42       0.33
   974      126.94      46.94       5.75       52.69      2.31       0.31
   974      138.48      46.84       5.85       52.69      2.31       0.30
   977      150.02      46.90       5.78       52.69      1.93       0.26
   978      161.56      47.70       4.99       52.70      1.93       0.28
  1223      173.10      48.83       3.86       52.69      1.73       0.28
  1225      184.64      49.88       2.75       52.63      1.64       0.32
  1225      196.18      50.38       2.22       52.60      1.64       0.35
  1224      207.72      51.01       1.60       52.61      1.64       0.41
  1231      219.26      51.33       1.35       52.68      1.86       0.51
  1230      230.81      51.22       1.53       52.75      1.66       0.43
  1230      242.35      51.04       1.75       52.79      1.66       0.40
  1070      253.89      50.81       2.00       52.80      1.66       0.38
   610      265.43      50.76       2.09       52.84      1.75       0.39
   607      276.97      50.86       2.04       52.90      1.53       0.34
   607      288.51      50.98       1.95       52.93      1.53       0.35
```

TIME SERIES OUT OUTPUT FILES

For user selected nodes (INTS variable in DAT file), these files report the time series of velocities, depths, water surface and bed elevations, bed elevation changes, and Froude number. The file name is:

RESvsT_NODE_NODENUMBER for metric units and,
RESvsT_NODE_ENG_NODENUMBER for English units.

An example is shown below.

```
RESULTS FOR NODE            1
LOCATED AT COORDINATE: (   243695.39),(     93202.88)

  TIME         U         V       VELOC      DEPTH    WSURFELEV  ZBED_ORI     ZBED   DELTA_BED  FROUDE
 (hours)     (m/s)     (m/s)     (m/s)      (m)        (m)        (m)        (m)      (m)
 0.25000    -0.638    -0.947     1.142      1.748     51.158     49.410     49.410    0.000     0.276
 0.50000    -0.543    -0.805     0.971      1.562     50.972     49.410     49.410    0.000     0.248
 0.75000    -0.525    -0.780     0.940      1.601     51.011     49.410     49.410    0.000     0.237
 1.00000    -0.511    -0.758     0.914      1.648     51.058     49.410     49.410    0.000     0.227
 1.25000    -0.496    -0.736     0.887      1.696     51.106     49.410     49.410    0.000     0.218
 1.50000    -0.482    -0.715     0.862      1.746     51.156     49.410     49.410    0.000     0.208
 1.75000    -0.469    -0.695     0.838      1.797     51.207     49.410     49.410    0.000     0.200
 2.00000    -0.455    -0.676     0.815      1.849     51.259     49.410     49.410    0.000     0.191
```

UVHB and VTP Output Files

These are binary files for model internal use and contain the simulation results at the end of each output interval. These files are then used to restart a simulation from the completion time (hot start option). For example, the user stops the simulation at 4.25 hours to review results. The RiverFLO-2D can then be restarted from the last report output time interval by reading the last state from the VTP file. UVHB file also serves to save all model results for each report time interval as redundancy storage.

NOTE: *UVHB and VTP files are in binary format and are not readable by a text editor.*

DXF FILES

These files contain the velocity field for each output interval using colored vectors in AutoCAD DXF format. The file name is of this form: VelField_TIME_ DDDD_HH_MM_SS.

*Argus ONE ASCII files *.EXP*

The ASCII files allow seamless transfer to Argus ONE data layers. These files use the *.EXP extension and are named as follows:

For English units: TIME_ENG_DDDD_HH_MM_SS.EXP

For Metric units: TIME_METRIC_ DDDD_HH_MM_SS.EXP

Where DDDD is days, HH is hours, MM is minutes and SS seconds. For example TIME_ENG_0001_12_01_34.EXP corresponds to a file in English units for time: 1 day, 12 hours, 1 minute and 34 seconds.

The format for these files is as follows. The first line indicates the number of node parameters (10 by default). Following, there is a line with results for each node in the finite element mesh as shown:

COLUMN	VALUE	ENGLISH UNITS	METRIC UNITS
1	Node number	-	-
2	Velocity component in x direction U	ft/s	m/s
3	Velocity component in y direction V	ft/s	m/s
4	Velocity magnitude SQRT(U^2 + V^2)	ft/s	m/s
5	Water surface elevation	ft	m
6	Depth	ft	m
7	Initial bed elevation	ft	m
8	Bed elevation	ft	m
9	Variation in bed elevation since time = 0	ft	m
10	Dry or wet	Dry=1 / Wet=0	Dry=1 / Wet=0
11	Froude number	-	-

An example of a typical Argus ONE ASCII EXP file is as follows:

10

1	-2.092	-3.104	3.743	167.730	5.730	162.000	162.000	0.000
2	0.418	-1.442	1.502	167.125	7.376	159.750	159.750	0.000
3	3.523	-1.830	3.970	168.113	11.153	156.960	156.960	0.000
4	0.000	0.000	0.000	187.361	23.784	163.576	163.576	0.000
5	0.000	0.000	0.000	198.724	0.000	198.724	198.724	0.000
6	0.000	0.000	0.000	190.195	0.000	190.195	190.195	0.000
7	0.000	0.000	0.000	165.595	0.864	164.731	164.731	0.000
8	0.000	0.000	0.000	165.495	1.574	163.921	163.921	0.000
9	0.000	0.000	0.000	167.444	0.000	167.444	167.444	0.000
10	-0.625	0.325	0.705	165.377	3.533	161.844	161.844	0.000
11	1.032	-0.097	1.036	181.547	4.246	177.301	177.301	0.000
12	0.589	-0.145	0.607	181.519	4.547	176.972	176.972	0.000
13	0.561	0.229	0.606	181.373	6.553	174.820	174.820	0.000
14	0.842	-0.937	1.259	181.482	4.098	177.384	177.384	0.000
15	1.221	-0.090	1.225	181.443	8.717	172.726	172.726	0.000
16	0.272	0.111	0.294	181.489	8.939	172.551	172.551	0.000
17	2.325	-0.156	2.330	181.307	10.146	171.160	171.160	0.000
18	2.582	1.056	2.790	180.273	8.726	171.548	171.548	0.000
19	6.491	-4.355	7.817	181.127	11.692	169.435	169.435	0.000
20	3.586	-8.767	9.472	182.657	11.887	170.771	170.771	0.000

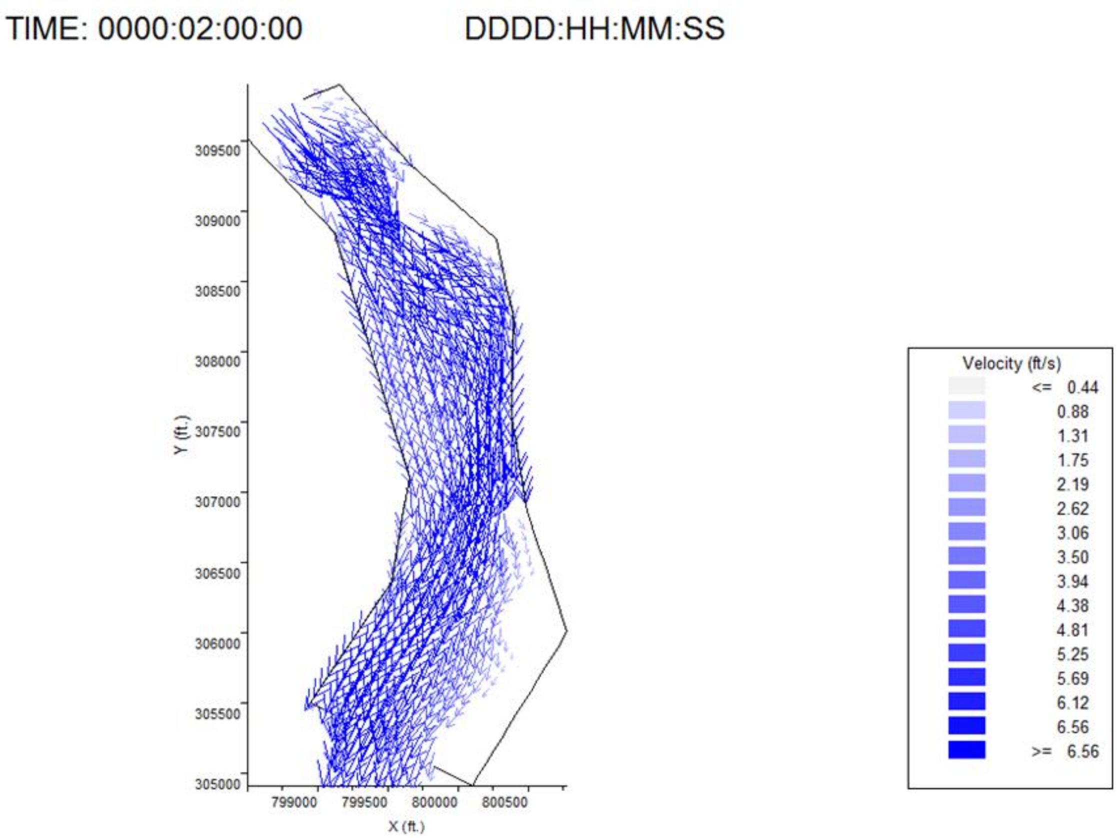
.

.

.

GIF Output Files

These are raster output files showing the plot for each output interval. They can be imported in other word, spreadsheet or presentation programs.



References

Ackers, P., and W.R. White, Sediment transport, new approach and analysis*, J. Hyd. Div.* ASCE, 99, HY 11, 1975

Engelund, F. and Hensen, E. *A Monograph on Sediment Transport to Alluvial Streams*. Copenhagen: Teknique Vorlag, 1967.

Garcia, R., Espinoza, R., Valera, E., and Gonzalez, M. (2006). An explicit two-dimensional finite element model to simulate short- and long-term bed evolution in alluvial rivers*. J. Hyd. Res.* (44), No. 6, 755–766.

Karim, F. Bed material discharge prediction for non-uniform bed sediments. *J. Hyd. Eng.,* 6, 1998.

Meyer-Peter, E. and Muller, R. *Formulas for Bed-load Transport*. Proc. of the Seconds Meeting. IAHR, Stockholm, Sweden, pp. 39–64. 1948.

Parker, G.; Klingeman, P. C.; McLean, D. G. Bedload and size distribution in paved gravel bed streams. Journal of Hydraulic Engineering. 108(4): 544-571. 1982.

Van Rijn, L.C., Sediment Transport, Part I: Bed load transport, *J. Hyd. Eng.*, ASCE, no 10. 1984a

Van Rijn, L.C., Sediment Transport, Part II: Suspended load transport, *J. Hyd. Eng.*, ASCE, no 11. 1984b

Van Rijn, L.C., Sediment pick-up functions, *J. Hyd. Eng.*, ASCE, no 10. 1984c

Yang, C.T. *Sediment Transport Theory and Practice*. McGraw Hill, New York, 1996.