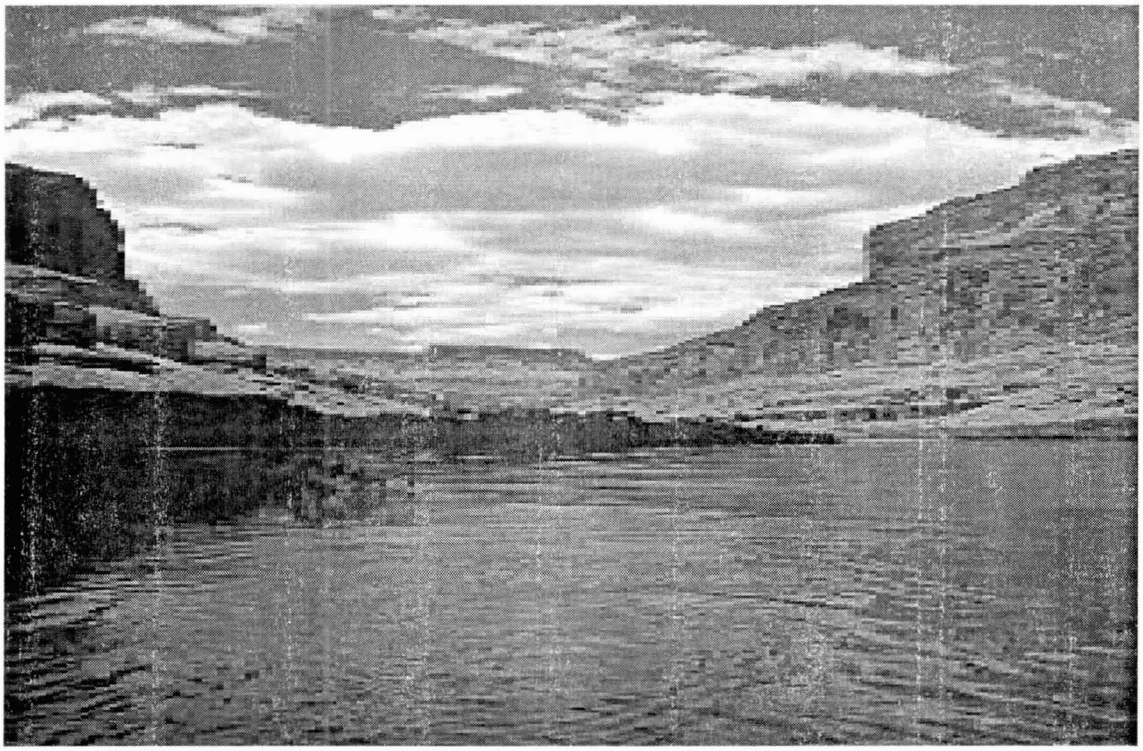
US Army Corps
of Engineers
Professional Development
Support Center, Huntsville

PROSPECT

PROPONENT SPONSORED ENGINEER CORPS TRAINING

Unsteady Flow Analysis with HEC-RAS for Maricopa County

27 – 29 November 2007



Unsteady Flow Analysis with HEC-RAS for Maricopa County

27 – 29 November 2007

UNSTEADY FLOW ANALYSIS USING HEC-RAS FOR MARICOPA COUNTY

AGENDA INDEX

Tuesday

Type	Title	Tab	Prefix	Instructor
Lecture	Overview of HEC-RAS	1.1	L-01	Brunner
Lecture	Data Requirements for Unsteady Flow Models	1.2	L-02	Ackerman
Workshop	Pre-Processing Geometric Data	1.3	W-01	Ackerman/Brunner
Lecture	Boundary and Initial Conditions	1.4	L-03	Brunner
Workshop	Basic Unsteady Flow Modeling	1.5	W-02	Brunner/Ackerman

Wednesday

Lecture	Modeling Bridges and Culverts	2.1	L-04	Ackerman
Workshop	Modeling Bridges and Culverts	2.2	W-03	Ackerman/Brunner
Lecture	Inline and Lateral Hydraulic Structures	2.3	L-05	Brunner
Lecture	Storage Areas and Storage Area Connections	2.4	L-06	Ackerman
Workshop	Lateral Hydraulic Structures, Storage Areas, and Storage Area Connections	2.5	W-04	Brunner/Ackerman

Thursday

Lecture	Model Stability, Accuracy, and Sensitivity	3.1	L-07	Brunner
Workshop	Trouble Shooting	3.2	W-05	Brunner/Ackerman
Lecture	Advanced Features for Unsteady Flow	3.3	L-08	Ackerman
Workshop	Dam and Levee Breaching	3.4	W-06	Ackerman/Brunner

Cameron T. Ackerman:	Hydraulic Engineer, Hydrology and Hydraulics Technology Division, 530/756-1104/ext/ 358.
Education:	BS, Civil Engineering, University of California, Davis, 1997 MS, Civil Engineering, University of California, Davis, 1999
Registration/Professional Affiliations:	Registered Civil Engineer, California, 2001; Member ASCE/EWRI; AGU; Tau Beta Pi;.
Experience:	5 years, Hydraulic Engineer, Hydrology and Hydraulics Technology Division, HEC. Dam break modeling and river analysis studies; research, development, and application of geographic information systems (GIS) methods for surface water modeling; and development of GIS-based flood warning and response systems. 3 years, Hydraulic Engineer, Water Resource Systems Division and Planning Analysis Division, HEC. Research, development, and application of GIS methods for surface water modeling, flood damage analysis, and risk communication; riverine, wetland restoration, hydrology, and watershed impact analysis studies. 3 years, Civil Engineer Intern, HEC. Riverine and watershed studies; investigation of remote sensing techniques; effects of riverine wetlands for surface water hydrology; application of GIS methods for hydrologic analysis; development of GIS methods for surface water modeling. 2 seasons, Hydrology Crew Leader, USFS, Tahoe National Forest. Surveyed Sierra Nevada streams; fish and invertebrate sampling, movement of woody debris, channel characterization and measurement, Rosgen channel typing, monitor hydrologic impacts to watershed.
Technical Subjects:	River hydraulics, dam break analysis, surface water hydrology, and GIS for surface water modeling.

Gary W. Brunner: Senior Technical Hydraulic Engineer (River Hydraulics), GS-14, (530) 756-1104/ext. 352.

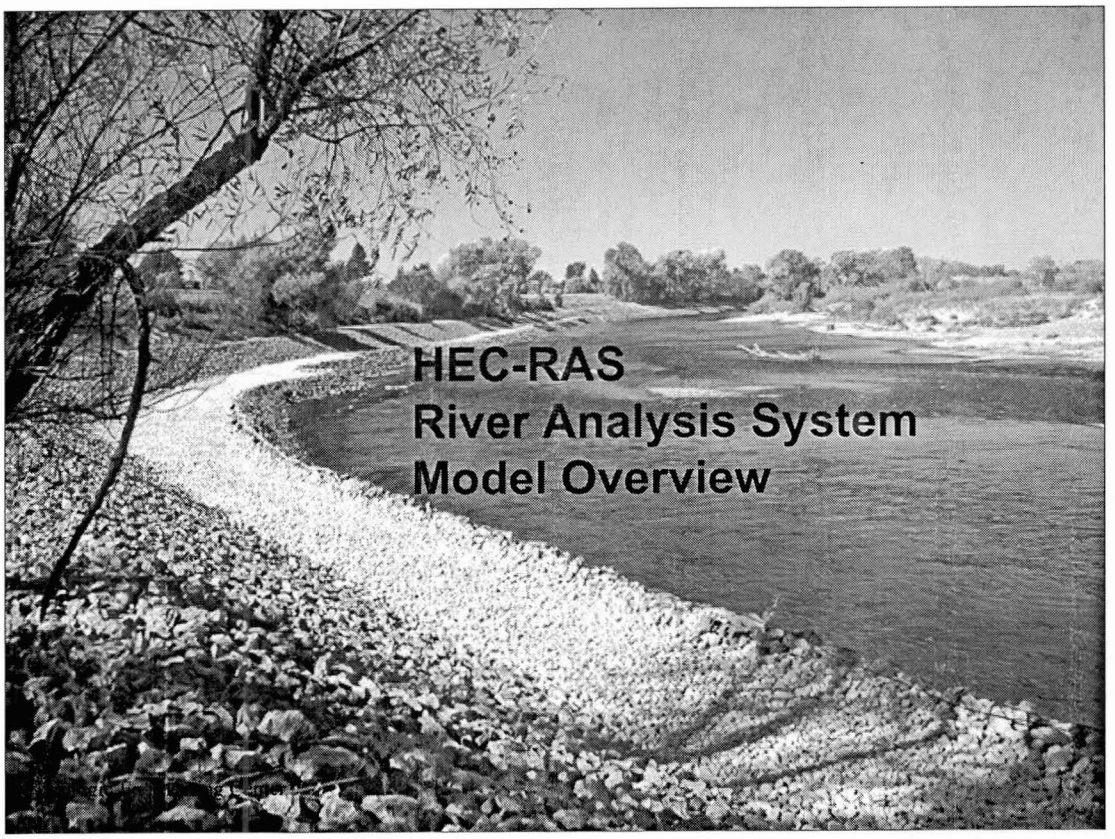
Education: BS, Civil Engineering, Pennsylvania State University, State College, PA, 1983
MS Civil Engineering, Pennsylvania State University, State College, PA, 1985

Registration/Professional Affiliations: Registered Civil Engineer, California, 1987; member Tau Beta Pi, Chi Epsilon, Transportation Research Board, FMA (Floodplain Management Association), EWRI, and A.S.C.E.

Experience: 21 years, Senior Technical Hydraulic Engineer, Hydrology and Hydraulics Technology Division, Hydrologic Engineering Center, IWR. In charge of the development of the HEC River Analysis System (HEC-RAS); perform research in the area of river hydraulics; provide technical assistance to Corps field personnel; teach engineering classes in river hydraulics and surface water hydrology; perform complex riverine studies.

2 years, Research and Teaching Assistant, Penn State University. Research on overland flow processes, and teaching surveying to undergraduates.

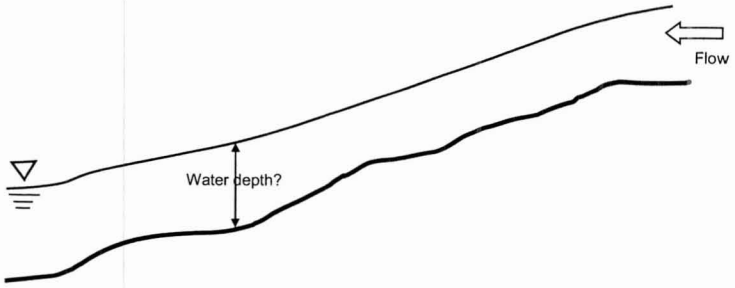
Technical Subjects: River hydraulics; steady flow, unsteady flow, and Dam Break analyses; surface water hydrology; risk and uncertainty analysis; sediment transport processes; real time river forecasting; hydrologic statistics; utilizing GIS with hydrology and hydraulic models; also have extensive teaching experience in river hydraulics and surface water hydrology.



This lecture is intended to be an overview of the HEC-RAS software.

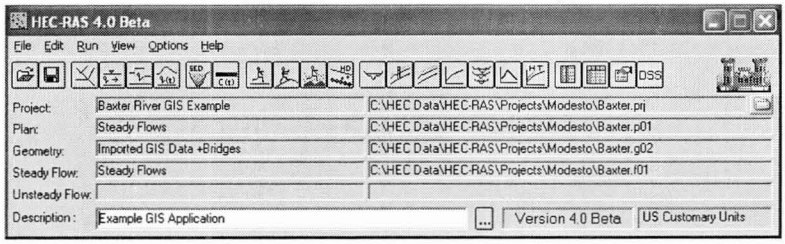
HEC-RAS

- 1-Dimensional hydraulics program
- Compute water surface profiles from channel geometry and flow
- Steady and Unsteady Flow



Hydrologic Engineering Center 2

HEC-RAS



- **Graphical User Interface**
- **Data storage/management**
- **Graphics, Tabular Output & Reporting**
- **GeoRAS – HEC-RAS GIS Interface**

Hydrologic Engineering Center 3

Type and Scale of Applications

- Types of Studies
  - Initial screening and reconnaissance studies
  - Detailed Investigations and alternative evaluations
  - Design Studies
  - Real-time forecasting
- Spatial Scale of Application
  - Very small urban areas with small drainage systems
  - Moderate-size river systems with natural and constructed channels
  - Large-scale river systems
- Time Scale of Applications
  - Peak flow profiles
  - Single event simulations
  - Long term simulations

Hydrologic Engineering Center 4

HEC-RAS has been designed for modeling a wide range of spatial and time scales. The software can be used to model from large scale river systems, down to very small streams (even flume data could be modeled). Time scales can range from modeling peak flows (steady flow), full hydrographs (unsteady flow), and even period of record analysis. HEC-RAS can be used to perform a range of studies from initial screening level studies, up to and including detailed hydraulic designs.

Steps in Developing a Hydraulic Model with HEC-RAS

- There are five main steps in creating an HEC-RAS model:
  - Starting a new project
  - Entering Geometric Data
    - Create or import geometry and add structures
  - Entering flow data
    - Steady flow (Peaks) or unsteady flow (hydrographs)
  - Creating a Plan and performing the computations
  - Viewing Results

Hydrologic Engineering Center 5

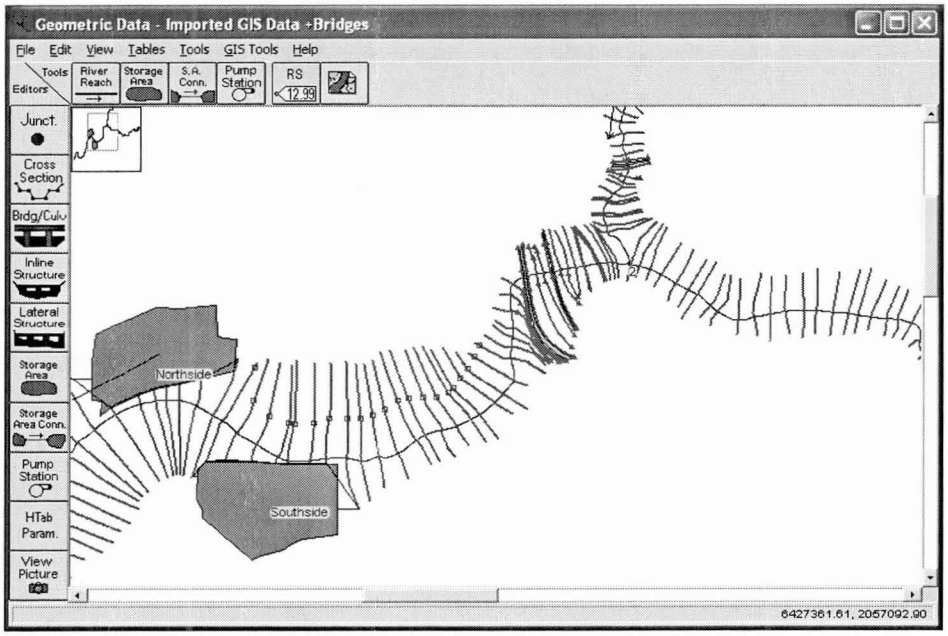
Geometric Data

- Cross Sections
- Detailed Bridge Analysis
- Detailed Culvert Analysis (9 shapes)
- Multiple openings (bridge, culverts, conveyance)
- Inline Structures (weirs/spillways, gates)
- Lateral Structures (weirs/spillways, gates, culverts)
- Storage Areas and Storage Area Connections
- Rating Curves
- Pump Stations
- Floating ice
- Levees
- Extensive data import and export
- GIS Connections

Hydrologic Engineering Center 6

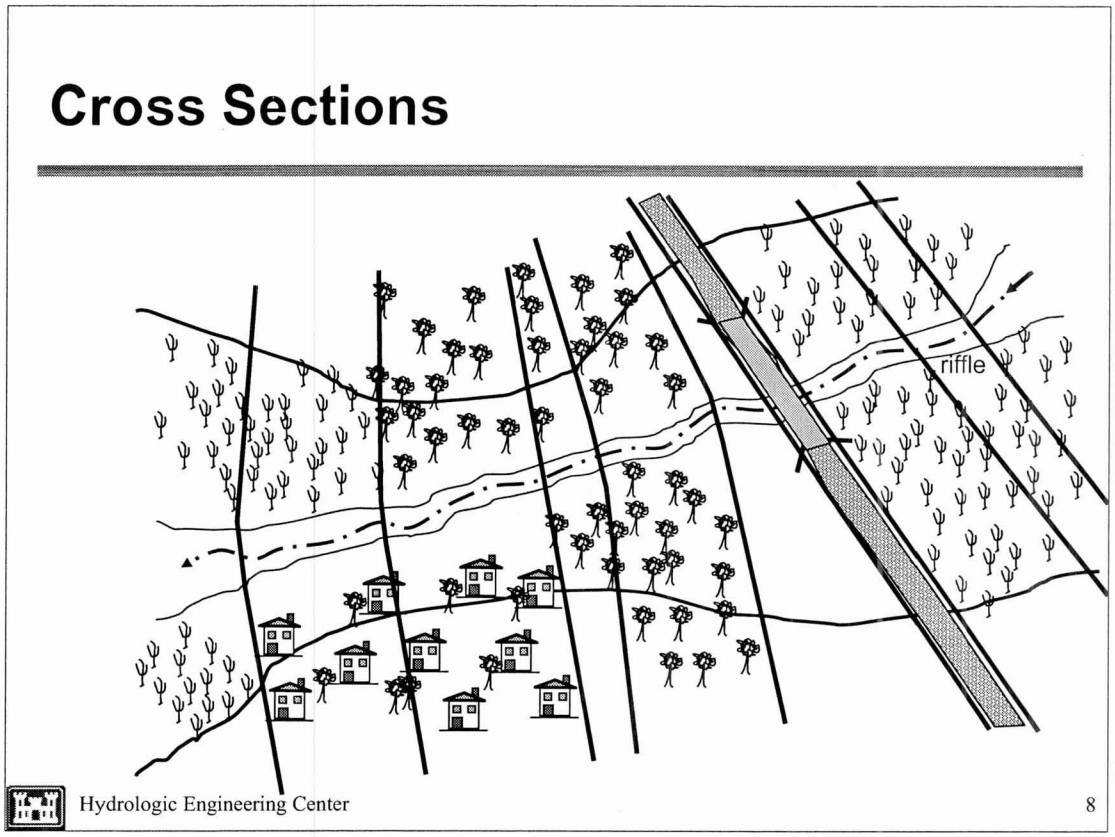
Geometric data consist of establishing the connectivity of the river system (River System Schematic), entering cross-section data, defining all the necessary junction information, adding hydraulic structure data (bridges, culverts, dams, weirs, etc.) and cross section interpolation. The geometric data is entered by selecting Geometric Data from the Edit menu on the HEC-RAS main window.

RAS Geometric Data Editor - Schematic



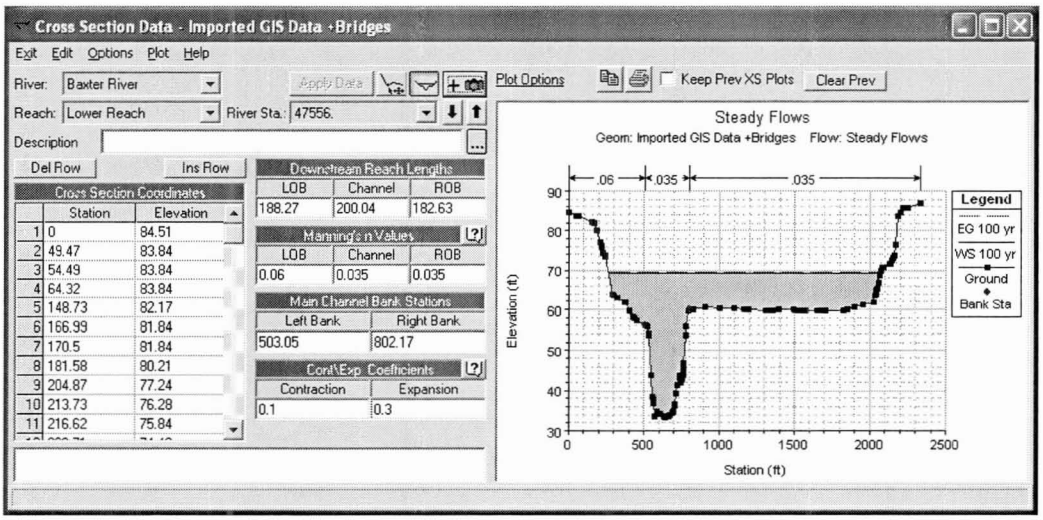
Hydrologic Engineering Center 7

The HEC-RAS Geometry Editor allows the user to enter data from scratch, import data from other models, or import data from a GIS. User's can add Background maps for improved visualization.



After the river system schematic is completed, the next step for the modeler is to enter the cross section data. Cross section data represent the geometric boundary of the stream. Cross sections are located at relatively short intervals along the stream to characterize the flow carrying capacity of the stream and its adjacent floodplain. Cross sections are required at representative locations throughout the stream and at locations where changes occur in discharge, slope, shape, roughness, at locations where levees begin and end, and at hydraulic structures (bridges, culverts, inline weirs/spillways, and lateral weirs/spillways).

Cross Section Editor



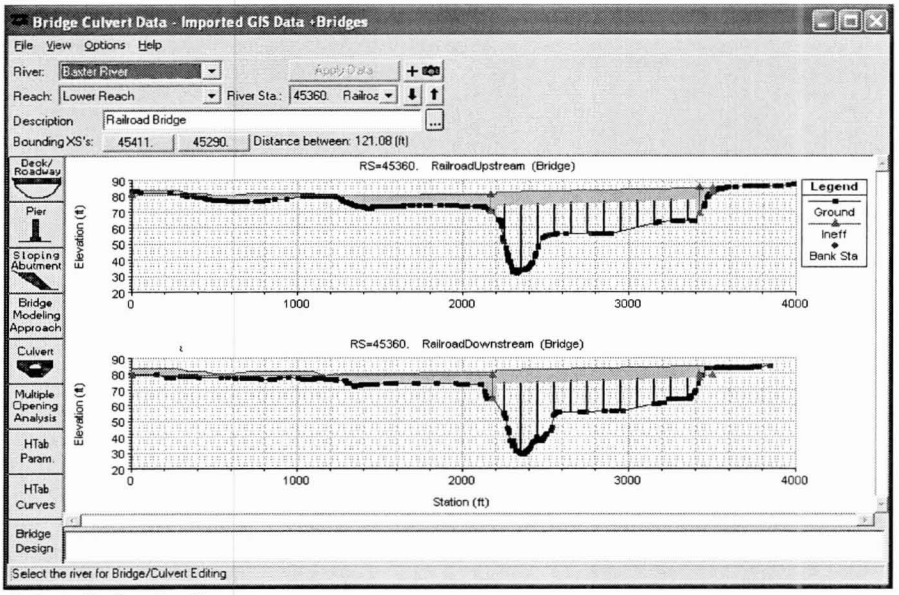
Hydrologic Engineering Center 9

To enter cross section data, the user presses the **Cross Section** button on the Geometric Data window. Once the cross section button is pressed, the Cross Section Data Editor will appear as shown above (Except yours will be blank until you have added some data). To add a cross section to the model, the user must do the following:

From the Cross Section Editor, select the river and the reach that you would like to place the cross section in. This is accomplished by pressing the down arrow on the River and Reach boxes, and then selecting the river and reach of choice.

1. Go to the **Options** menu and select **Add a new Cross Section** from the list. An input box will appear prompting you to enter a river station identifier for the new cross section.
2. Enter all of the required data for the new cross section. Required data is the data that is openly displayed in the cross section editor window.
3. Enter any desired optional information (i.e., ineffective flow areas, levees, blocked obstructions, etc.). Optional cross section information is found under the **Options** menu.
4. Press the **Apply Data** button in order for the interface to accept the data. The apply data button does not save the data to the hard disk, it is used as a mechanism for telling the interface to use the information that was just entered. If you want the data to be saved to the hard disk you must do that from the **File** menu on the geometric data window.

Bridge Editor

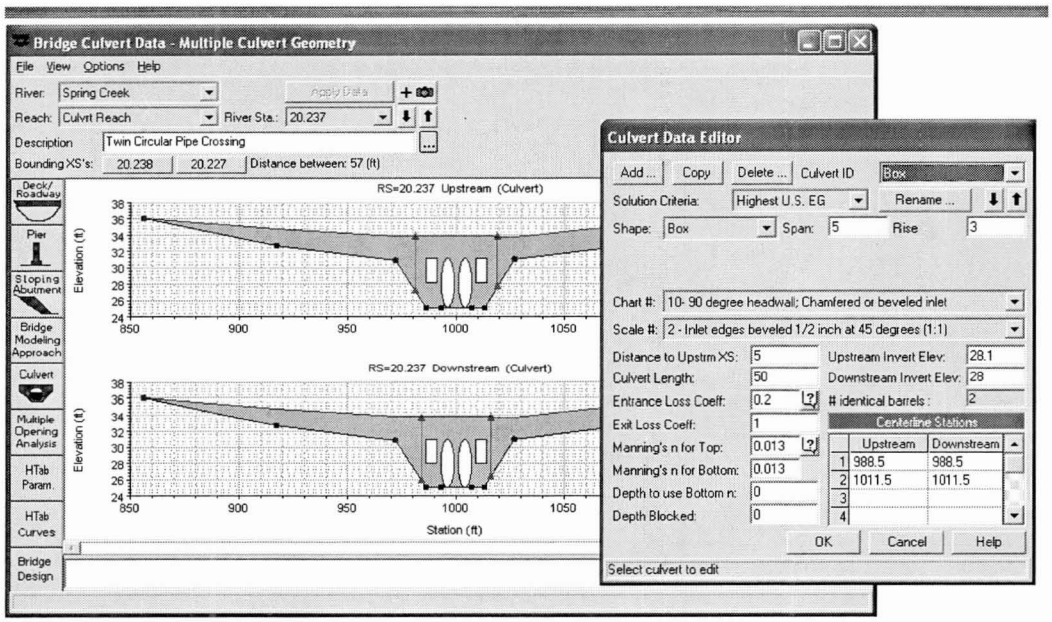


Hydrologic Engineering Center 10

To enter bridge data the user presses the **Bridge/Culvert** button on the geometric data window. Once the bridge/culvert button is pressed, the Bridge/Culvert Data Editor will appear as shown above (your bridge/culvert editor will come up with a blank window until you have entered the bridge data). To add a bridge to the model, do the following:

1. Select the river and reach that you would like to place the bridge in. Selecting a reach is accomplished by pressing the down arrow on the river and reach box, then selecting the river and reach of choice.
2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.
3. Enter all of the required data for the new bridge. This includes:
   - Bridge Deck
   - Sloping Abutments (optional)
   - Piers (optional)
   - Bridge modeling approach information
4. Enter any desired optional information. Optional bridge information is found under the Options menu at the top of the window.
5. Press the Apply Data button for the interface to accept the data.

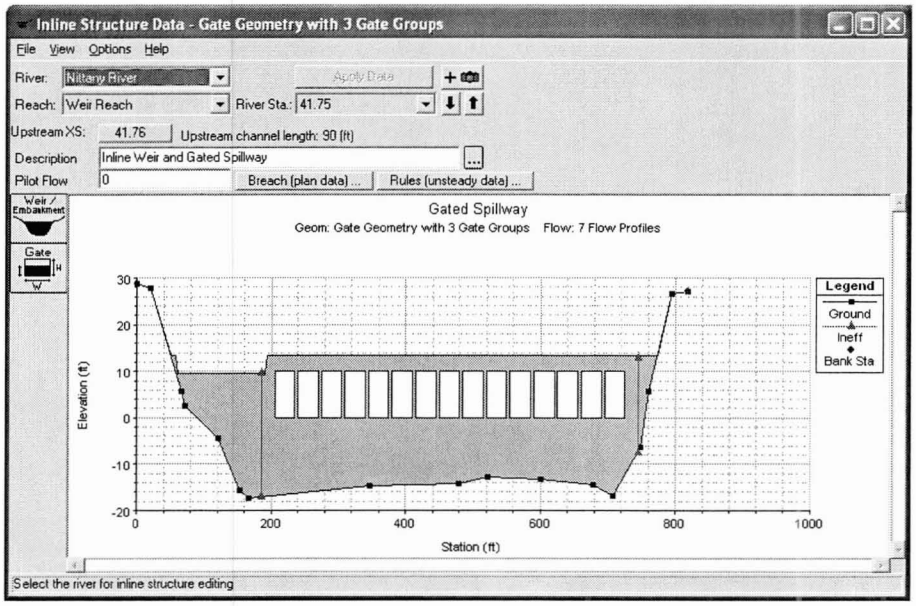
Culvert Editor



Culvert data are entered in the same manner as bridge data. To enter culvert data the user presses the **Bridge/Culvert** button on the Geometric Data window. Once this button is pressed, the Bridge/Culvert Data Editor will appear. To add a culvert group to the model the user must then do the following:

1. Select the river and reach that you would like to place the culvert in. This selection is accomplished by pressing the down arrow on the river and reach boxes and then selecting the river and reach of choice.
2. Go to the **Options** menu of the Bridge/Culvert editor and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new culvert group. After entering the river station, press the OK button and the cross sections that bound the new culvert group will appear in the editor.
3. Enter all of the required data for the culvert group. This includes the road embankment information and the culvert specific data. The roadway information is entered in the same manner as a bridge (using the deck/roadway editor). To enter culvert specific data, press the Culvert button on the Bridge/Culvert Data editor.
4. Once all of the culvert data are entered, press the OK button in order for the interface to accept the information.

Inline Structure Editor



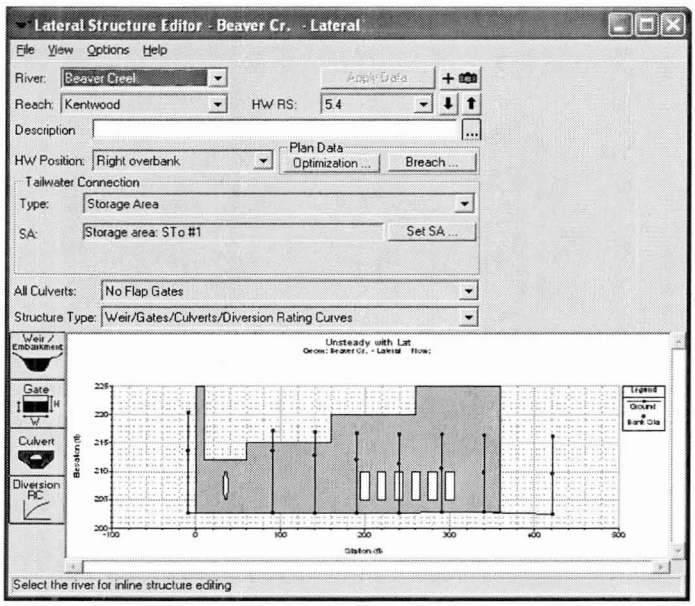
Hydrologic Engineering Center 12

Inline structure data are entered in a similar manner as bridge and culvert data. To enter an inline structure press the Inline Structure button from the Geometric Data window. Once this button is pressed, the Inline Structure Data editor will appear as shown above (except yours will be blank until you enter some data).

To add an inline structure to a model, the user must do the following:

1. Select the river and reach that you would like to place this inline structure into. This is accomplished by first selecting a River, then selecting a specific reach within that river. The River and Reach selection buttons are at the top of the Inline Structure Data editor.
2. Go to the **Options** menu at the top of the window and select **Add an Inline Structure** from the list. An input box will appear asking you to enter a river station identifier for locating this structure within the reach. After entering the river station, press the **OK** button and a copy of the cross section just upstream of this river station will appear on the screen. This cross section is used in formulating the inline structure crossing.
3. Enter all of the data for the Inline structure. This data will include a Weir/Embankment profile, and any gated spillways that you may be modeling. Gated spillways are optional. If the user does not enter any gated spillways, then the program assumes that there is only an inline weir.
4. Once all of the Inline Structure data are entered, press the **Apply Data** button in order for the interface to accept the data. The editor can then be closed by selecting **Exit** from the **File** menu at the top of the window.

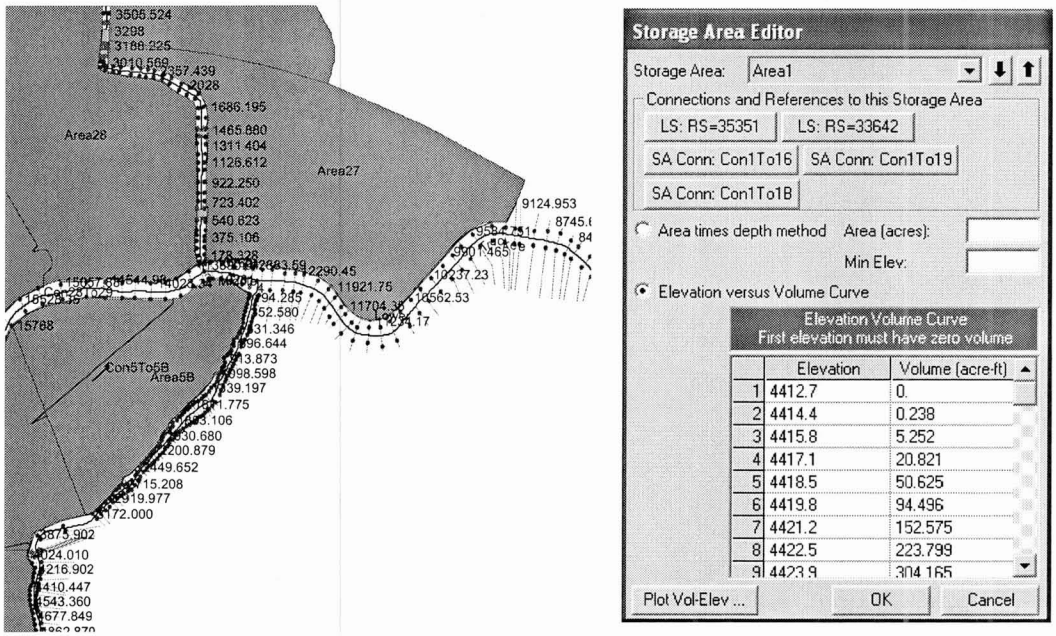
Lateral Structure Editor



Hydrologic Engineering Center 13

Lateral weir, gated spillway, and culvert data are entered in a similar manner as bridge and culvert data. To enter a lateral structure, press the **Lateral Structure** button from the Geometric Data window. Once this button is pressed, the Lateral Structure Data editor will appear as shown above (except yours will be blank until you have entered some data).

Storage Area Editor



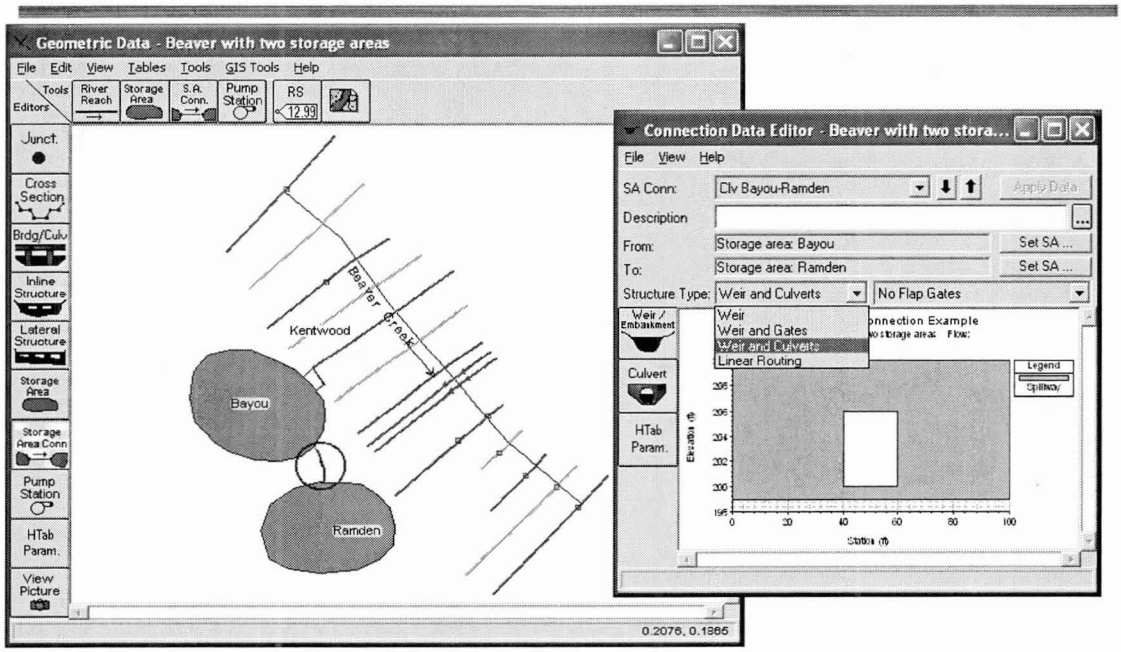
Hydrologic Engineering Center 14

Storage Areas are lake like regions in which water can be diverted into or from. Storage areas can be located at the beginning of a reach, at the end of a reach, or they can be located laterally to a reach. Storage areas can be connected to a river reach by using a lateral structure connection. Storage areas can be connected to another storage area by using a storage area connection. Storage area connections can consist of a weir and gated spillways; a weir and culverts; just a weir; or a linear routing option. An example river system with storage areas is shown in above.

To add a storage area to the river system, first use the storage area drawing tool at the top of the geometric editor. Once the storage area, drawing tool is selected, the user single clicks the left mouse button to begin drawing the storage area. Additional points are added by moving the mouse and single clicking. The storage area will be represented as a polygon. To finish drawing the storage area, double click the left mouse button. The first and last point will then be connected, and the storage area will be filled in with a blue color. The user will then be prompted to enter a name for the storage area.

After the storage area is drawn and labeled, the user must enter data to describe the storage area. This is accomplished with the storage area editor, which is one of the buttons on the left side of the geometric editor.

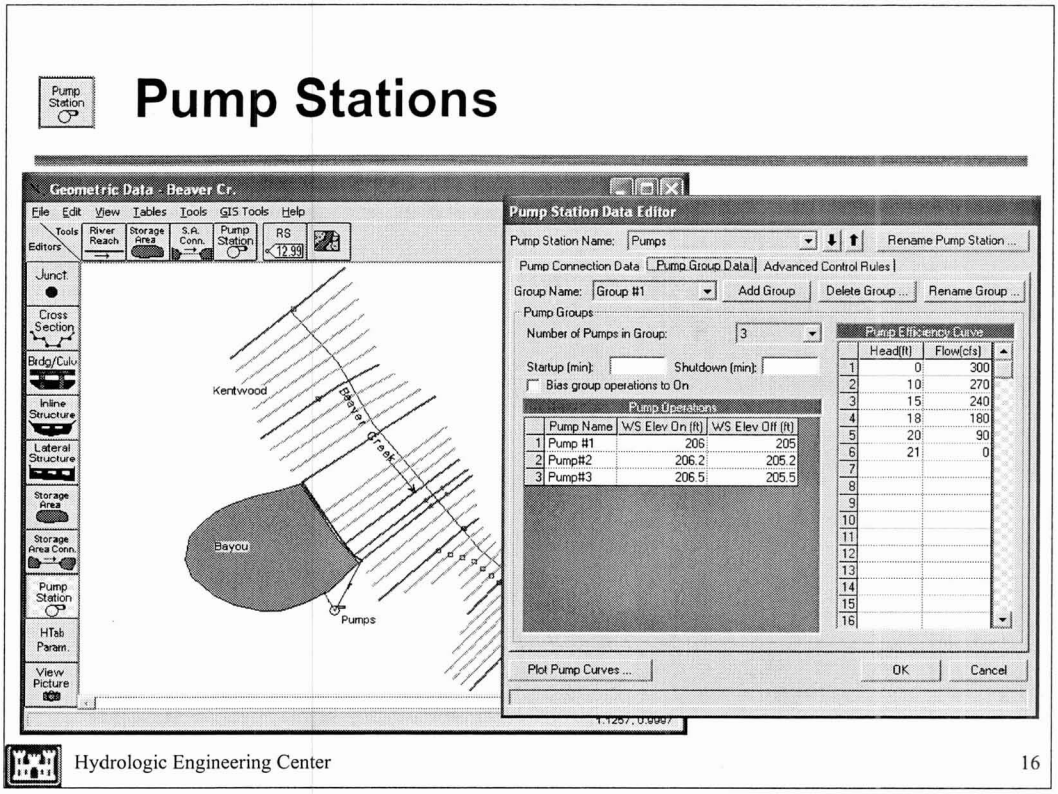
Storage Area Connections



Hydrologic Engineering Center 15

Storage Area connections are used to link two storage areas together with a hydraulic structure. Storage area connections can consist of a weir; culverts and a weir; gated spillways and a weir; or a linear routing option. To establish a hydraulic connection between two storage areas, press the "**Storage Area Conn**" button at the top of the geometric data window. Once the storage area connection drawing tool is invoked, the user simple presses the left mouse button one time while over the from storage area, then double click the left mouse button while over the too storage area. This establishes the connection between the storage areas, as well as what is considered to be the positive flow direction.

Once a connection is made between two storage areas, the user must enter information describing the hydraulics of the connection. This is accomplished by pressing the **Storage Area Conn** editor button on the left hand side of the geometric data editor.



A pump station can be used to pump water between two storage areas, a storage area and a river reach, or between two river reaches. Each pump station can have up to ten different pump groups, and each pump group can have up to twenty identical pumps. To add a pump station to the system, select the **Pump Station** drawing tool at the top of the geometric data editor. When this button is pressed, the mouse icon will change to a set of cross hairs with a pump. Move the mouse to the location that represents where the pump station will be pumping from, press the left mouse button to start the connection. Next, move the mouse to the location where the pump station will be pumping too, then press the left mouse button again. This will establish a pump station connection between two elements.

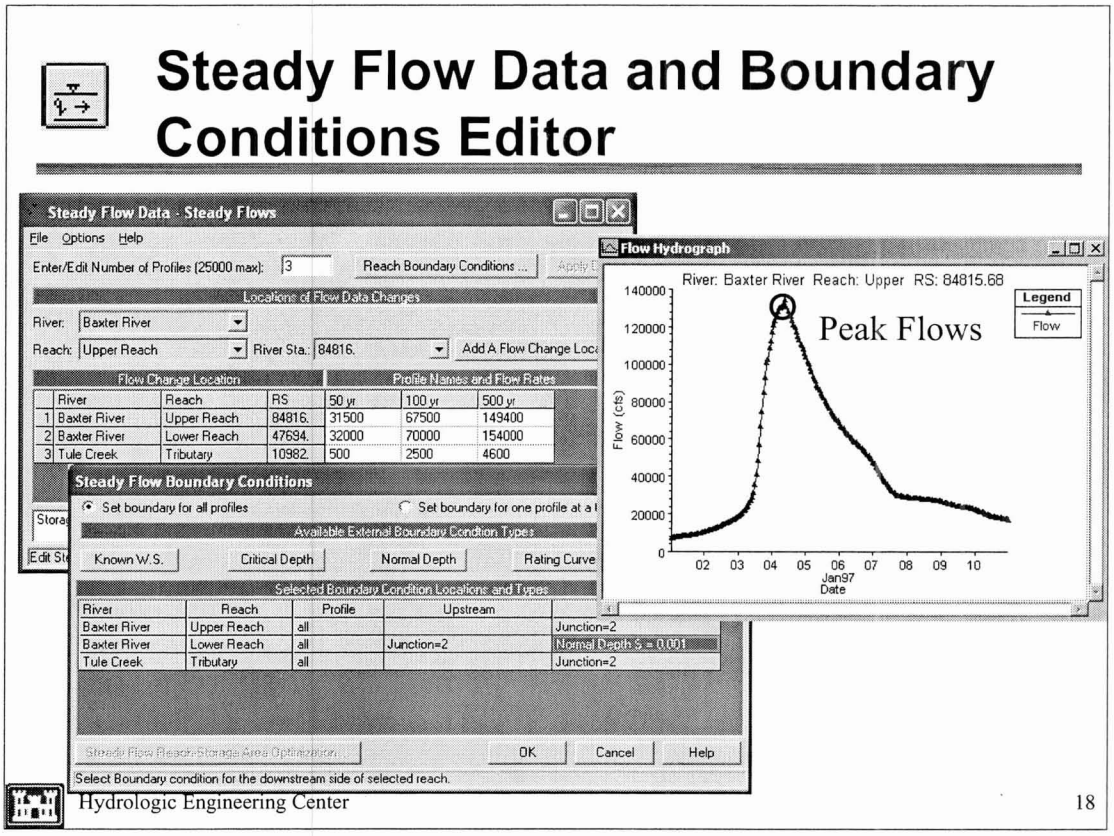
Once a pump station is added to the system, the user must edit the pump station and fill in the required data. To bring up the pump station editor, select the pump station editor button on the left hand side of the geometric data editor, or move the mouse over the pump station icon on the schematic, press down on the left mouse button, then select **Edit Pump Station**.

Flow Data and Boundary Conditions

- Steady flow data - Peak flows or time lines
- Unsteady flow data - Hydrographs
- Boundary Conditions
  - Stage and/or flow hydrographs
  - Rating Curves
  - Normal or critical depth
  - Lateral and uniform lateral inflow hydrographs
  - Groundwater interflow
  - Time series of gate openings
  - Elevation controlled gates
  - Navigation Dam Operations
  - Gates controlled by User Defined Rules

Hydrologic Engineering Center 17

Once the geometric data are entered, the modeler can then enter either steady flow or unsteady flow data. The type of flow data entered depends upon the type of analyses to be performed.

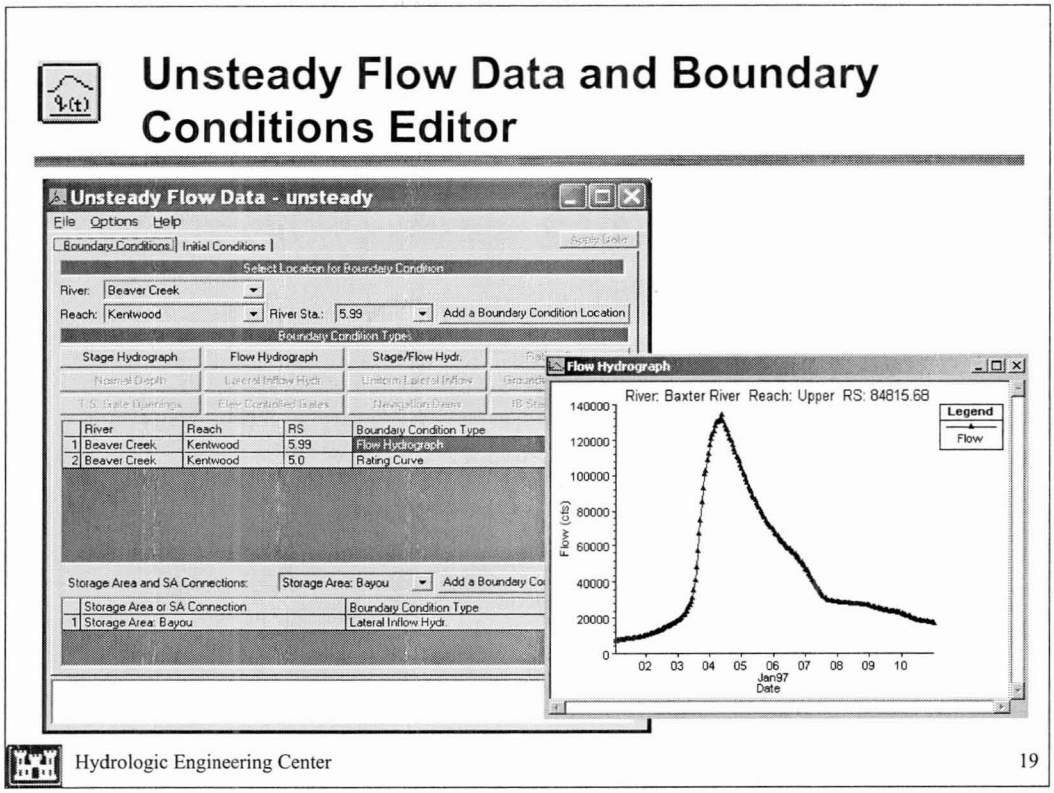


Once all of the geometric data are entered, the modeler can then enter any steady flow data that are required. To bring up the steady flow data editor, select **Steady Flow Data** from the **Edit** menu on the HEC-RAS main window.

The user is required to enter the following information: the number of profiles to be calculated; the peak flow data (at least one flow for every river reach and every profile); and any required boundary conditions. The user should enter the number of profiles first.

After all of the flow data have been entered into the table, the next step is to enter any boundary conditions that may be required. To enter boundary conditions data press the **Boundary Conditions** button at the top right of the steady flow data editor.

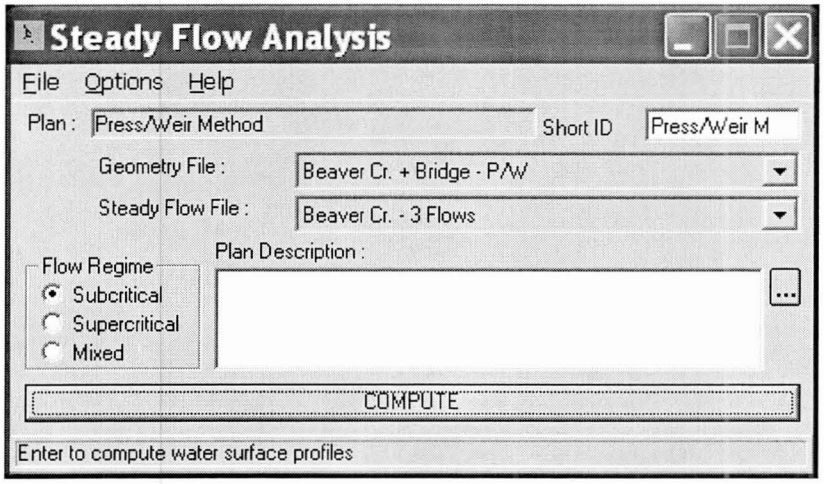
Boundary conditions are necessary to establish the starting water surface at the ends of the river system (upstream and downstream). A starting water surface is necessary in order for the program to begin the calculations. In a subcritical flow regime, boundary conditions are only necessary at the downstream ends of the river system. If a supercritical flow regime is going to be calculated, boundary conditions are only necessary at the upstream ends of the river system. If a mixed flow regime calculation is going to be made, then boundary conditions must be entered at all ends of the river system.



To bring up the unsteady flow data editor, select **Unsteady Flow Data** from the **Edit** menu on the HEC-RAS main window. The user is required to enter boundary conditions at all of the external boundaries of the system, as well as any desired internal locations, and set the initial flow and storage area conditions at the beginning of the simulation.

Boundary conditions are entered by first selecting the **Boundary Conditions** tab from the Unsteady Flow Data editor. River, Reach, and River Station locations of the external bounds of the system will automatically be entered into the table. Boundary conditions are entered by first selecting a cell in the table for a particular location, then selecting the boundary condition type that is desired at that location. Not all boundary condition types are available for use at all locations. The program will automatically gray-out the boundary condition types that are not relevant when the user highlights a particular location in the table. Users can also add locations for entering internal boundary conditions. To add an additional boundary condition location, select the desired River, Reach, and River Station, then press the **Add a Boundary Condition Location** button.

Steady Flow Hydraulic Computations



Hydrologic Engineering Center 20

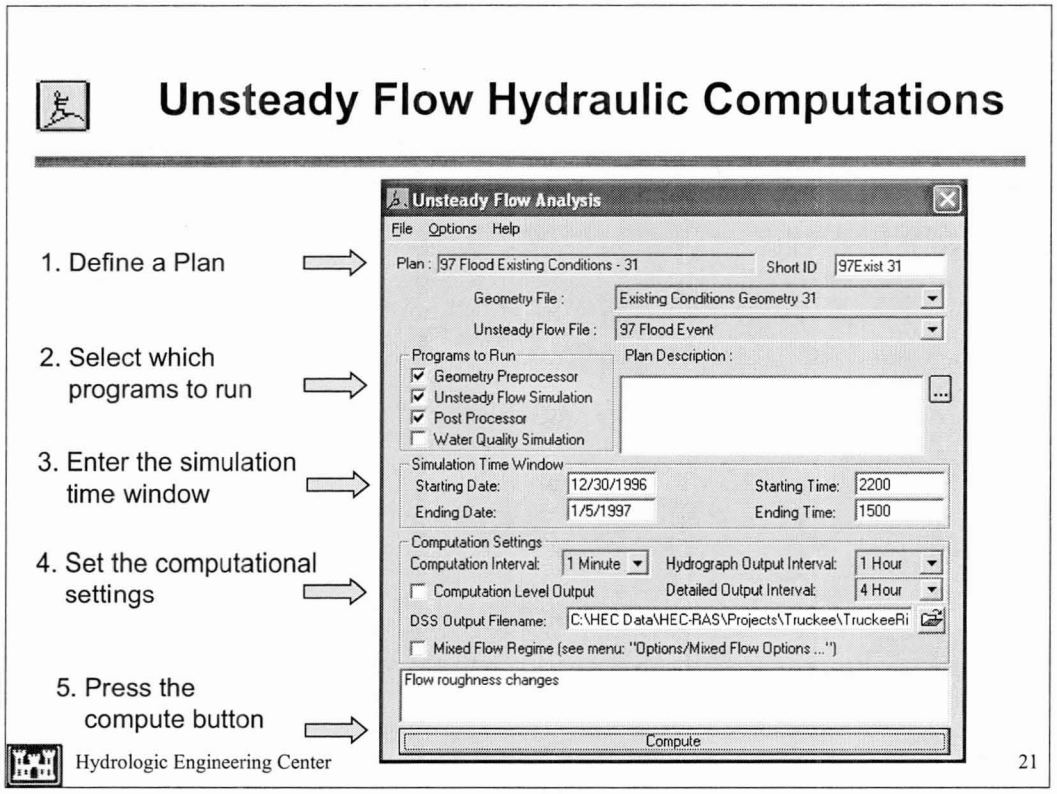
Once all of the geometry and steady flow data have been entered, the user can begin calculating the steady flow water surface profiles. To perform the simulations, go to the HEC-RAS main window and select Steady Flow Analysis from the Run menu. The Steady Flow Analysis window will appear as shown above.

The first step in performing a simulation is to put together a Plan. The Plan defines which geometry and flow data are to be used, as well as provide a description and short identifier for the run. Also included in the plan information are the selected flow regime and the simulation options.

Before a Plan is defined, the user should select which geometry and flow data will be used in the plan. To select a geometry or flow file, press the down arrow button next to the desired data type. When this button is pressed, a list will appear displaying all of the available files of that type that are currently available for the project. Select the geometry and flow files that you want to use for the current plan.

To establish a Plan, select **New Plan** from the **File** menu on the steady flow analysis window. When **New Plan** is selected, a window will appear prompting you to enter a title for the plan. After you enter the title, press the **OK** button to close the window and accept the title. The user will also be prompted to enter a short identifier for the plan. The short identifier is limited to 16 characters. It is very important to enter a short identifier that is descriptive of the plan. When viewing multiple plan output from the graphics and tables, the Short ID will be used to identify each plan.

The last step is to select the desired flow regime for which the model will perform calculations. The user can select between subcritical, supercritical, or mixed flow regime calculations.



Once all of the geometry and unsteady flow data have been entered, the user can begin performing the unsteady flow calculations. To run the simulation, go to the HEC-RAS main window and select **Unsteady Flow Analysis** from the **Run** menu. The Unsteady Flow Analysis window will appear as shown above.

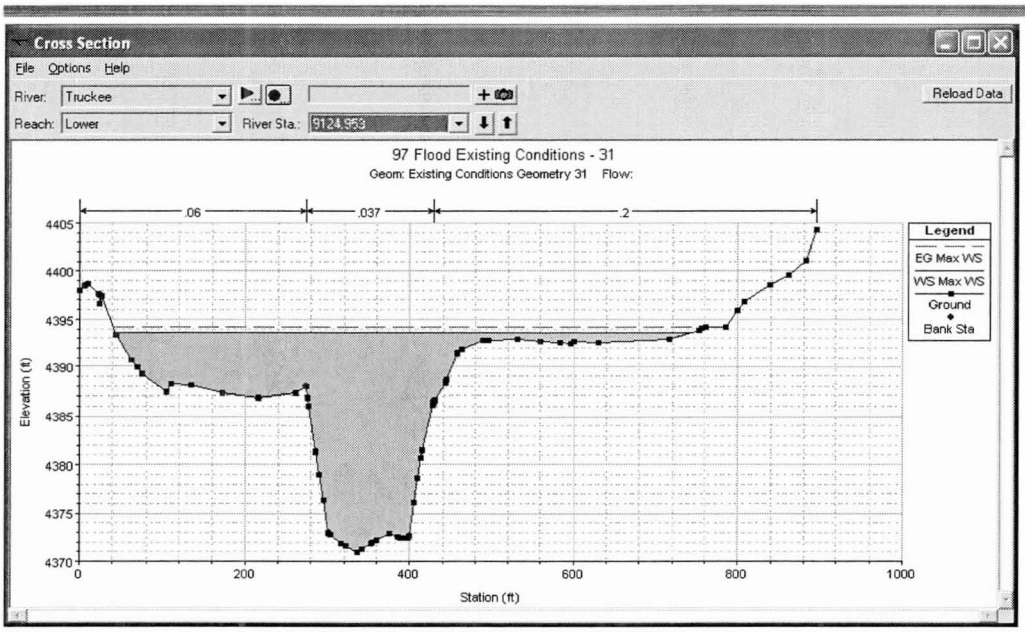
Viewing Results

- **Graphics**
  - **Cross sections**
  - **Water surface profiles**
  - **Generic plots – Any variable in profile**
  - **Rating Curves**
  - **XYZ Plot**
  - **Stage and flow hydrographs**
  - **Animation – (cross section, profile, 3D plots)**
- **Tabular Output**
  - **Pre-defined detailed tables**
  - **Pre-defined summary tables**
  - **User-define output tables**

Hydrologic Engineering Center 22

After the model has finished the steady or unsteady flow computations the user can begin to view the output. Output is available in a graphical and tabular format. The current version of the program allows the user to view cross sections, water surface profiles, general profiles, rating curves, hydrographs, X-Y-Z perspective plots, detailed tabular output at a single location, and summary tabular output at many cross sections. Users also have the ability to develop their own output tables.

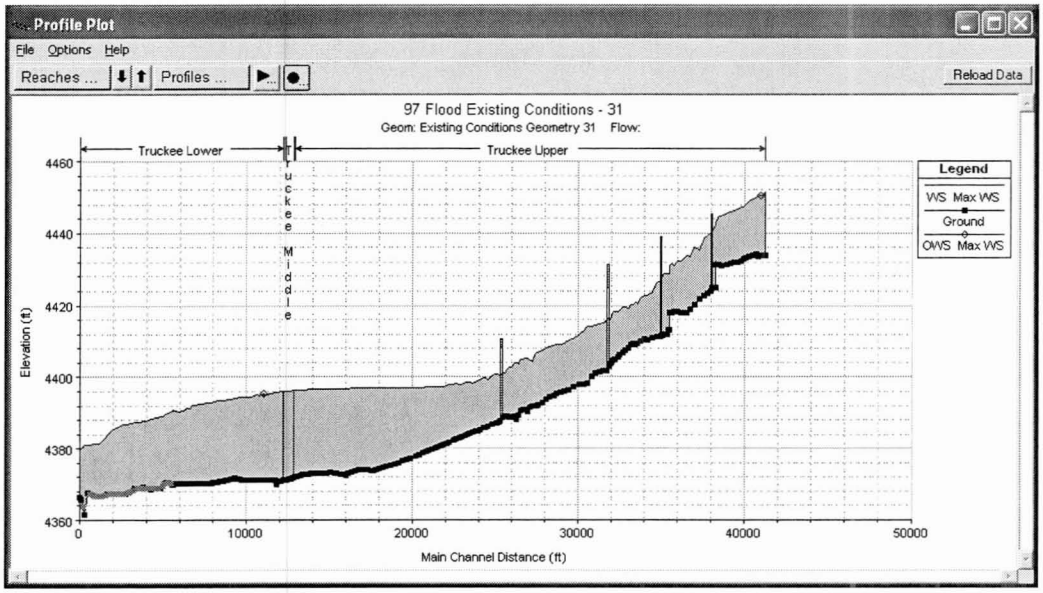
Cross Section Plot



Hydrologic Engineering Center 23

To view a graphic on the screen, select **Cross Sections, Water Surface Profiles, Rating Curves,** or any other graphic option from the **View** menu on the HEC-RAS main window. Once you have selected one of these options, a window will appear with the graphic plotted in the viewing area. An example cross-section plot is shown above. The user can plot any cross section by simply selecting the appropriate reach and river station from the list boxes at the top of the plot. The user can also step through the cross section plots by using the up and down arrows.

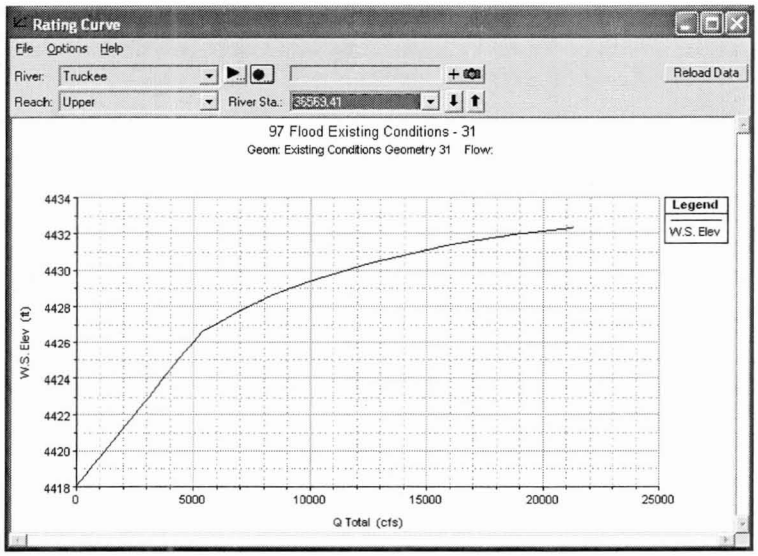
Profile Plot



Hydrologic Engineering Center 24

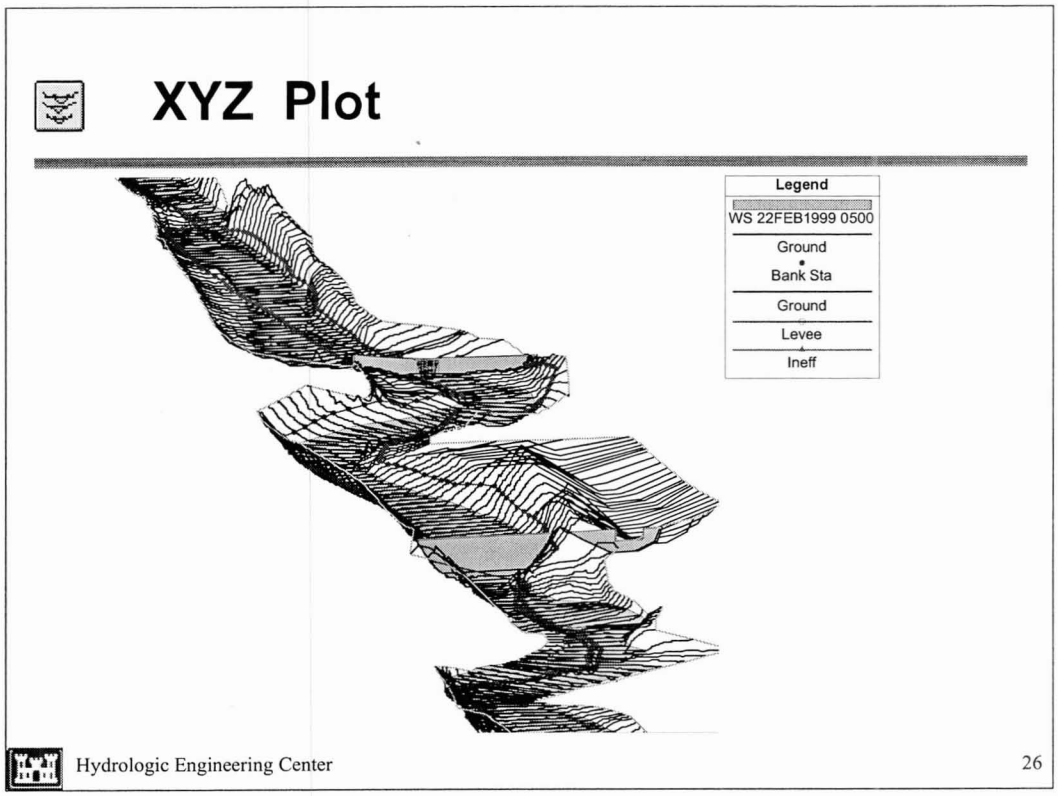
An example profile plot is shown above. The profile plot displays the water surface profile for the first reach in the river system. If there is more than one reach, additional reaches can be selected from the Options menu on or the reach button at the top of the window.

Rating Curves



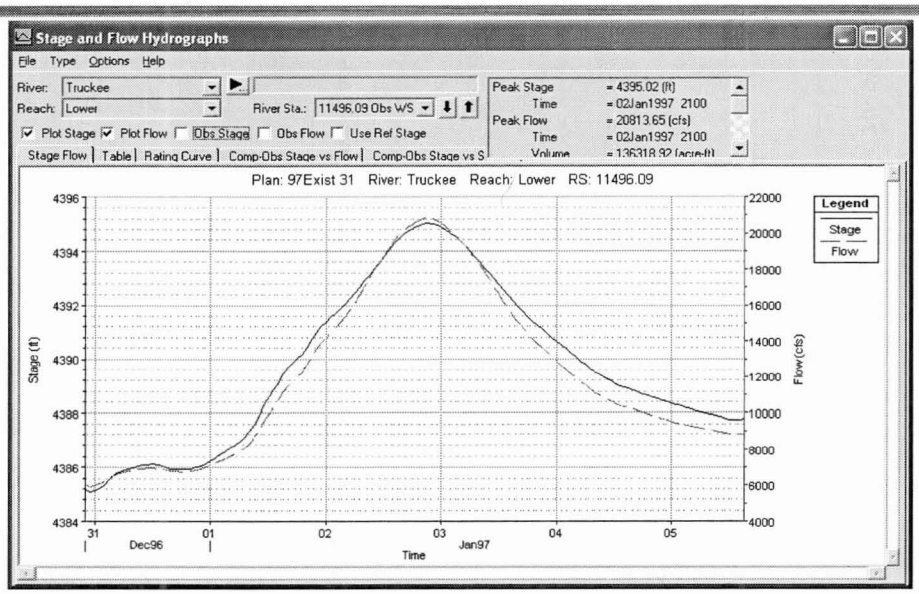
Hydrologic Engineering Center 25

An example rating curve plot is shown in above. The rating curve is a plot of the water surface elevation versus flow rate for the profiles that were computed. A rating curve can be plotted at any location by selecting the appropriate reach and river station from the list boxes at the top of the plot.



Another type of graphic available to the user is the X-Y-Z Perspective Plot. The X-Y-Z plot is a 3-dimensional plot of multiple cross sections within a reach. An example X-Y-Z Perspective plot is shown in above.

Stage and Flow Hydrographs



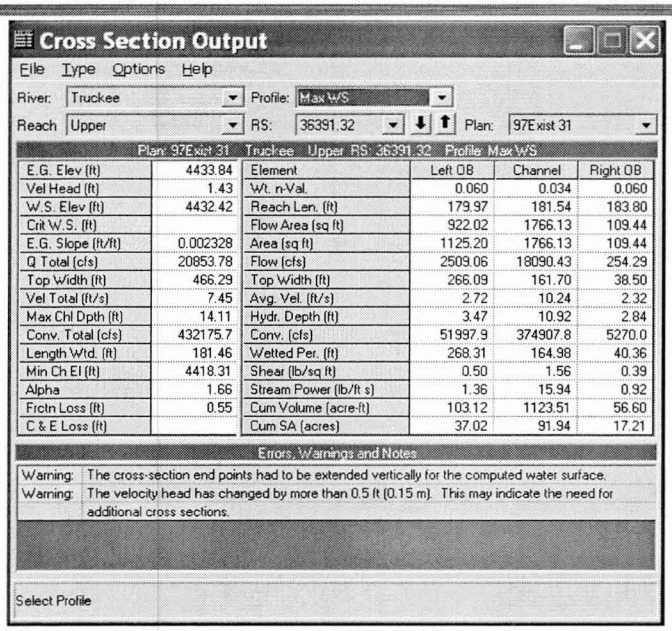
Hydrologic Engineering Center 27

If the user has performed an unsteady flow analysis, then stage and flow hydrographs will be available for viewing. To view a stage and/or flow hydrograph, the user selects **Stage and Flow Hydrographs** from the **View** menu of the main HEC-RAS window. When this option is selected a plot will appear as shown above. The user has the option to plot just the stage hydrograph, just the flow hydrograph, or both as shown in the figure. Additionally, there are three tabs on the plot. The tabs are for plotting (**Plot**), viewing the data in tabular form (**Table**), and plotting a rating curve of the event (**Rating Curve**). By default the window comes up in a plotting mode.

The stage and flow hydrograph plot also has a menu option to select the specific node types to be viewed. By default the plot comes up with a node type of cross section selected. This allows the user to view hydrographs at cross sections only. Other available node types include: Bridges/Culverts; Inline Structures; Lateral Structures; Storage Areas; Storage Area Connections; and Pump Stations.

Additional output for the hydrograph plot includes statistics about the hydrographs (peak stage and flow, time of peak, and volume). Also, the user can simultaneously plot observed hydrograph data at locations where they have gaged information stored in a DSS file. The user attaches gaged hydrograph information to cross section locations from the Unsteady Flow Data editor.

Detailed Tabular Output



		Element	Left OB	Channel	Right OB
E.G. Elev (ft)	4433.84				
Vel Head (ft)	1.43	Wt. n-Val.	0.060	0.034	0.060
W.S. Elev (ft)	4432.42	Reach Len. (ft)	179.97	181.54	183.80
Crit W.S. (ft)		Flow Area (sq ft)	922.02	1766.13	109.44
E.G. Slope (ft/ft)	0.002328	Area (sq ft)	1125.20	1766.13	109.44
Q Total (cfs)	20853.78	Flow (cfs)	2509.06	18090.43	254.29
Top Width (ft)	466.29	Top Width (ft)	266.09	161.70	38.50
Vel Total (ft/s)	7.45	Avg. Vel. (ft/s)	2.72	10.24	2.32
Max Chl Dpth (ft)	14.11	Hydr. Depth (ft)	3.47	10.92	2.84
Conv. Total (cfs)	432175.7	Conv. (cfs)	51997.9	374907.8	5270.0
Length Wtd. (ft)	181.46	Wetted Per. (ft)	268.31	164.98	40.36
Min Ch El (ft)	4418.31	Shear (lb/sq ft)	0.50	1.56	0.39
Alpha	1.66	Stream Power (lb/ft s)	1.36	15.94	0.92
Frctn Loss (ft)	0.55	Cum Volume (acre-ft)	103.12	1123.51	56.60
C & E Loss (ft)		Cum SA (acres)	37.02	91.94	17.21

Hydrologic Engineering Center 28

Detailed output tables show hydraulic information at a single location, for a single profile. To display a detailed output table on the screen, select **Detailed Output Tables** from the **View** menu of the main HEC-RAS window. An example detailed output table is shown above.

By default, this table comes up displaying detailed output for cross sections. Any cross section can be displayed in the table by selecting the appropriate river, reach and river station from the list boxes at the top of the table. Also, any of the computed profiles can be displayed by selecting the desired profile from the profile list box. Additionally, different plans can be viewed by selecting a plan from the plan list box.

Users can also view detailed hydraulic information for other types of nodes. Other table types are selected from the **Type** menu on the detailed output table window.

Summary Tabular Output

Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

HEC-RAS Plan: Unsteady River: Beaver Creek Reach: Kentwood Profile: Max WS Reload Data

Reach	River Sta	Profile	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area	Top Width	Froude # Chl
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)	(ft)	
Kentwood	5.99	Max WS	13996.17	209.90	219.85		220.18	0.002570	7.42	6331.47	1855.91	0.46
Kentwood	5.9325*	Max WS	13992.26	208.67	219.28		219.63	0.002459	7.03	6362.24	1819.58	0.45
Kentwood	5.875*	Max WS	13986.51	207.45	218.79		219.10	0.002096	6.37	6703.48	1784.99	0.42
Kentwood	5.8175*	Max WS	13983.76	206.23	218.40		218.64	0.001567	5.50	7421.29	1755.34	0.36
Kentwood	5.76	Max WS	13981.22	205.00	218.14		218.30	0.001058	4.58	8524.62	1732.38	0.30
Kentwood	5.71*	Max WS	13979.01	204.80	217.97		218.11	0.000883	4.57	8596.28	1800.36	0.27
Kentwood	5.66*	Max WS	13978.72	204.60	217.84		217.95	0.000790	4.65	8680.50	1868.11	0.26
Kentwood	5.61	Max WS	13977.01	204.40	217.73		217.79	0.000681	4.39	8786.53	1932.07	0.23
Kentwood	5.55333*	Max WS	13976.91	204.23	217.63		217.70	0.000591	4.30	8780.18	1952.55	0.23
Kentwood	5.49666*	Max WS	13976.74	204.07	217.53		217.62	0.000549	4.26	8733.22	1907.18	0.22
Kentwood	5.44	Max WS	13976.49	203.90	217.44		217.54	0.000494	4.14	8615.68	1844.11	0.22
Kentwood	5.41	Max WS	13975.87	202.70	216.79	212.23	217.40	0.001534	6.57	7795.38	1841.45	0.37
Kentwood	5.4 Highway 149		Bridge									
Kentwood	5.39	Max WS	13970.40	202.70	215.53		216.33	0.002445	7.56	6481.47	1695.13	0.46
Kentwood	5.34*	Max WS	13970.08	202.75	215.41		215.56	0.001007	4.67	6329.81	1665.20	0.29
Kentwood	5.29	Max WS	13968.39	202.80	215.20		215.31	0.000960	4.11	5797.61	1619.80	0.27
Kentwood	5.23666*	Max WS	13966.93	201.83	214.90		215.03	0.001191	5.12	6093.02	1652.62	0.31
Kentwood	5.18333*	Max WS	13965.64	200.87	214.46		214.69	0.001706	6.65	5800.47	1675.41	0.37
Kentwood	5.13	Max WS	13963.72	199.90	213.83		214.14	0.001883	7.41	5809.88	1683.71	0.40
Kentwood	5.08666*	Max WS	13963.59	199.90	213.40		213.71	0.001923	7.02	6110.63	1781.07	0.40
Kentwood	5.04333*	Max WS	13962.91	199.90	213.02		213.28	0.001781	6.30	6669.60	1883.69	0.38
Kentwood	5.0	Max WS	13962.81	199.90	212.67	210.51	212.81	0.001203	4.88	8236.80	1973.56	0.31

Total flow in cross section.

Hydrologic Engineering Center 29

Profile summary tables are used to show a limited number of hydraulic variables for several cross sections. To display a profile summary table on the screen, select **Profile Summary Table** from the **View** menu of the main HEC-RAS window. An example profile summary table is shown above.

here are several standard table (Std. Tables) types available to the user. Some of the tables are designed to provide specific information at hydraulic structures (e.g., bridges and culverts), while others provide generic information at all cross sections.

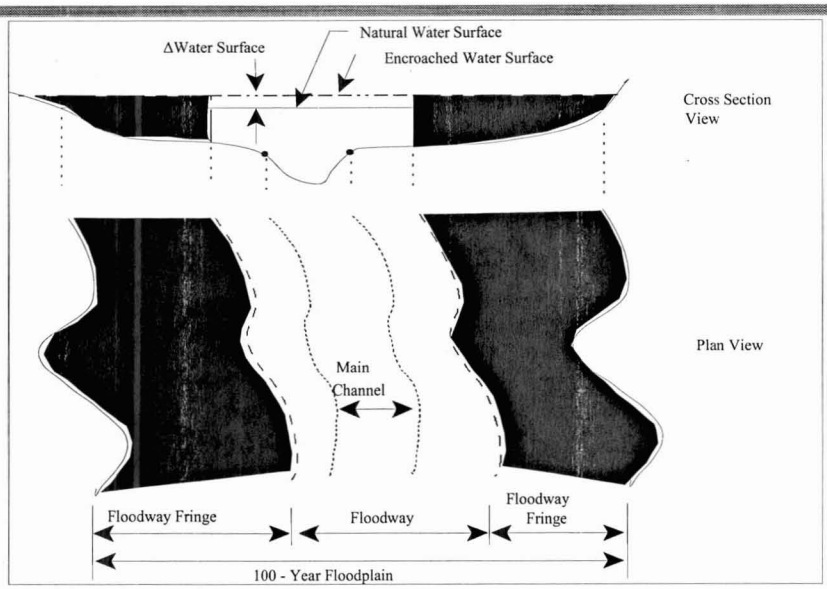
Simulation Options

- FEMA Floodway Analysis
- Channel Design/Modifications
- Bridge Scour Analysis
- Detailed Velocity Distribution
- Mixed Flow Regime in Unsteady Flow
- Dam Break Analysis
- Levee Breaching and Overtopping
- Navigation Dams

Hydrologic Engineering Center 30

Besides computing steady and unsteady flow water surface profiles, HEC-RAS has several options to perform specific types of simulations. Some of the unique simulation options contained in HEC-RAS are listed above.

FEMA Floodway Analyses



Hydrologic Engineering Center 31

The evaluation of the impact of floodplain encroachments on water surface profiles can be of substantial interest to planners, land developers, and engineers. Floodplain and floodway evaluations are the basis for floodplain management programs. Most of the studies are conducted under the National Flood Insurance Program and follow the procedures in the "Flood Insurance Study Guidelines and Specifications for Study Contractors," FEMA 37 (Federal Emergency Management Agency, 11085).

HEC-RAS contains five optional methods for specifying floodplain encroachments. For information on the computational details of each of the five encroachment methods, as well as special considerations for encroachments at bridges, culverts, and multiple openings.

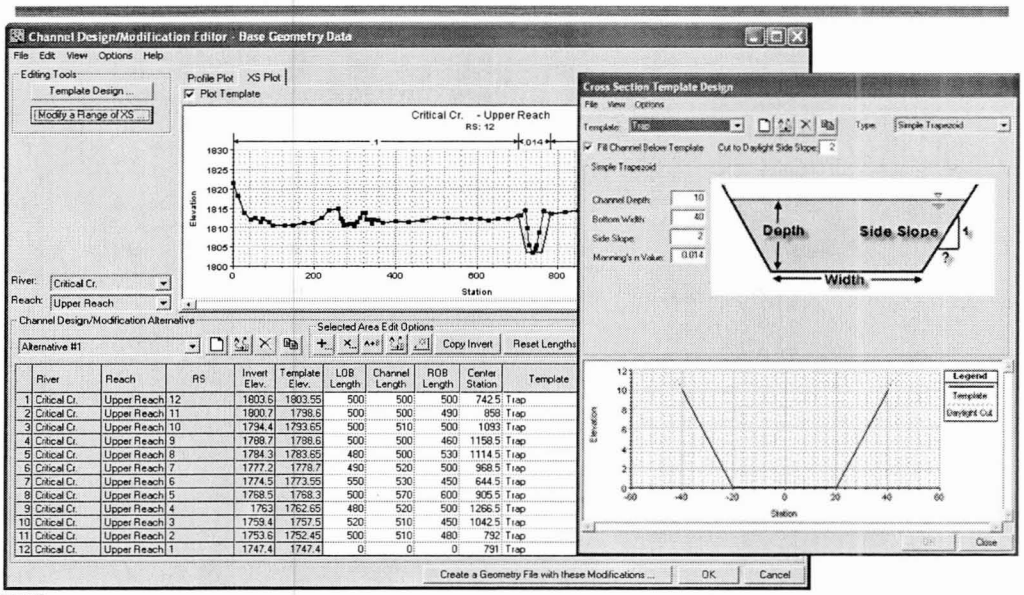
Floodway Definitions:

FLOODPLAIN: The channel of a water course and those portions of the adjoining land area which are required to carry a flood.

FLOODWAY: The channel of a water course and the adjacent land areas that must be reserved in order to discharge the base flood without cumulatively increasing the water-surface elevation more than a designated height. (FEMA 37)

FLOODWAY FRINGE: That portion of the floodplain between the floodway and the natural outline of the floodplain of the base flood.

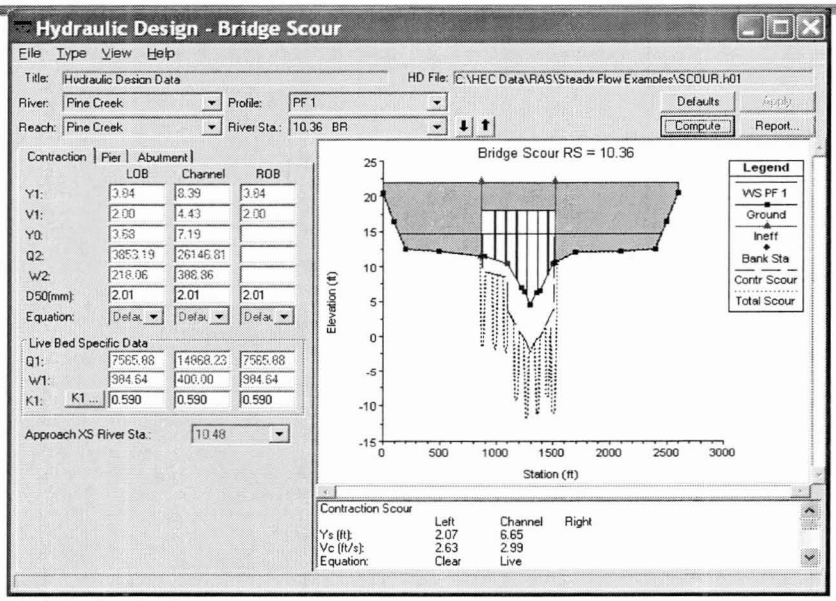
Channel Design/Modification Editor



Hydrologic Engineering Center 32

The channel design/modification tools in HEC-RAS allow the user to perform a series of trapezoidal cuts into the existing channel geometry or to create new channel geometry. The current version of HEC-RAS has two tools for performing channel modifications. These tools are available from the **Tools** menu of the Geometric Data editor and are labeled **Channel Design/Modification** and **Channel Modification (original)**. The tool labeled **Channel Design/Modification** is a new tool for HEC-RAS version 4.0. The tool labeled **Channel Modification (original)** is the original channel modification tool developed for HEC-RAS. The original channel modification tool has been left in HEC-RAS for those user's who may prefer this tool to the new one. Both channel modification tools will be described in this chapter. In general, these tools are used for planning studies, but it can also be used for hydraulic design of flood control channels.

Bridge Scour Analysis

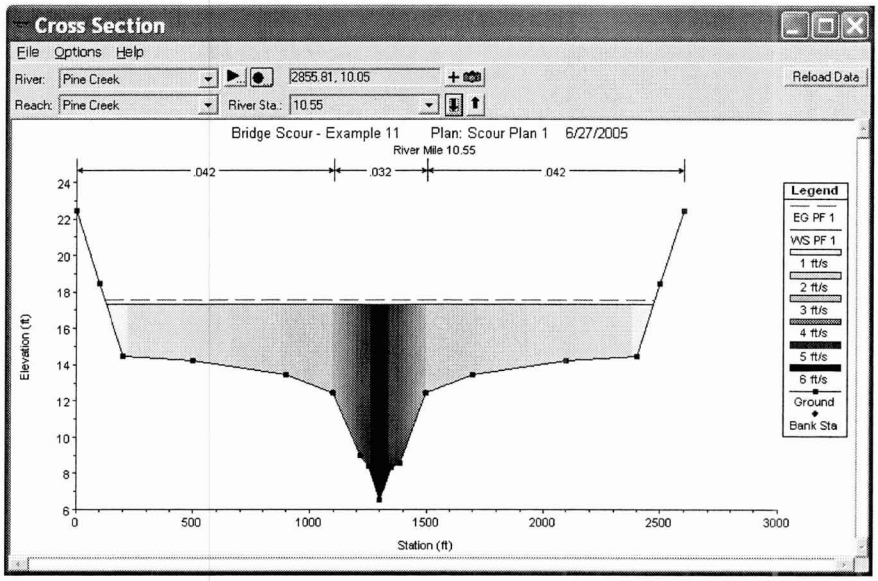


Hydrologic Engineering Center 33

The computation of scour at bridges within HEC-RAS is based upon the methods outlined in Hydraulic Engineering Circular No. 18 (FHWA, 2001). Before performing a scour analysis with the HEC-RAS software, the engineer should thoroughly review the procedures outlined in the Hydraulic Engineering Circular No. 18 (HEC 18) report.

After performing the water surface profile calculations for the design events, and computing the flow distribution output, the bridge scour can then be evaluated. The total scour at a highway crossing is comprised of three components: long-term aggradation and degradation; contraction scour; and local scour at piers and abutments. The scour computations in the HEC-RAS software allow the user to compute contraction scour and local scour at piers and abutments. The current version of the HEC-RAS software does not allow the user to evaluate long-term aggradation and degradation. Long term aggradation and degradation should be evaluated before performing the bridge scour analysis. Procedures for performing this type of analyses are outlined in the HEC No. 18 report.

Velocity Distribution



Hydrologic Engineering Center 34

The user has the option of plotting velocity distribution output from the cross section viewer. Velocity distributions can only be plotted at locations in which the user has specified that flow distribution output be calculated during the computations. To view the velocity distribution plot, first bring up a cross section plot (select "Cross Sections" from the view menu of the main HEC-RAS window). Next, select the cross section in which you would like to see the velocity distribution output. Select **Velocity Distribution** from the **Options** menu of the cross section window. This will bring up a pop up window that will allow you to set the minimum velocity, maximum velocity, and velocity increment for plotting. In general, it is better to let the program use the maximum velocity range for plotting. Next, the user selects **Plot Velocity Distribution**, then press the "OK" button and the velocity distribution plot will appear as shown above.

Mixed Flow Regime for Unsteady Flow

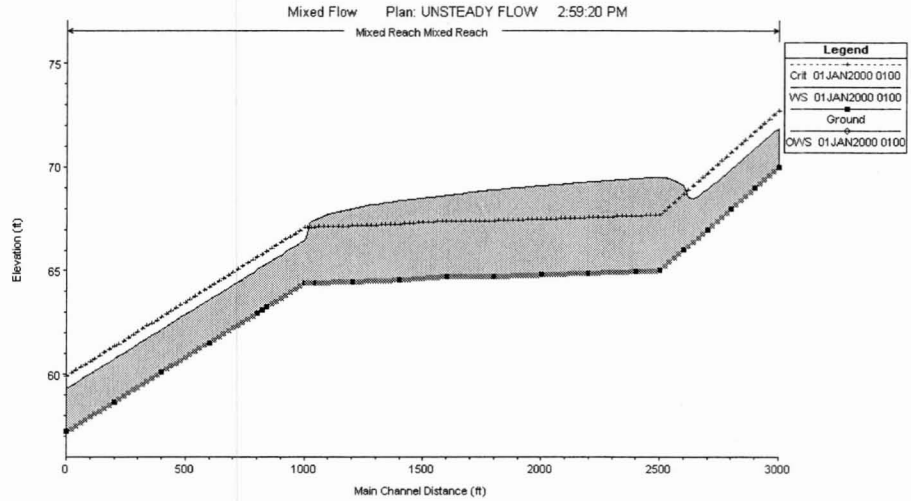
- **Subcritical and Supercritical Flow**
- **Hydraulic Jumps**
- **Draw Downs (sub to super)**

Hydrologic Engineering Center 35

Modeling mixed flow regime (subcritical, supercritical, hydraulic jumps, and draw downs) is quite complex with an unsteady flow model. In general, most unsteady flow solution algorithms become unstable when the flow passes through critical depth. The solution of the unsteady flow equations is accomplished by calculating derivatives (changes in depth and velocity with respect to time and space) in order to solve the equations numerically. When the flow passes through critical depth, the derivatives become very large and begin to cause oscillations in the solution. These oscillations tend to grow larger until the solution goes completely unstable.

In order to solve the stability problem for a mixed flow regime system, Dr. Danny Fread (Fread, 1986) developed a methodology called the "Local Partial Inertia Technique." The LPI method has been adapted to HEC-RAS as an option for solving mixed flow regime problems when using the unsteady flow analysis portion of HEC-RAS. This methodology applies a reduction factor to the two inertia terms in the momentum equation as the Froude number goes towards 1.0.

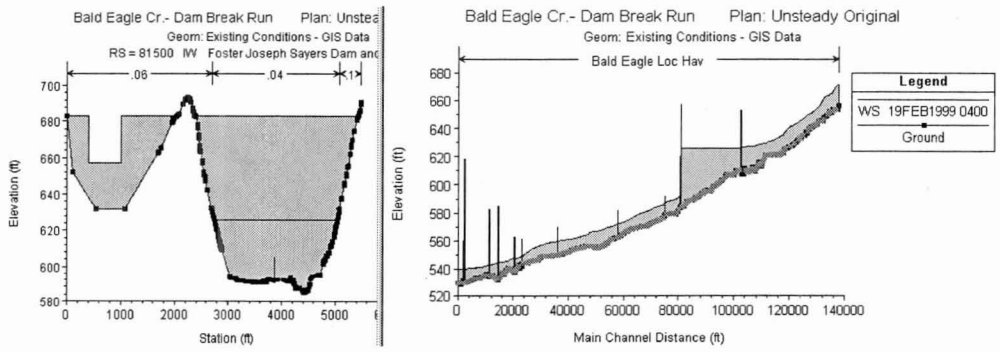
Mixed Flow Regime Example



Hydrologic Engineering Center 36

A profile plot of a mixed flow regime problem is shown above. This example was run with the unsteady flow simulation capability within HEC-RAS using the mixed flow regime option. The example shows a steep reach flowing supercritical, which then transitions into a mild reach. A hydraulic jump occurs on the mild reach. The mild reach then transitions back to a steep reach, such that the flow goes from subcritical to supercritical. Because of a high downstream boundary condition (for example backwater from a lake), the flow then goes from supercritical to subcritical though another hydraulic jump.

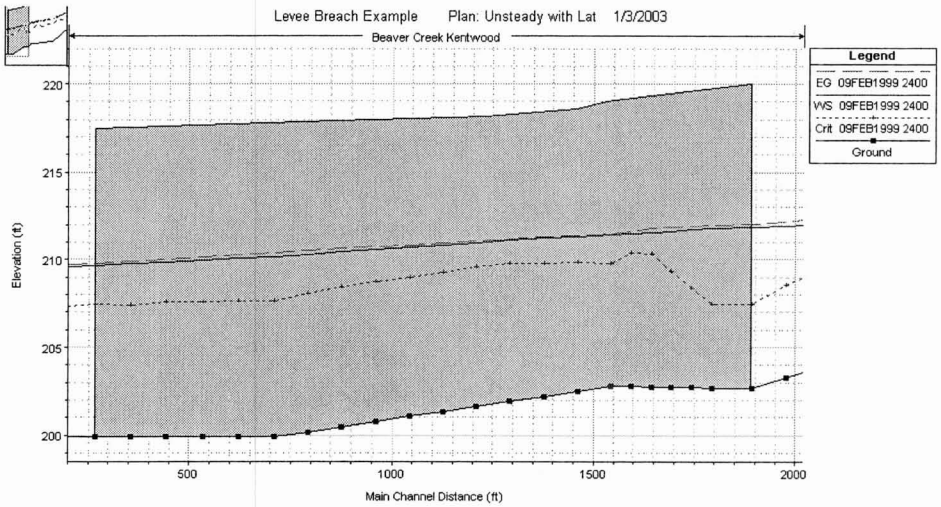
Cross Section and Profile Animation of Breach



Hydrologic Engineering Center 37

HEC-RAS can be used to model both overtopping as well as piping failure breaches for earthen dams. Additionally, the more instantaneous type of failures of concrete dams (generally occurring from earthquakes) can also be modeled. The resulting flood wave is routed downstream using the unsteady flow equations. Inundation mapping of the resulting flood can be done with the HEC-GeoRAS program (companion product to HEC-RAS) when GIS data (terrain data) are available. An example plot of the Dam and the profile for a Dam Break analysis is shown above.

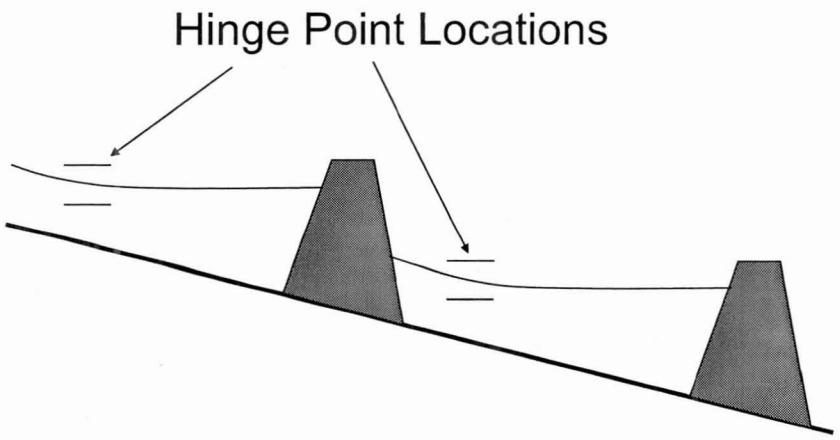
Profile Plot of Levee Breach



Hydrologic Engineering Center 38

Levee overtopping and breaching can be analyzed within HEC-RAS by modeling the levee as a lateral structure. When modeling a levee with a lateral structure, the area behind the levee should not be included in the cross section data of the main river. The cross sections should stop at the bottom of the levee. The lateral structure (levee) can be connected to a storage area or another river reach. The strategy for modeling the area behind the levee will depend upon what will happen to the water if the levee overtops or breaches. If the water going over or through the levee will pond, then a storage area would be more appropriate for modeling the area behind the levee. If the water will continue to flow in the downstream direction, and possibly join back into the main river, then it would be more appropriate to model that area as a separate river reach.

Navigation Dam Operations

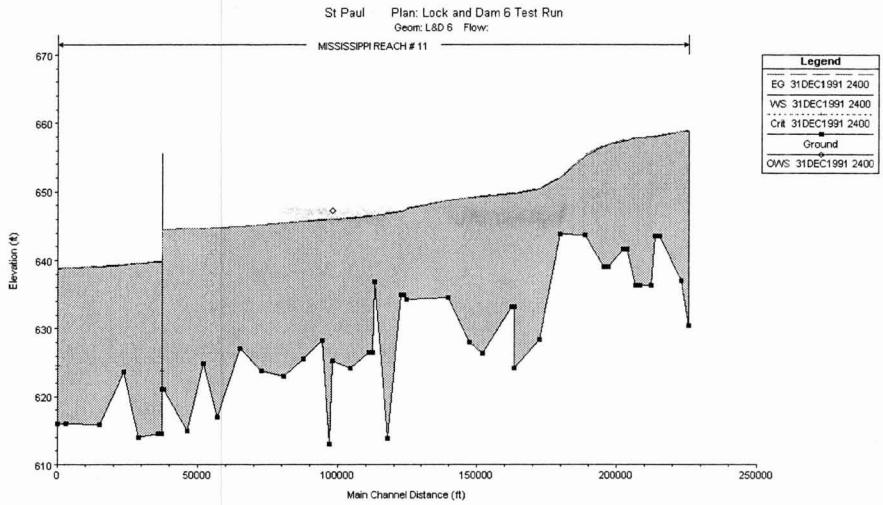


Hydrologic Engineering Center 39

Navigation Dam Operations can be used to control gate operations to maintain an upstream water surface elevation (Hinge point).

For a navigation dam, the program will try to maintain both a minimum and maximum water surface at one or more locations along a navigation channel. The program does this by controlling the gate settings on an inline structure. The user enters a target water surface (and various other calibration data) and the program will adjust the gate settings at user specified time intervals in order to meet the target water surface as closely as possible.

Navigation Dams



Hydrologic Engineering Center 40

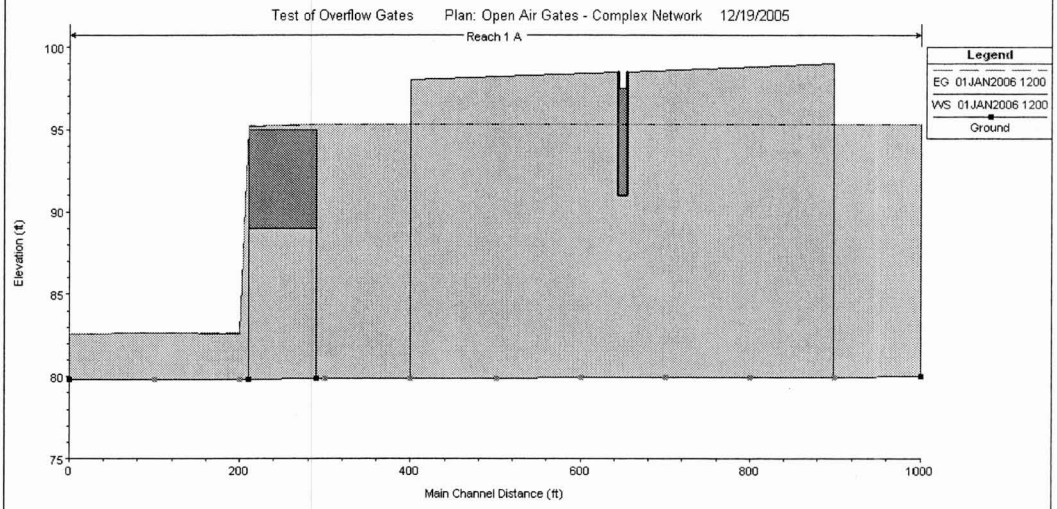
Shown in the example above is lock and dam no. 6 on the Mississippi river.

Recently Added Features

- Overflow Gates
- User Defined Rules for Gate Operations
- Pressure Flow in Pipes
- Pump Station Rules
- Geo-referencing Tools
- Water Quality – Temperature Modeling
- SIAM (Sediment Impact Analysis Module)
- Sediment Transport (erosion and deposition)

Hydrologic Engineering Center 41

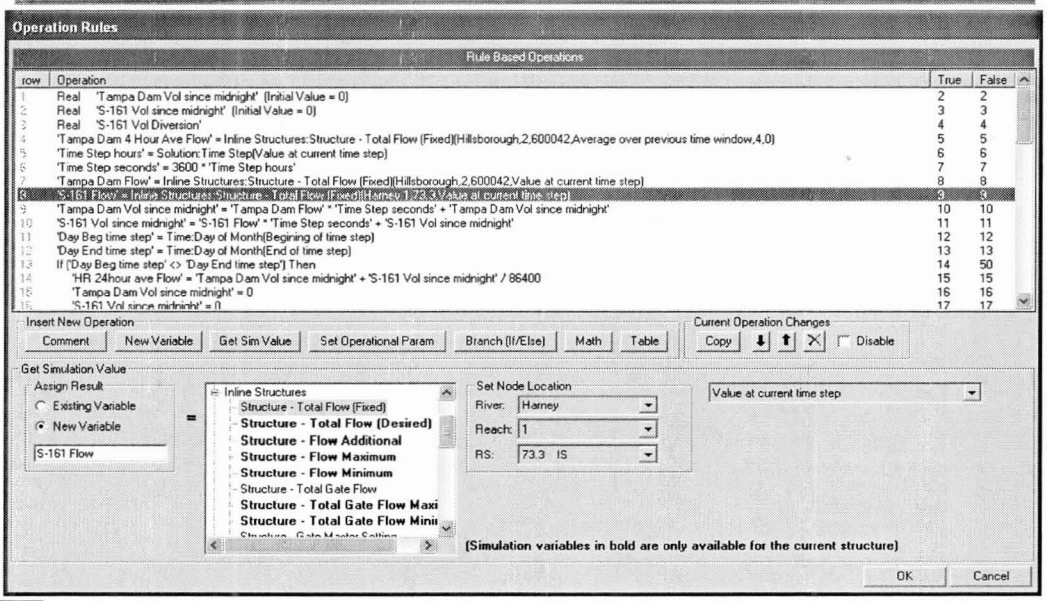
Overflow Gates Example



Hydrologic Engineering Center 42

The example above demonstrates the use of overflow gates for both an inline and a lateral structure.

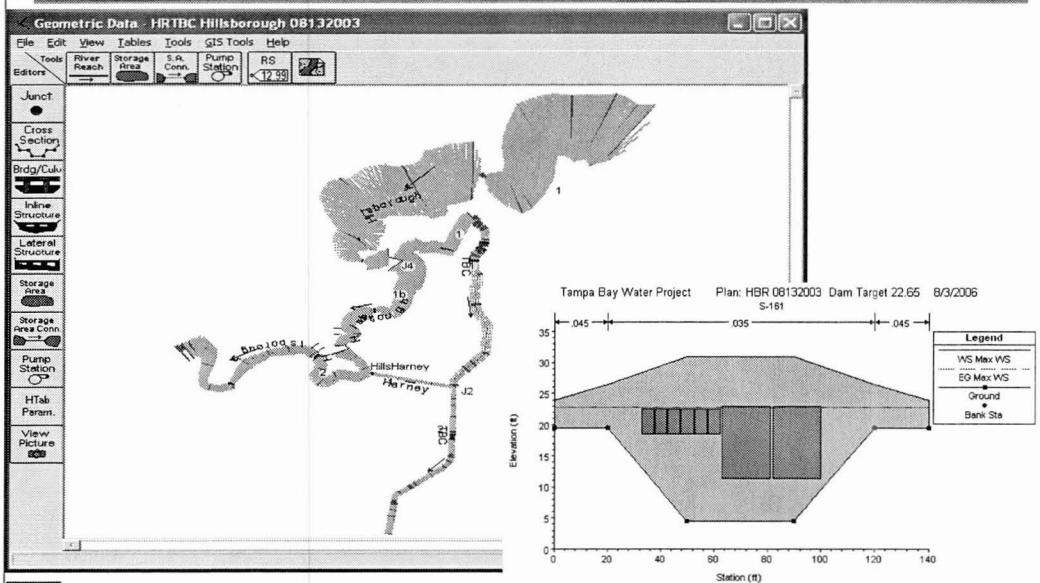
User Defined Rules Editor for Operating Gated Structures



Hydrologic Engineering Center 43

The operating procedures for determining and controlling the releases from reservoirs and other types of hydraulic structures can be quite complex. HEC-RAS allows flexibility in modeling and controlling the operations of gates at hydraulic structures through the use of rules (Figure above). Examples of variables that could be used to control releases from a hydraulic structure are: current flows and water surfaces at the structure, current flows and stages at a downstream or upstream cross section location, time considerations (winter, morning, etc), and/or previously computed values (accumulated outflows, running averages, etc). Rule operations in HEC-RAS are available for inline hydraulic structures, lateral hydraulic structures, and storage area connections that contain gates.

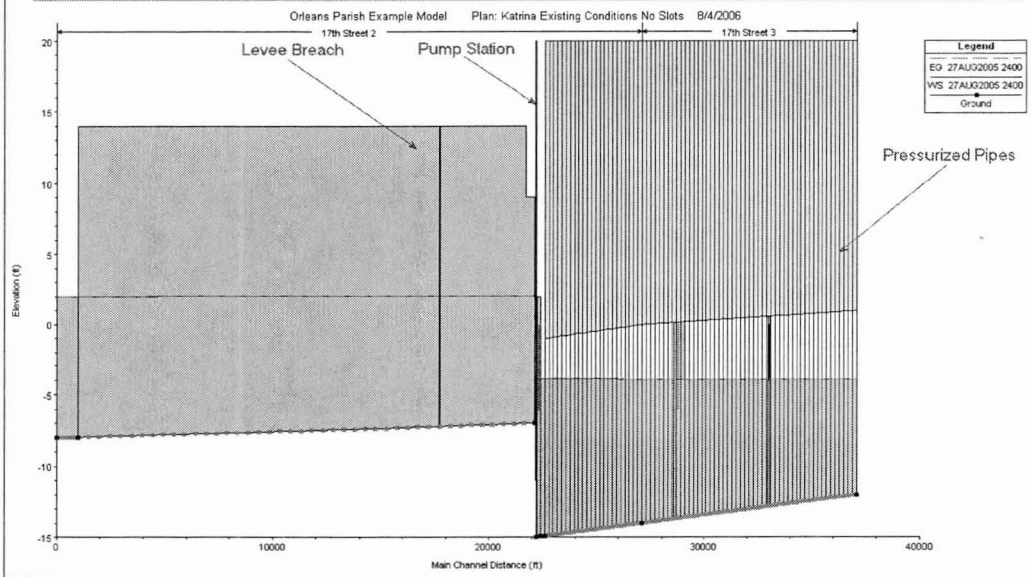
Generic Rules Example
Tampa Bay Water Project Hillsborough River – Harney Canal



Hydrologic Engineering Center 44

Sown above is an example data set in which rules have been applied to control gate settings in order to control flow going from the main river to a canal. This example is a water supply model in which water is taken from the main river for use as drinking water for the city of Tampa Bay, FL. The rules for how much water can be taken on any given day are quite complex. The new HEC-RAS Rules Editor allowed these rules to be modeled in an automated fashion within the simulation process.

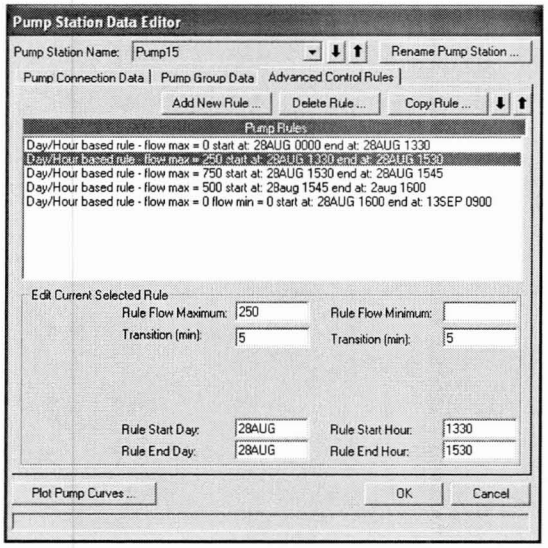
Pressurized Pipes, Pump Station, And Levee Breach Animation



Hydrologic Engineering Center 45

HEC-RAS can be used to model pressurized pipe flow during unsteady flow calculations. This is accomplished by using the Priessmann slot theory applied to the open channel flow equations. To model pressure flow with HEC-RAS, the user must use cross sections with a **Lid** option. The cross section is entered as the bottom half of the pipe and the Lid is entered as the top half of the pipe. Any shape pipe can be modeled, however, the details of the pipe shape will depend on how many points the user puts in for the bottom (cross section) and the top (Lid).

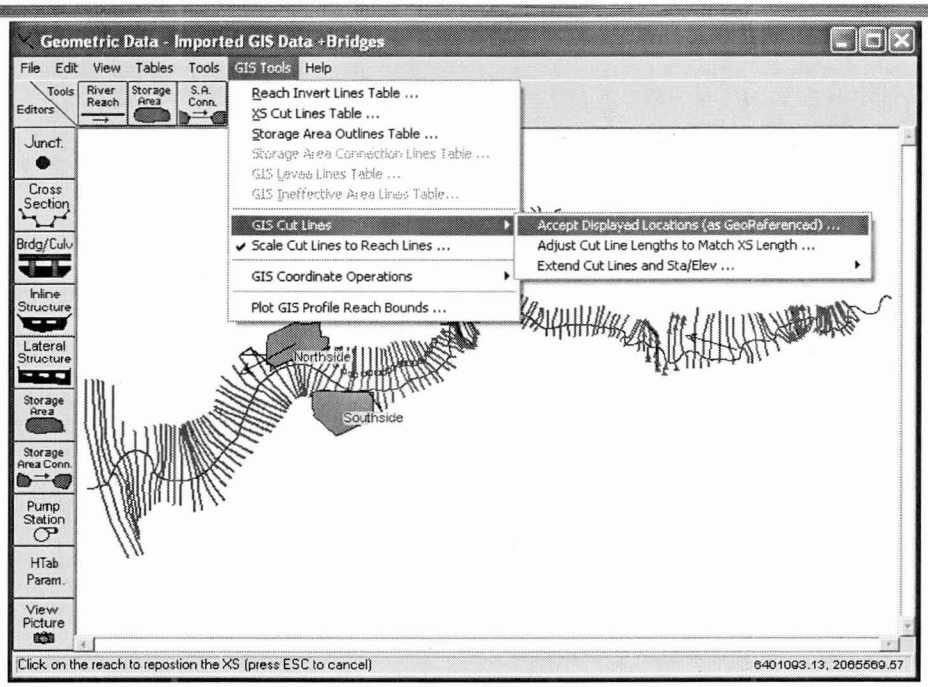
Pump Station Override Rules



Hydrologic Engineering Center 46

Advanced control rules have been added to the Pump Station capabilities of HEC-RAS in order to override normal pump operations. Override rules make it easy to turn pumps on and off based on time of day, as well as target flows and stages from any location in the model. Rules can also be set to override the total pump station maximum or minimum flow capacity.

GIS Tools in HEC-RAS



Hydrologic Engineering Center 47

If cross sections are added to an HEC-RAS model from the cross section editor or through the HEC-2 import feature, the user will need to enter the coordinates of the cross section cut lines to develop a geo-spatially referenced schematic. The cross section cut line coordinates can be entered by selecting **XS Cut Lines Table** from the GIS Tools menu on the Geometric Data editor window. The coordinates for the river network can be entered similarly by selecting **Reach Invert Lines Table** from the GIS Tools menu.

The coordinates that are entered for the cross section cut lines must be consistent with the previously entered GIS data (i.e. if the GIS data is in state plane coordinates, then the user entered data must also be in state plane coordinates). Once all of the cross section cut lines have consistent coordinates, the schematic and XYZ plot will display the cross sections geo-spatially correct.

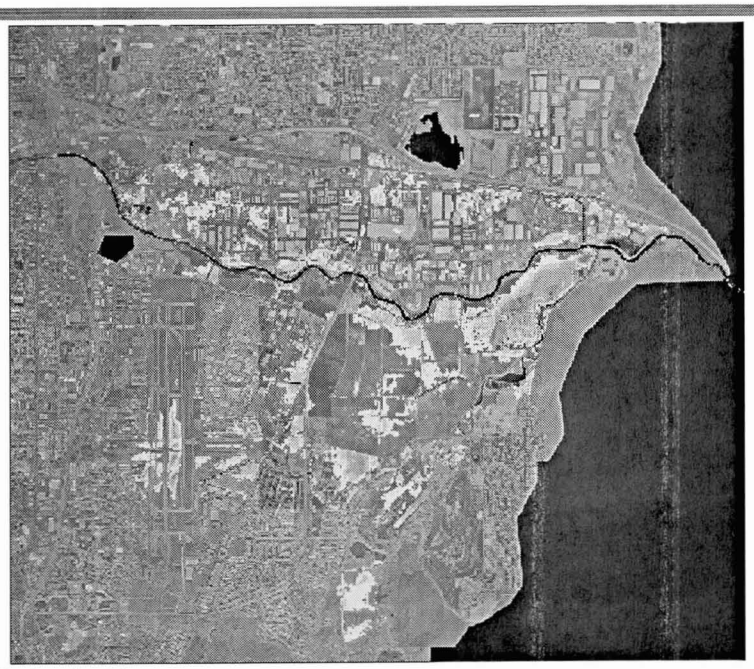
HEC-GeoRAS

- **Pre-processor for generating geometric data for HEC-RAS**
- **Post-processor for mapping and displaying results from hydraulic simulations**

Hydrologic Engineering Center 48

HEC-GeoRAS is a pre and post-processor for assisting engineers in performing river hydraulics studies. GeoRAS is designed to work seamlesslessly with HEC's River Analysis System (HEC-RAS). GeoRAS can be used to pre-process geometric data for import into RAS and can be used as a post-process to RAS to create inundation maps from RAS results.

Truckee River Animation



Hydrologic Engineering Center

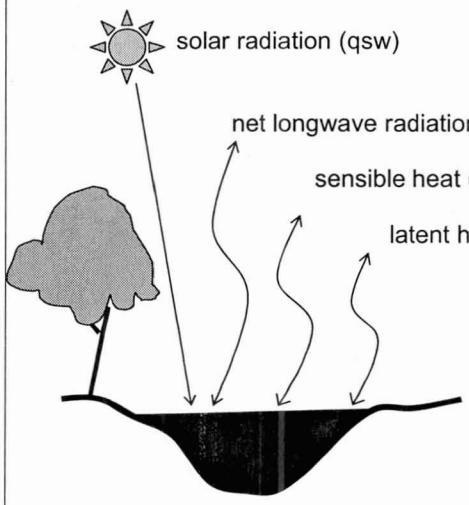
49

Water Quality (Temperature) Model

- Based on unreleased version of CE-QUAL-RIV1
- QUICKEST-ULTIMATE numerical scheme
  - Finite Volume
  - Variable grid size
  - Automatic time step selection
- Full energy budget

Hydrologic Engineering Center 50

Source/Sink Term for Temperature (Energy Budget)



solar radiation (qsw) — f (site location, time of day, day of year, atmospheric turbidity, cloud cover)

net longwave radiation (qlw) — f (air temperature, water temperature)

sensible heat (qh) — f (temperature gradient, wind, a&b)

latent heat (qe) — f (vapor pressure gradient, wind, a&b)

$$q_{net} = q_{sw} + q_{lwn} + q_h + q_e$$

Planned:

- ground heat conduction
- shading (topographic, riparian)

Hydrologic Engineering Center 51

The water quality model will compute the source term. In this case the energy balance. We'll compute each term as a function of atmospheric parameters and water temperature.

Shortwave is a function of location of sun, atmospheric conditions, time of day and year. It may be measured directly or we have a tool to compute.

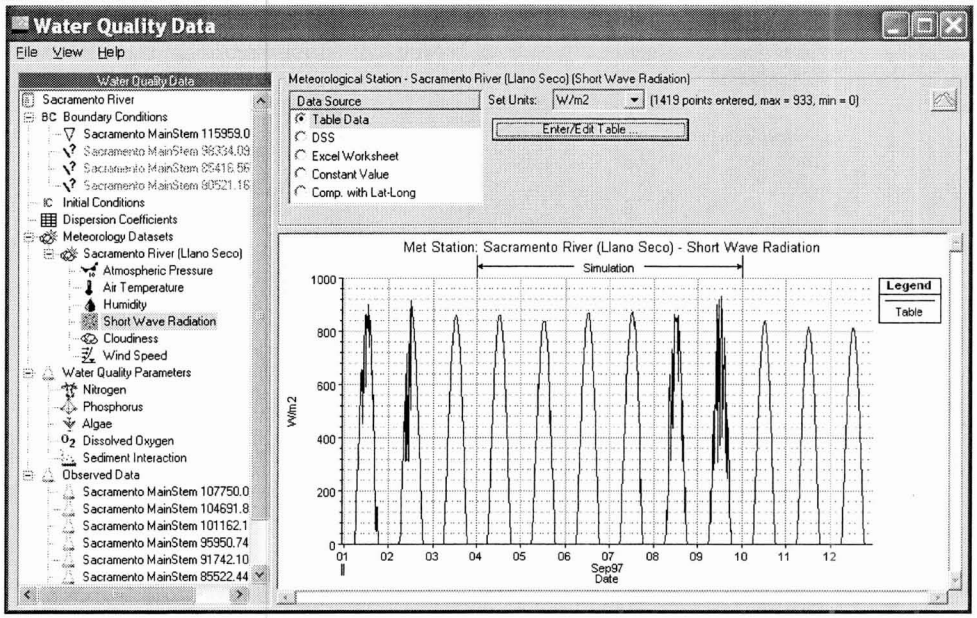
Longwave radiation comes from atmosphere, and it is also radiated by the water surface. Changes with water temperature and air temperature.

Sensible heat is also called conduction. Air mass above the water interface conducts heat into a parcel of water. Function of the temperature gradient and turbulence which we'll characterized with mean wind speed and a fitting parameter.

Latent heat is evaporation. As water evaporates, it changes phase and loses energy in the form of heat. As water evaporates, it cools. This is a function of the vapor gradient (humidity) and turbulence characterized by wind speed and a fitting parameter.

Sensible and latent are collectively called the surface fluxes and are the primary calibration variables.

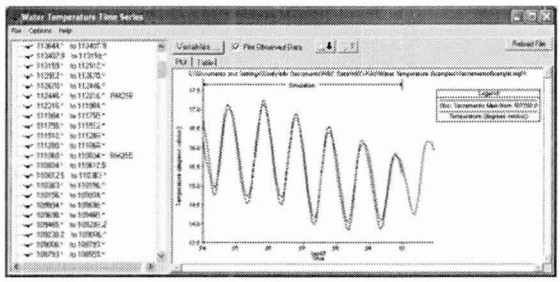
Water Quality Input Data Editor



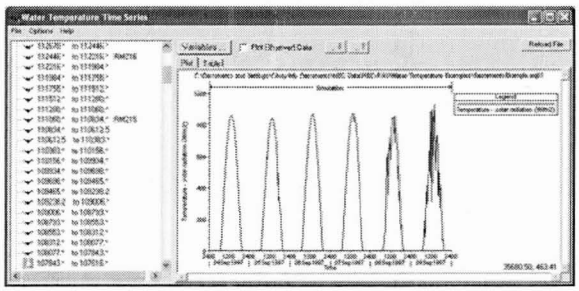
Hydrologic Engineering Center 52

Solar radiation can be entered directly, or can be computed from lat/long and fitting parameters, these plots are measured and computed solar radiation. Useful for synthetic data creation, as well as filling in gaps in data sets.

Time Series Plots



Water temperature

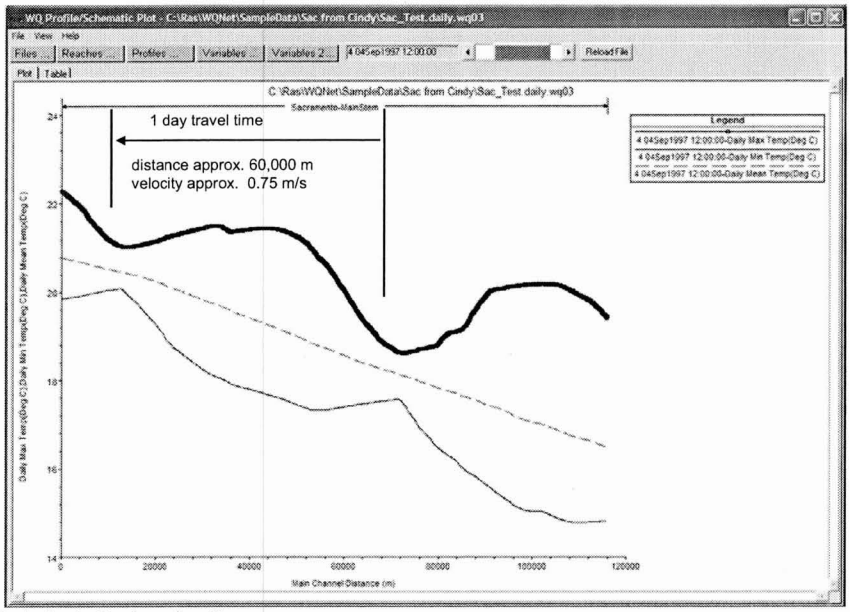


Solar Radiation

Hydrologic Engineering Center 53

These two time series plots compare water temperature and solar radiation, but could be many different variables, including hydrodynamics.

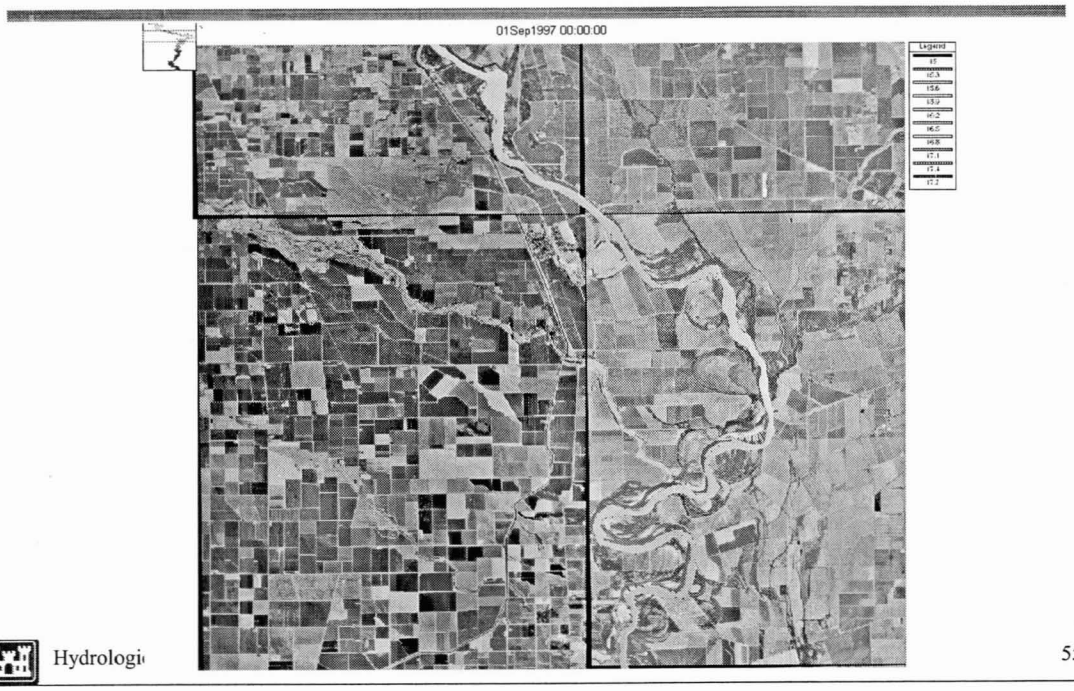
Profile Plot of Temperature



Hydrologic Engineering Center

54

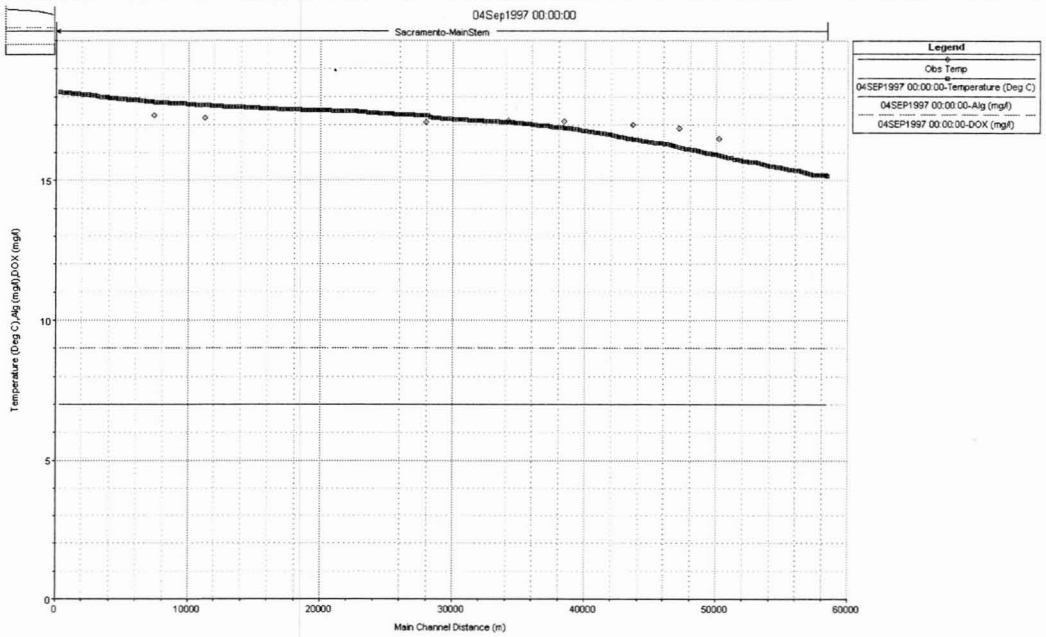
Map View of Temperature Animation



Hydrologi

55

HEC-RAS Animation of Temperature, Algae, and Dissolved Oxygen



Hydrologic Engineering Center

56

Mobile Bed Sediment Transport

- Quasi-Steady Hydrodynamics
- Transport Capacity
- Sediment continuity
- Sorting and Armoring
- Erosion and Deposition
- Graphical User Design

Hydrologic Engineering Center 57

Goals of adding Sediment Routing into HEC-RAS

- Replicate the capabilities of HEC-6
  - Re-coding general capabilities in RAS
  - Differences exist in hydraulic computations
- Add new capabilities beyond current HEC-6 Features
- Improve the capabilities where we have know deficiencies

Hydrologic Engineering Center 58

Transport Potential Functions

- Ackers-White
- Englund-Hansen
- Laursen (Copland)
- Myer-Peter-Meuler
- Toffaleti
- Yang (Sand and Gravel)
- Wilcock

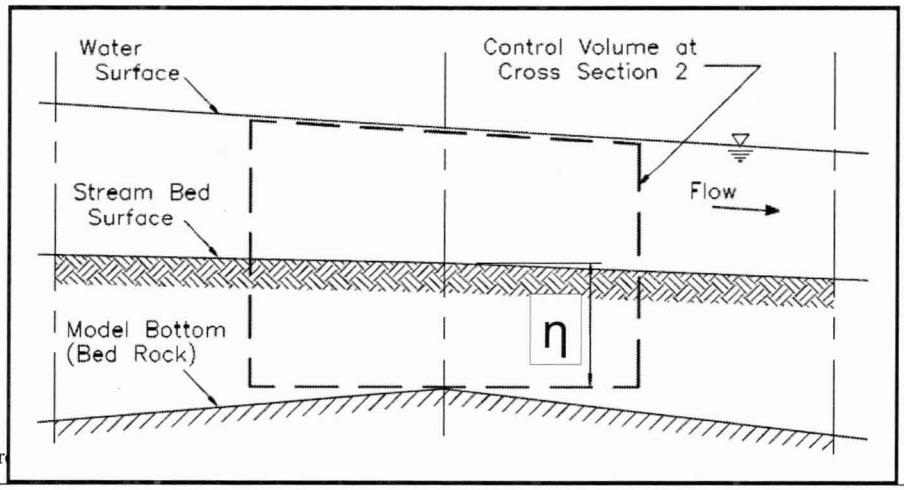
Hydrologic Engineering Center 59

Transport Capacity

- Bed Material and Inflowing Load divided into separate grain classes (up to 20)
- Transport potential is calculated for each grain size
- Transport Capacity = (Transport Potential for each grain size) X (fraction of that material in active layer of bed)

Hydrologic Engineering Center 60

Sediment Continuity: Exner Equation



Hydr

61

Lambda = porosity of the material

Nu = depth of erodeable material

Qs = average sediment discharge rate during the time step

Temporal Constraints on Eroding and Depositing

- **Erosion and deposition does not occur instantaneously.**
- **Deposition is based on settling velocity:**
  - **Deposition efficiency coefficient =**
- **Erosion is based on "Characteristic Flow Length"**
  - **Erosion = (Gs – Qs) x Ce Entrainment Coefficient**
  - **Where:**

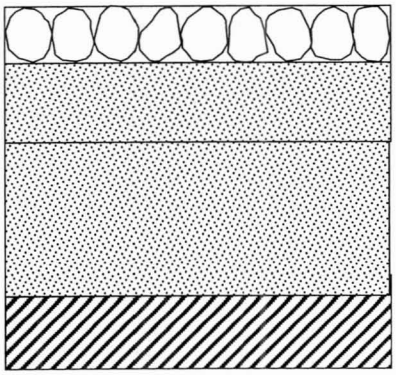
Hydrologic Engineering Center 62

Vs = settling velocity by grain size; De = effective depth of water (average water depth)

Gs = Transport Capacity; Qs = Current load in the flow field; Ce = Entrainment coefficient

Cover layer constraint is also applied when you have erosion occurring.

Sorting and Armoring



Diagramed and Conceptualized HEC 6 Code

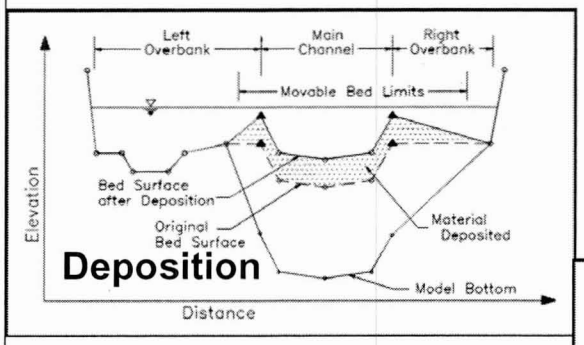
Exner 5 implemented Currently in RAS

Also Simple Active Layer Method

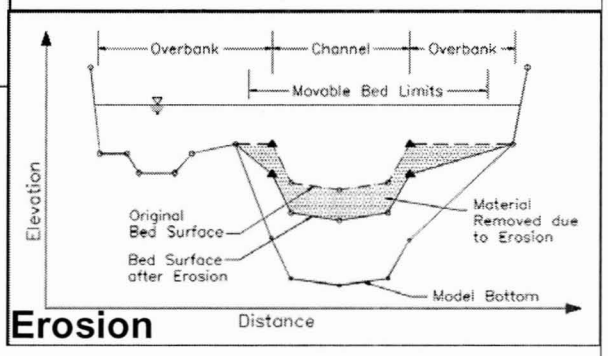
* Erosion can be further constrained by the cover Layer

Hydrologic Engineering Center 63

Erosion and Deposition to RAS Cross Sections

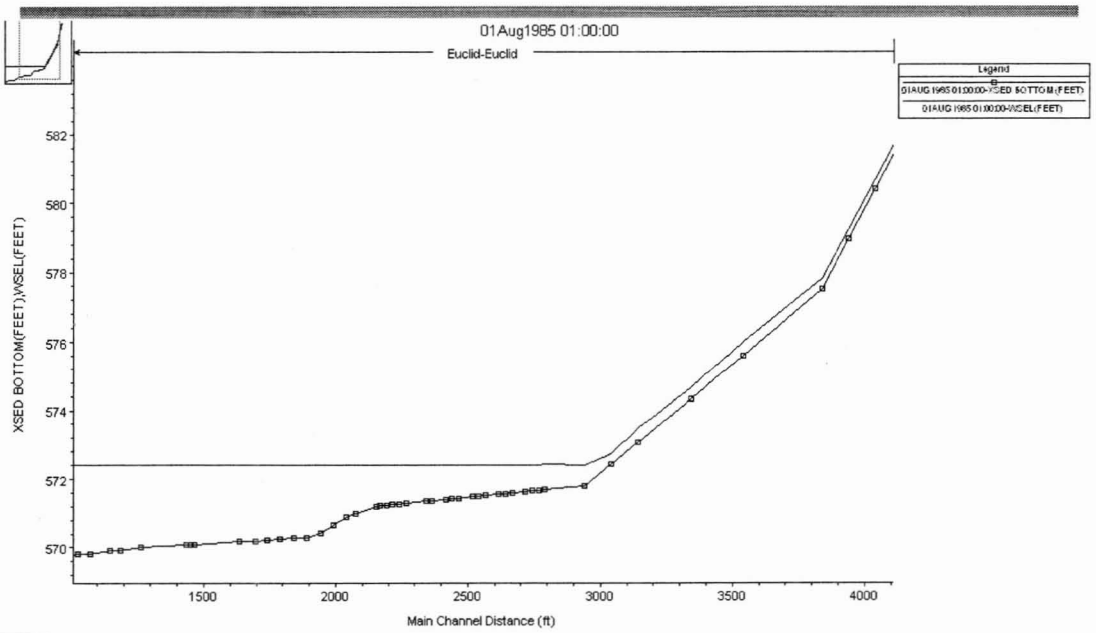


RAS computations modified to compute bed changes and modify cross sections before each time step



- **Cross Sections**
- **Bridges**

Animation of Bed Movement



Hydrologic Engineering Center 65

Bed gradations vary per cross section, courser at upstream boundary, finer at second cross section.

Data Requirements for Unsteady Flow Models

Cameron Ackerman, P.E.

1

The objective of this lecture is to discuss geomtric data options in HEC-RAS and how the geometric pre-processsor uses the geometric to preprocess hydraulic tables for use in unsteady flow simulations.

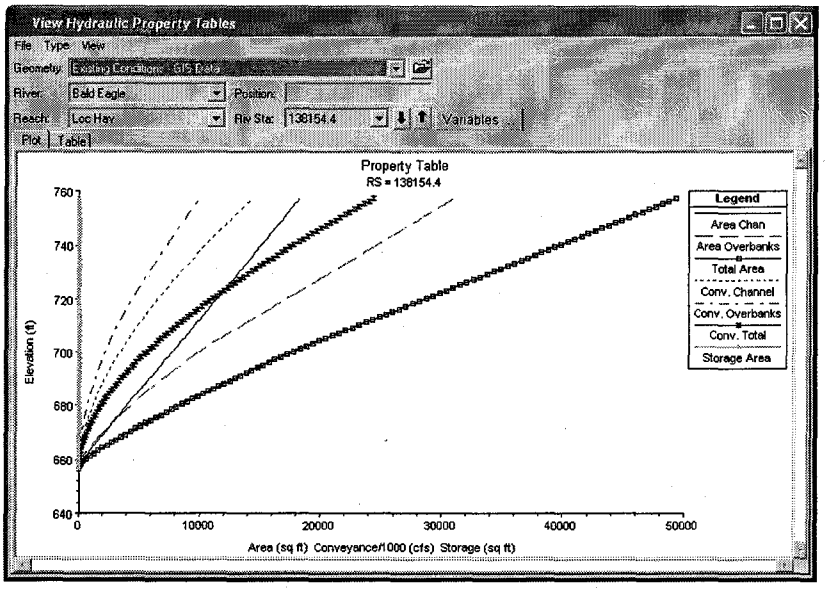
Geometry Preprocessor

- What does it do?
  - Processes geometric data into a series of hydraulic tables and rating curves.
- Why do we use it for unsteady flow?
  - Instead of calculating hydraulic variables for each cross-section during each iteration, the program interpolates the hydraulic variables from the tables.

2

The preprocessor is used to process the geometric data into a series of hydraulic property tables and rating curves. This is done in order to speed up the unsteady flow calculations. Instead of calculating hydraulic variables for each cross-section during each iteration, the program interpolates the hydraulic variables from the tables. The preprocessor must be executed at at least once, but then only needs to be re-executed if something in the geometric data has changed.

Hydraulic Property Plot



3

Cross sections are processed into tables of elevation versus hydraulic properties of areas, conveyances, and storage. Each table contains a minimum of 21 points (a zero point at the invert and 20 computed values). The user is required to set an interval to be used for spacing the points in the cross section tables. The interval can be the same for all cross sections or it can vary from cross section to cross section. This interval is very important, in that it will define the limits of the table that is built for each cross section. On one hand, the interval must be large enough to encompass the full range of stages that may be incurred during the unsteady flow simulations. On the other hand, if the interval is too large, the tables will not have enough detail to accurately depict changes in area, conveyance, and storage with respect to elevation.

Conveyance Calculations

- Manning's Equation

$$Q = K\,S_f^{1/2}$$

$$K = \frac{1.486}{n} A\,R^{2/3}$$

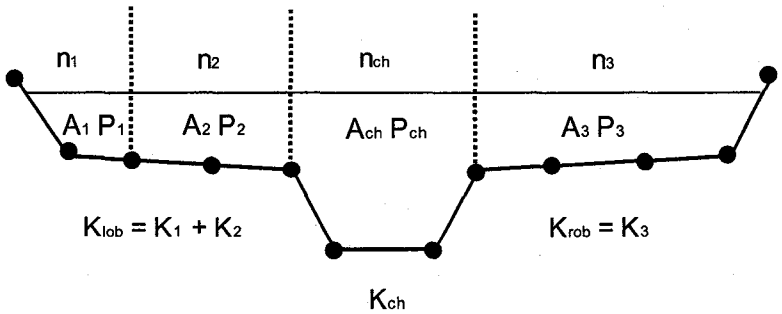
4

Conveyance is calculated from the above form of Manning's equation (based on English units):

where:

K	=	conveyance for subdivision
n	=	Manning's roughness coefficient for subdivision
A	=	flow area for subdivision
R	=	hydraulic radius for subdivision (area / wetted perimeter)
S_f	=	friction slope

Conveyance Calculations

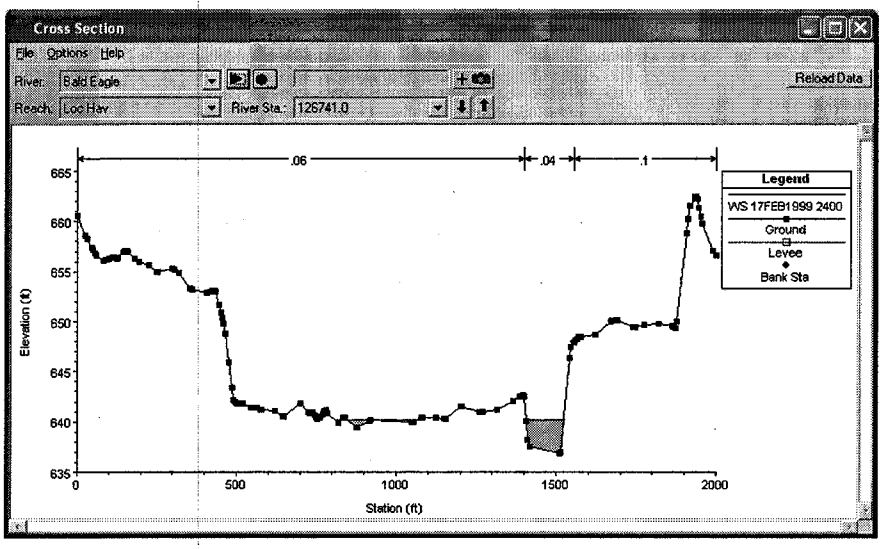


5

The determination of total conveyance and the velocity coefficient for a cross section requires that flow be subdivided into units for which the velocity is uniformly distributed. The approach used in HEC-RAS is to subdivide flow in the overbank areas using the input cross section n-value break points (locations where n-values change) as the basis for subdivision.

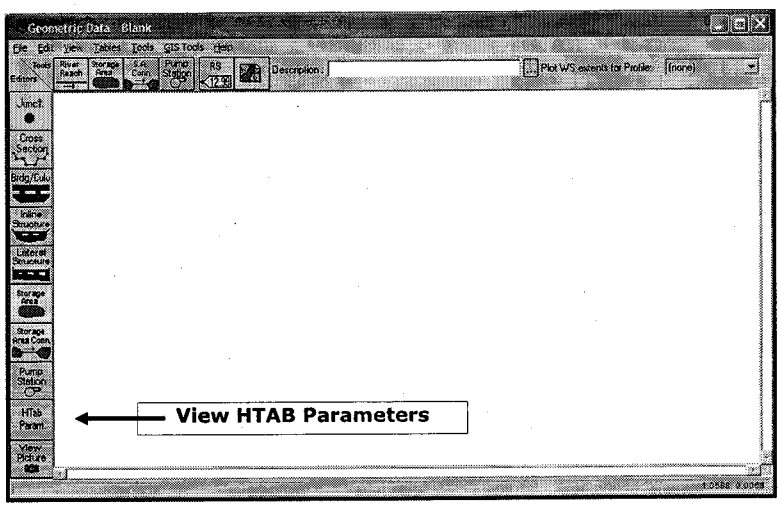
The program sums up all the incremental conveyances in the overbanks to obtain a conveyance for the left overbank and the right overbank. The main channel conveyance is normally computed as a single conveyance element. The total conveyance for the cross section is obtained by summing the three subdivision conveyances (left, channel, and right).

Cross Section Example



6

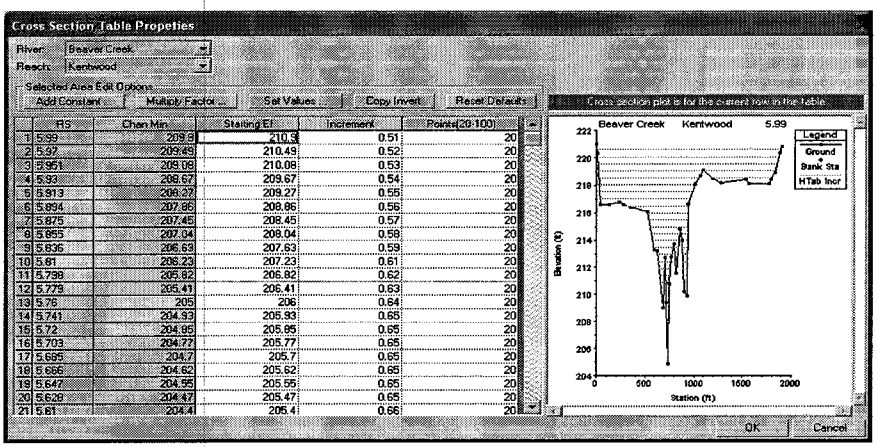
Cross Section Table Parameters



7

The **Geometric Data Editor** is shown in the above figure. From this editor, select the button labeled **Htab Param.** to view and edit the cross section table parameters.

Cross Section Table Parameters



- Starting Elevation
- Increment
- Number of Points

8

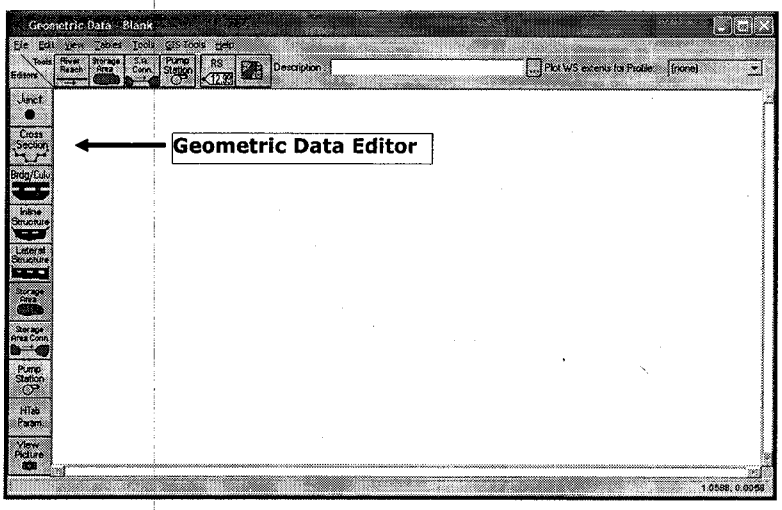
As shown in the figure, the table contains three columns in which the user can enter information: starting elevation; increment; and Number of Points. The first time the user opens this editor all of the columns are automatically filled. The starting elevation columns are automatically filled to an elevation one foot higher than the invert, however, the user can change these values to whatever they want. The second and third columns are used for the table increment and the number of points. These two variables will describe the extent to which the table encompasses the cross section data. A default value will be set for the increment and the number of points. Normally the increment will be set to one foot, and the number of points will be set to a value that will allow the table to extend to the top of the cross section. If this combination would end up with less than 20 points, then the number of points is set to 20 and the increment is reduced to get the table to the top of the cross section. The user can set these values individually for each cross section, or they can highlight a series of cross sections and use the **Set Values** button to enter the value for all of the highlighted sections. Other options are available to multiply highlighted fields by a factor or add a constant to all of them. Additionally, cut, copy, and paste buttons are available for manipulating the data.

Entering Geometric Parameters

- Manning's n Values
- Ineffective Flow Areas
  - Non-Permanent vs. Permanent
  - Normal vs. Multiple Blocks
- Blocked Obstruction
- Levees

9

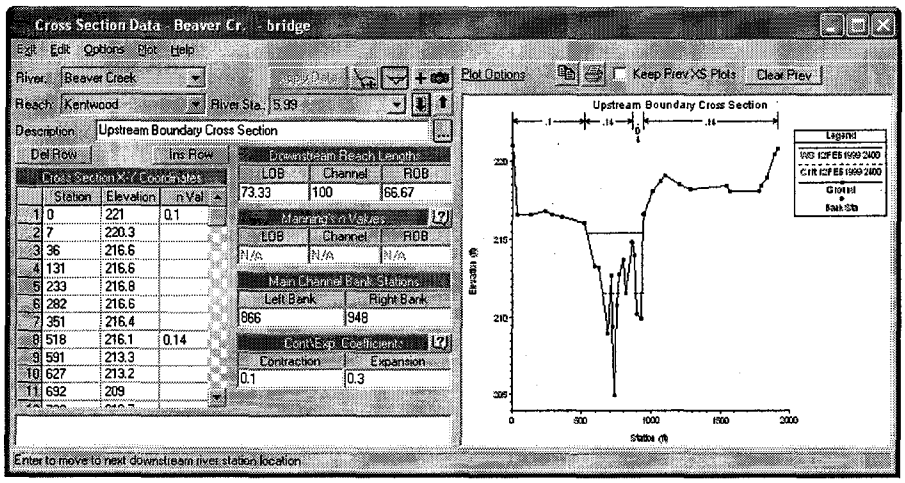
Entering Geometric Parameters



10

The **Geometric Data Editor** is shown in the above figure. From this editor, select the button labeled **Cross Section** to view and edit the cross section geometry.

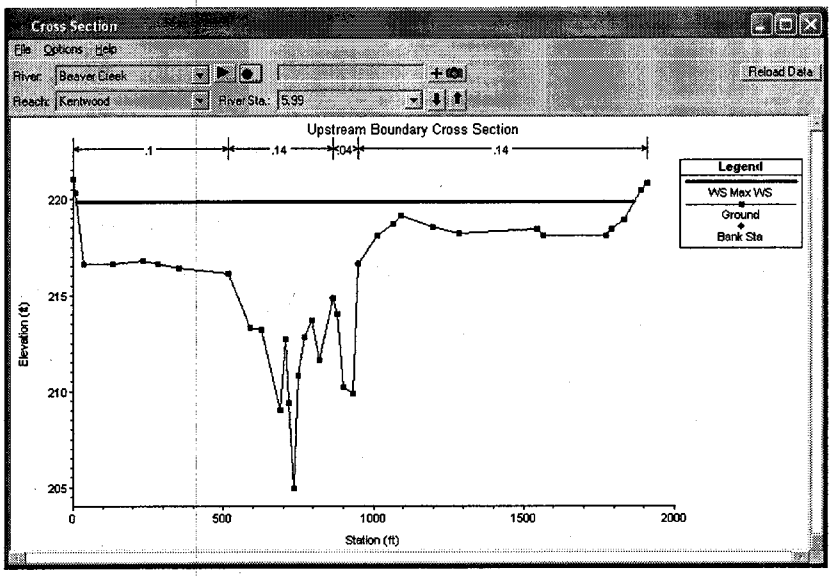
Manning's n Values



11

Selection of an appropriate value for Manning's n is very significant to the accuracy of the computed water surface profiles. The value of Manning's n is highly variable and depends on a number of factors including: surface roughness; vegetation; channel irregularities; channel alignment; scour and deposition; obstructions; size and shape of the channel; stage and discharge; seasonal change; temperature; and suspended material and bedload.

Manning's n Values



12

The above figure illustrates the horizontal variation in Manning's n values along the cross section.

There are several references a user can access that show Manning's n values for typical channels. An extensive compilation of n values for streams and floodplains can be found in Chow's book "Open-Channel Hydraulics" [Chow, 1959]. Excerpts from Chow's book, for the most common types of channels, are shown in Table 3.1 of the Hydraulic Reference Manual for HEC-RAS version 3.0 on pages 3-14 and 3-15.

Entering Geometric Parameters

- Manning's n Values
- Ineffective Flow Areas
  - Non-Permanent vs. Permanent
  - Normal vs. Multiple Blocks
- Blocked Obstruction
- Levees

13

Ineffective Flow Areas

- Areas where water is not actively being conveyed. The velocity of water in the downstream direction is close to zero.
- This water is included in the storage calculations and other wetted cross section parameters, but it is not included as part of the active flow area.

14

This option allows the user to define areas of the cross section that will contain water that is not actively being conveyed (ineffective flow). Ineffective flow areas are often used to describe portions of a cross section in which water will pond, but the velocity of that water, in the downstream direction, is close to zero. This water is included in the storage calculations and other wetted cross section parameters, but it is not included as part of the active flow area.

In unsteady flow modeling it is important to account for channel storage as well as conveyance. In steady flow modeling it is sometimes possible to completely ignore portions of the cross section that are dead storage, this is not the case in unsteady flow modeling because the hydrograph volume must be recognized. It may be useful to think of storage-only portions of a cross section as having extremely high n-values, which effectively nulls the conveyance.

Ineffective Flow Areas Requirements

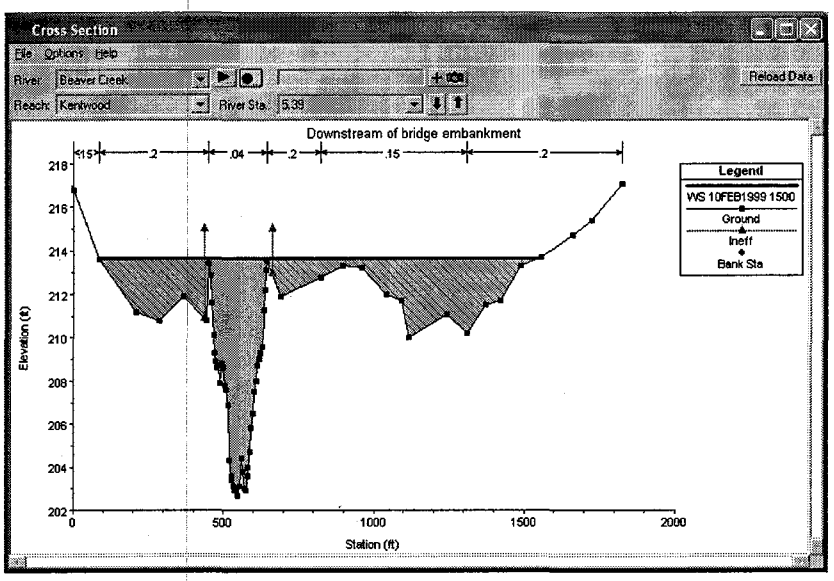
- Ineffective Flow Stations
  - Left and/or right ineffective flow stations denote the location of the ineffective flow areas along the cross section.
- Trigger Elevations
  - Water elevation at which the ineffective flow area begins to convey flow (non-permanent), or remains ineffective (permanent).

15

Setting an ineffective flow area requires at least one bank station and elevation. The bank station denotes the location of the ineffective flow area along the cross section. There are two different methods for setting bank stations (normal and multiple blocks).

When the water surface surpasses the trigger elevation, the ineffective flow area will either begin to convey flow or remain ineffective depending upon its type (permanent on non-permanent). The concepts of permanent and non-permanent ineffective flow areas are addressed in the next set of slides.

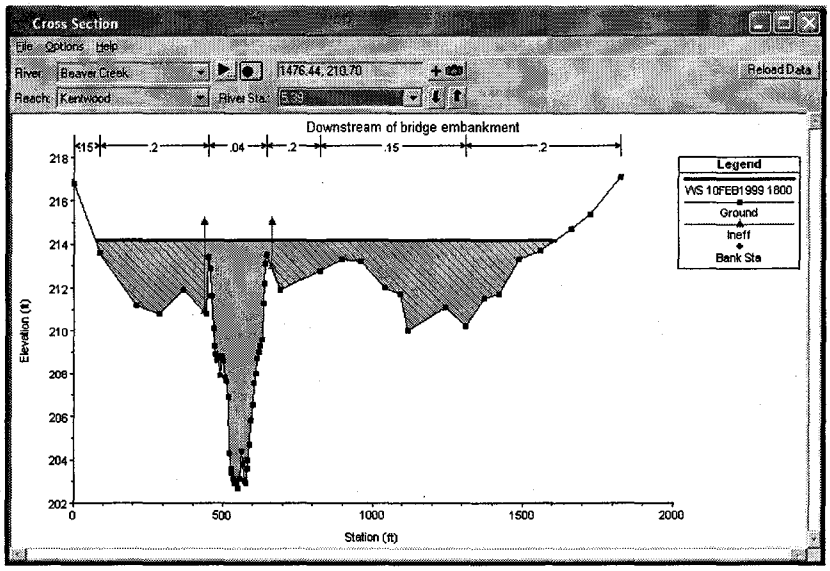
Non-Permanent Ineffective Flow Area



16

As shown in the slide, there are two ineffective flow areas (left bank and right bank). The ineffective flow areas are set to non-permanent. This means that the left bank and right bank areas will not convey flow until the trigger elevation has been reached. The trigger elevation is denoted by the end point of the two arrows. In this example the trigger elevation is 216 feet.

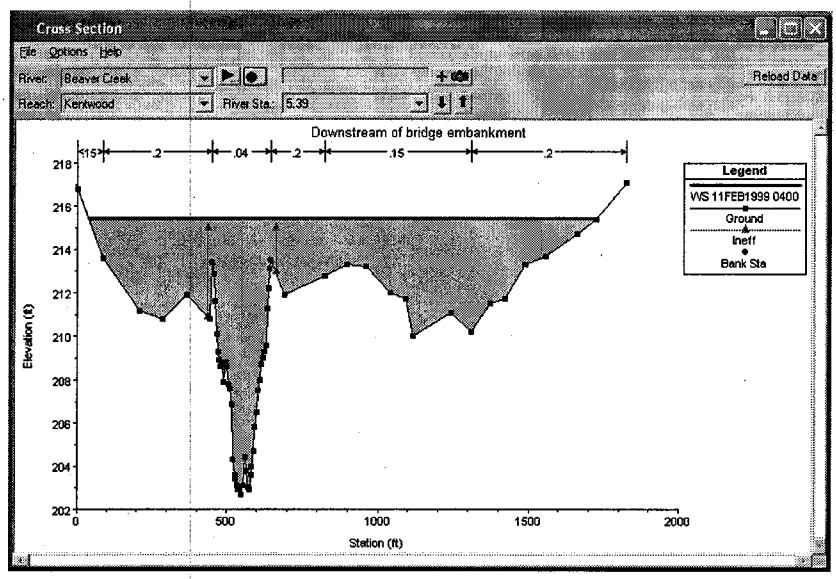
Non-Permanent Ineffective Flow Area



17

The water surface is rising but has not exceeded the trigger elevation. The rising water surface has increased the storage in the left and right overbanks but has not increased the conveyance in these areas because ineffective flow areas account for storage and not conveyance.

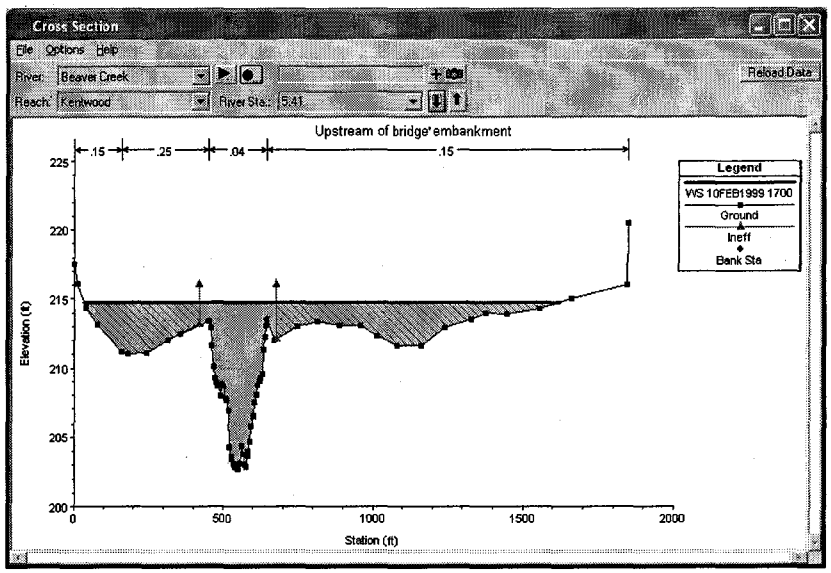
Non-Permanent Ineffective Flow Area



18

For non-permanent ineffective flow areas, once the water surface is higher than the trigger elevation, the entire ineffective flow area becomes effective (as shown in the slide). Water is assumed to be able to move freely in that area based on the roughness, wetted perimeter, and area of each subdivision. The left and right overbanks are no longer considered storage but are now active flow areas.

Permanent Ineffective Flow Area

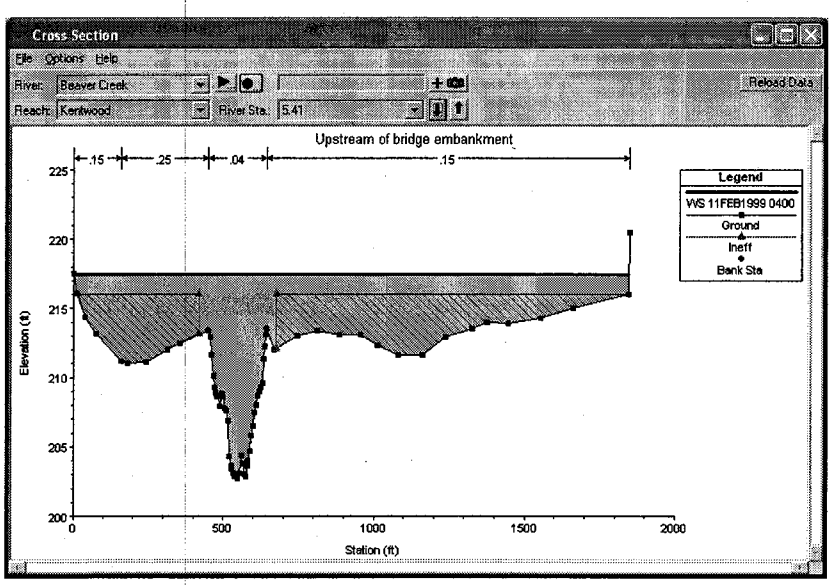


19

Occasionally you may need to have ineffective flow areas remain ineffective permanently. The ineffective flow areas can be set to the permanent mode individually from the cross section editor, or through a table from the geometric data editor.

In this example, the same ineffective flow areas were used except they were set to permanent. As seen in the slide when the water surface is below the trigger elevation the permanent ineffective flow area behaves like the non-permanent area.

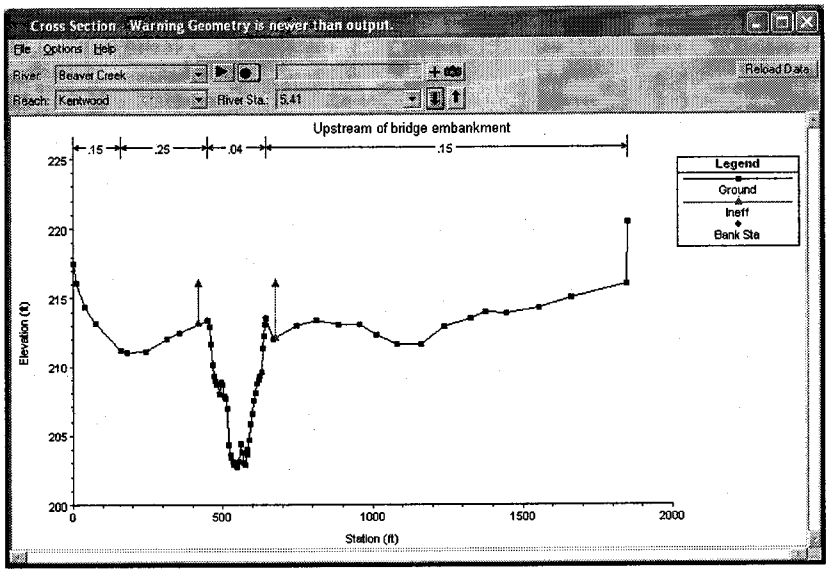
Permanent Ineffective Flow Area



20

For permanent ineffective flow areas, when the water surface elevation surpasses the trigger elevation, the area below the trigger elevation remains ineffective. Water above the trigger elevation is assumed to convey flow.

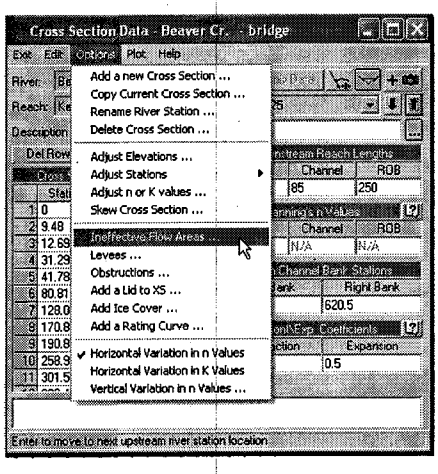
Normal Ineffective Flow Areas

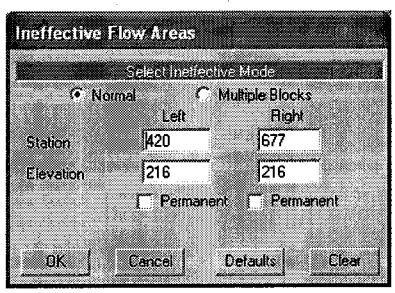


21

The normal ineffective flow area option allows the user to define a left station and elevation and a right station and elevation (normal ineffective areas). When this option is used, and if the water surface is below the established ineffective elevations, the areas to the left of the left station and to the right of the right station are considered ineffective.

Normal Ineffective Flow Areas

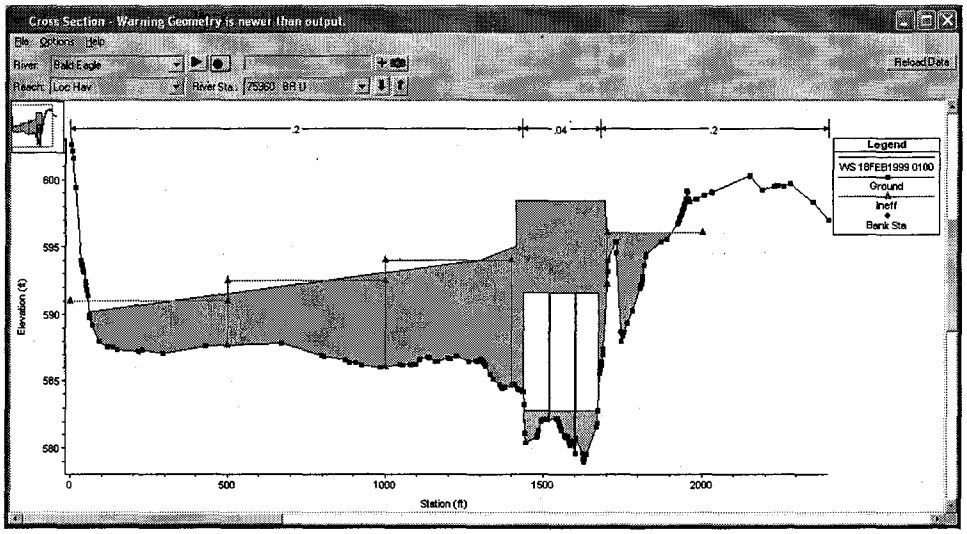




22

Set the normal ineffective flow area locations by selecting **Ineffective Flow Areas** under the **Options** menu of the **Cross Section Data Editor**. In the ineffective flow area window, select the radio button labeled **Normal**. Notice the box labeled **Permanent**, check this box if the ineffective flow area never conveys flow.

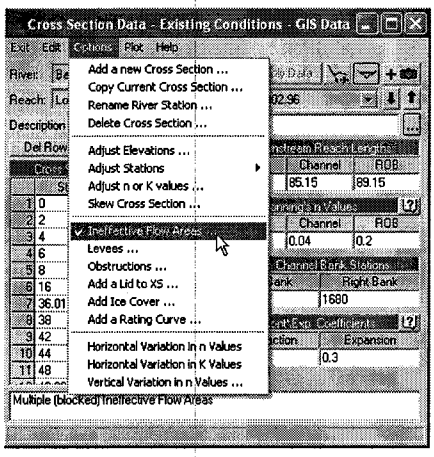
Blocked Ineffective Flow Areas

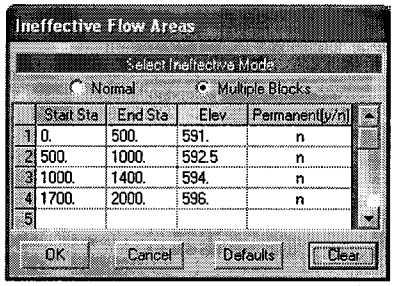


23

The multiple block ineffective area option allows for the establishment of blocked ineffective flow area . Multiple blocked ineffective flow areas are useful for bridge geometry with slopes or multiple openings.

Blocked Ineffective Flow Areas





24

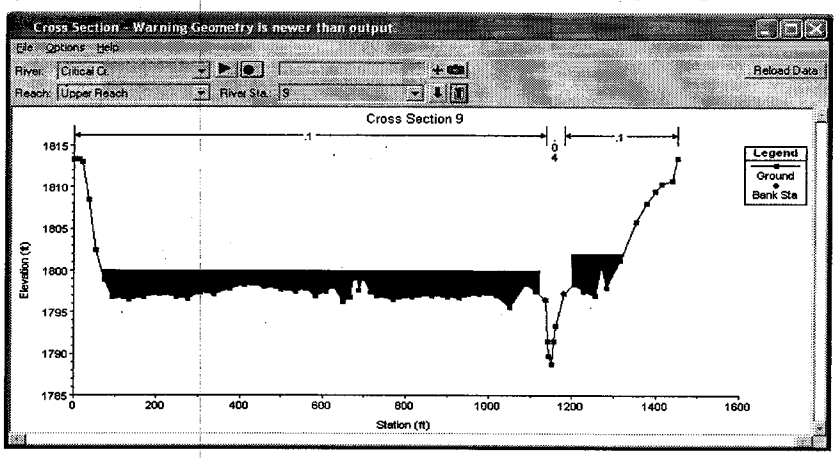
Blocked ineffective flow areas require the user to enter an elevation, a left station, and a right station for each ineffective block. Up to ten blocked ineffective flow areas can be entered at each cross section. Notice the column labeled **Permanent (y/n)**, change this column to "y" for permanent ineffective flow areas.

Entering Geometric Parameters

- Manning's n Values
- Ineffective Flow Areas
  - Non-Permanent vs. Permanent
  - Normal vs. Multiple Blocks
- Blocked Obstruction
- Levees

25

Blocked Obstructions

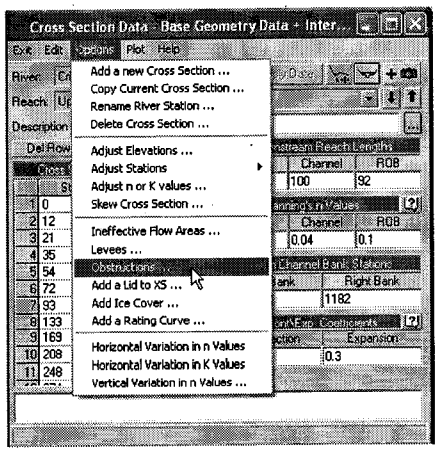


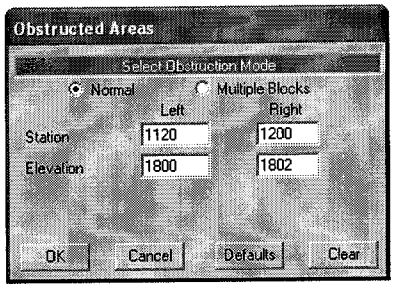
- Decrease Flow Area
- Add Wetted Perimeter

26

Blocked obstructions allow the user to define areas of the cross section that will be permanently blocked out. Blocked obstructions decrease flow area and add wetted perimeter when the water comes in contact with the obstruction. A blocked obstruction does not prevent water from going outside of the obstruction.

Blocked Obstructions





27

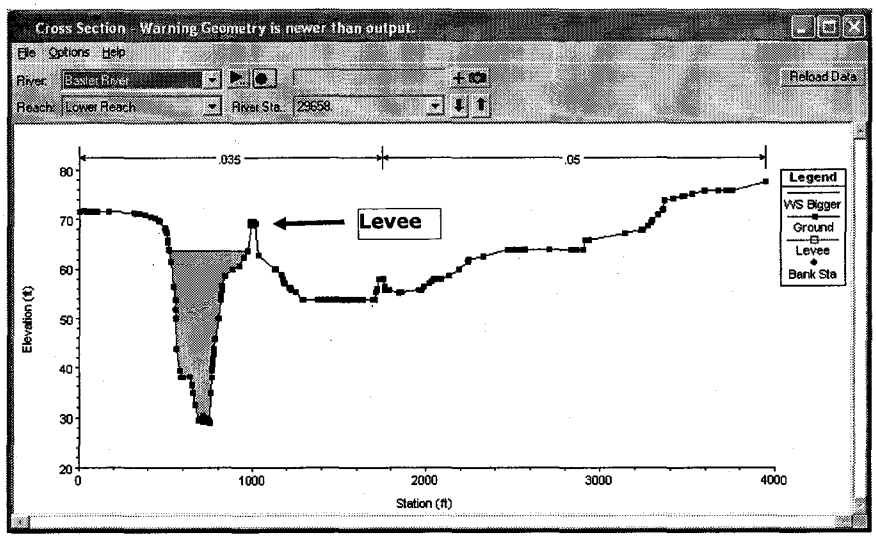
The user may define a left station and elevation and a right station and elevation (normal blocked areas). When this option is used, the area to the left of the left station and to the right of the right station will be completely blocked out.

Entering Geometric Parameters

- Manning's n Values
- Ineffective Flow Areas
  - Non-Permanent vs. Permanent
  - Normal vs. Multiple Blocks
- Blocked Obstruction
- Levees

28

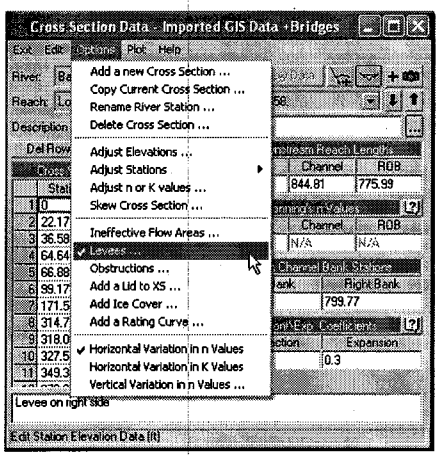
Levees

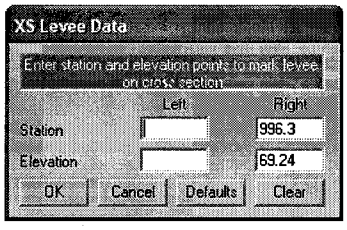


29

This option allows the user to establish a left and/or right levee station and elevation on any cross section. When levees are established, no water can go to the left of the left levee station or to the right of the right levee station until either of the levee elevations are exceeded. Levee stations must be defined explicitly, or the program assumes that water can go anywhere within the cross section. An example of a cross section with a levee on the left side is shown in the figures. In this example the levee station and elevation is associated with an existing point on the cross section.

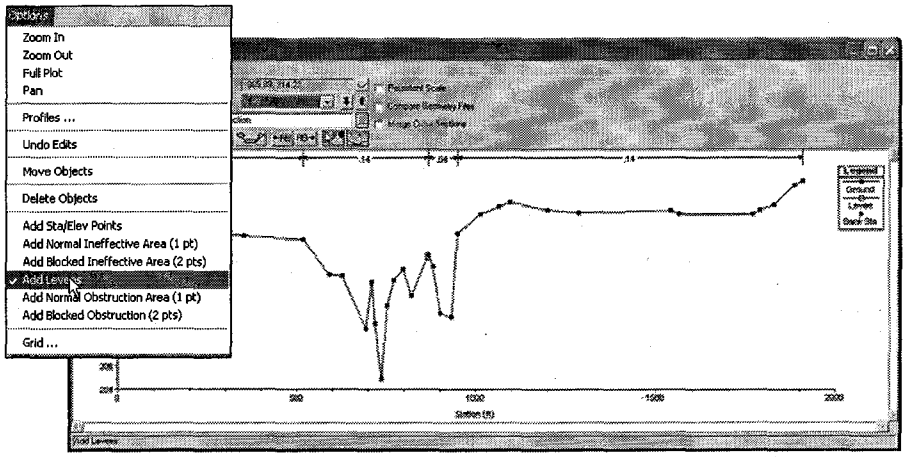
Levees


Cross Section Data - Imported GIS Data +Bridges
Exit Edit Options Plot Help
Add a new Cross Section ...
Copy Current Cross Section ...
Rename River Station ...
Delete Cross Section ...
Adjust Elevations ...
Adjust Stations
Adjust n or K values ...
Skew Cross Section ...
Ineffective Flow Areas ...
Levees ...
Obstructions ...
Add a Lid to XS ...
Add Ice Cover ...
Add a Rating Curve ...
Horizontal Variation in n Values
Horizontal Variation in K Values
Vertical Variation in n Values ...
Channel
ROB
844.81
775.99
N/A
N/A
Right Bank
799.77
Expansion
0.3
Levee on right side
Edit Station Elevation Data (ft)


XS Levee Data
Enter station and elevation points to mark levee on cross section
Left
Right
Station
996.3
Elevation
69.24
OK
Cancel
Defaults
Clear

30

Graphical Cross Section Editor



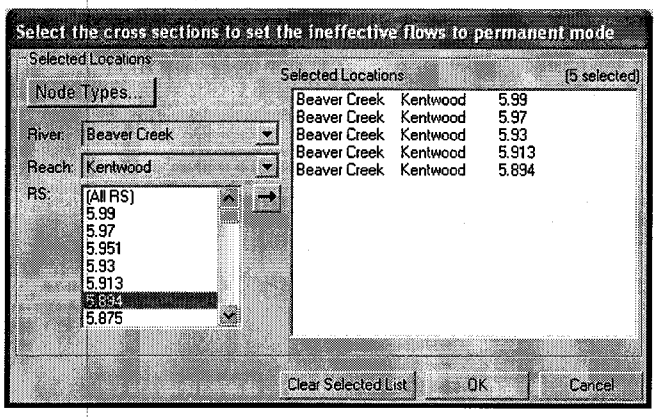
Located in the **Geometric Data Editor** under the **Tools** menu.

31

A graphical cross section editor is available from the **Tools** menu of the **Geometric Data Editor**. The user has the option to move objects (objects are ground points, main channel bank stations, ineffective flow areas, levees, and blocked obstructions), delete objects, or add new objects.

To move an object, the user first selects **Move Objects** from the **Options** menu. Then move the mouse pointer over the object that you want to move, press down the left mouse button, and then move the object. When you are finished moving the object, simply release the left mouse button and the object will be moved. To delete an object, first select **Delete Objects** from the **Options** menu. Next, move the mouse pointer over the object that you would like to delete and click the left mouse button. Whatever object is closest to the mouse pointer will be deleted. To add an object to the cross section, first select the type of object you want to add from the available list under the **Options** menu. Once you have selected an object type to add, move the mouse pointer to the location where you would like to add it and click the left mouse button. If the object that you are adding requires more than one point, such as blocked ineffective flow areas and blocked obstructions, then continue to move the mouse pointer and click the left mouse button to add the additional points.

Setting Ineffective Areas to Permanent



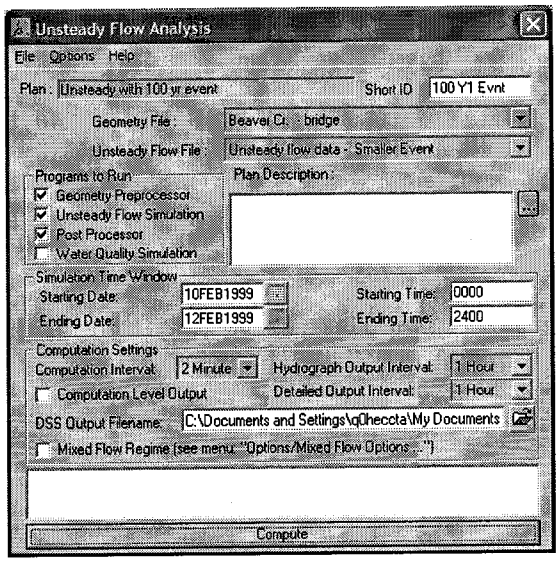
Located in the **Geometric Data Editor** under the **Tools** menu.

32

The default in HEC-RAS is to set ineffective flow areas to non-permanent. To change a set of river stations from non-permanent to permanent, select **Setting Ineffective Areas to Permanent Mode...** under the **Tools** menu located in the **Geometric Data Editor**. Then select the proper river, reach, and river stations.

This is especially useful after using the **Graphical Cross Section Editor** because there is no option in the editor to change between non-permanent and permanent.

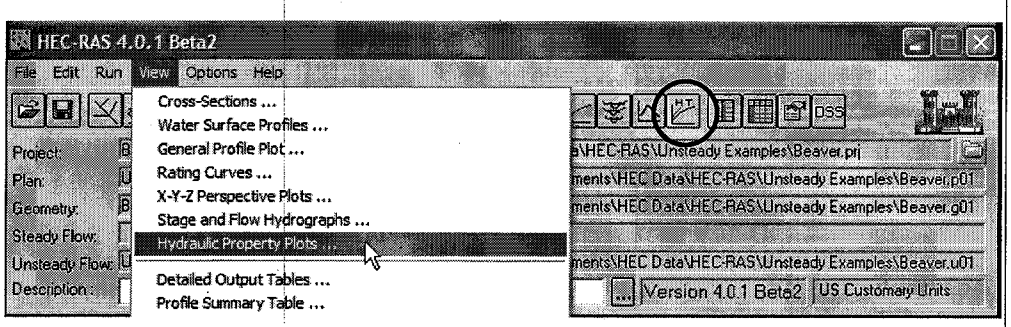
Geometry Preprocessor



33

The geometric preprocessor is located in the **Unsteady Flow Analysis Window**, found from the main program window under the **Run** menu.

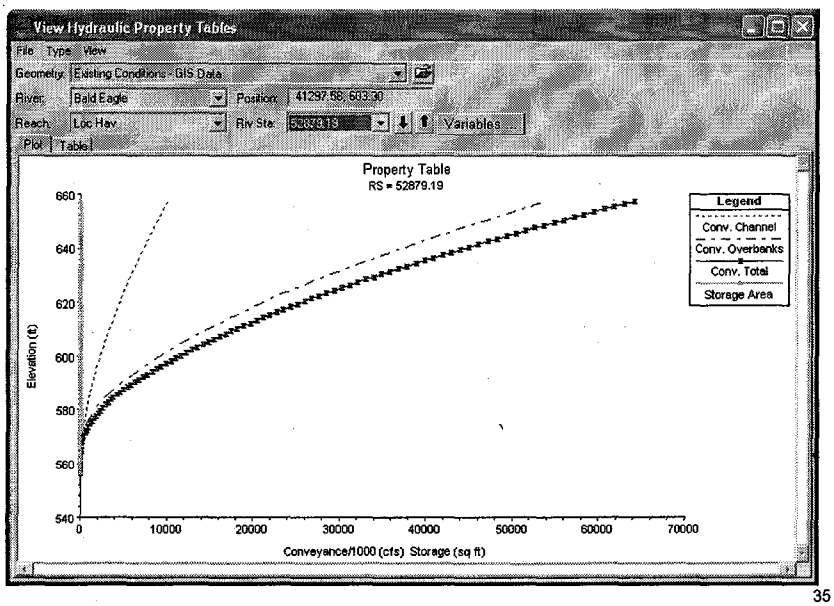
Hydraulic Property Tables



34

View output from the geometric preprocessor by examining the hydraulic property tables and plots. The output is located under the **View** menu, or the shaded button in the figure.

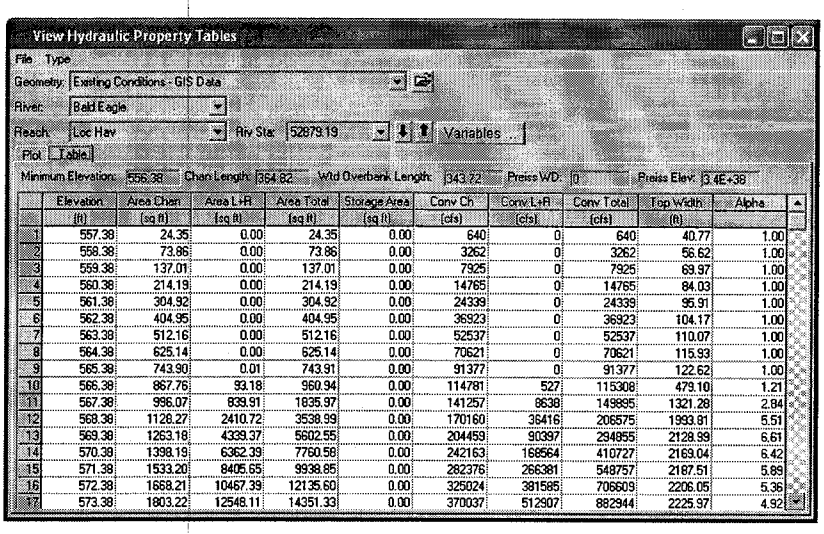
Hydraulic Property Plot



35

Output can be viewed in graphical form. Viewing the graphical output is a good diagnostic tool for examining cross section geometry. The relationship of area, storage, and conveyance should be examined for abrupt changes with elevation. Any abrupt changes should be reviewed to determine the overall significance within that particular model.

Hydraulic Property Table



	Elevation (ft)	Area Chan (sq ft)	Area L+R (sq ft)	Area Total (sq ft)	Storage Area (sq ft)	Conv Ch (cfs)	Conv L+R (cfs)	Conv Total (cfs)	Top Width (ft)	Alpha
1	557.38	24.35	0.00	24.35	0.00	640	0	640	40.77	1.00
2	558.38	73.86	0.00	73.86	0.00	3262	0	3262	56.62	1.00
3	559.38	137.01	0.00	137.01	0.00	7925	0	7925	69.97	1.00
4	560.38	214.19	0.00	214.19	0.00	14765	0	14765	84.03	1.00
5	561.38	304.92	0.00	304.92	0.00	24339	0	24339	95.91	1.00
6	562.38	404.95	0.00	404.95	0.00	36923	0	36923	104.17	1.00
7	563.38	512.16	0.00	512.16	0.00	52537	0	52537	110.07	1.00
8	564.38	625.14	0.00	625.14	0.00	70621	0	70621	115.93	1.00
9	565.38	743.90	0.01	743.91	0.00	91377	0	91377	122.62	1.00
10	566.38	867.76	93.18	960.94	0.00	114781	527	115308	479.10	1.21
11	567.38	996.07	839.91	1835.97	0.00	141257	8638	149895	1321.28	2.84
12	568.38	1128.27	2410.72	3538.99	0.00	170160	36416	206575	1993.81	5.51
13	569.38	1263.18	4339.37	5602.55	0.00	204459	90397	294855	2128.99	6.61
14	570.38	1398.19	6362.39	7760.58	0.00	242163	168564	410727	2169.04	6.42
15	571.38	1533.20	8405.65	9938.85	0.00	282376	266381	548757	2187.51	5.89
16	572.38	1668.21	10467.39	12135.60	0.00	325024	381585	706609	2206.05	5.36
17	573.38	1803.22	12548.11	14351.33	0.00	370037	512907	882944	2225.97	4.92

36

Detailed output is also available for each computed increment. Detailed output can be examined to determine a specific elevation or station where a problem is occurring.

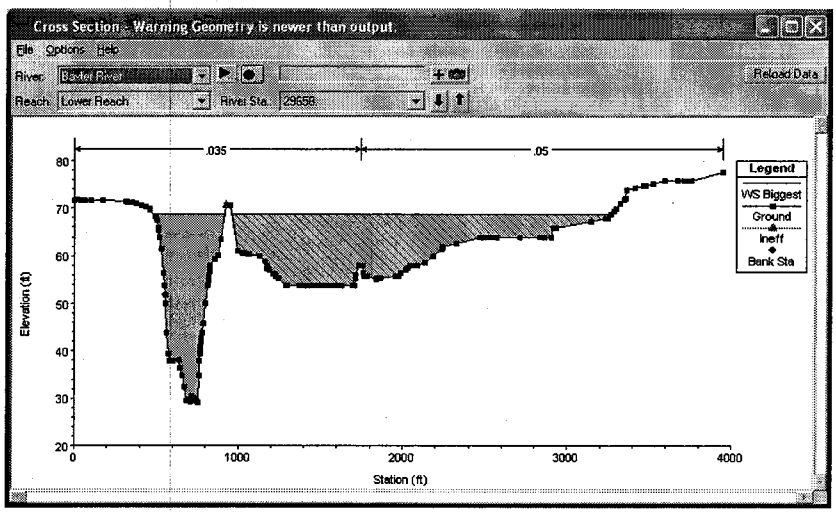
Geometry and Unsteady Flow

- What is going to happen to the water when it gets out of the channel?
  - Ineffective Flow Areas
  - Levees
  - Conveyance Calculations Using Subdivisions

37

When examining geometry, the user must consider where water may leave the main channel and how it will behave once out of the main channel. Knowing these factors gives the user a general idea of how to correctly model the situation within the framework of HEC-RAS.

Ineffective Flow Areas

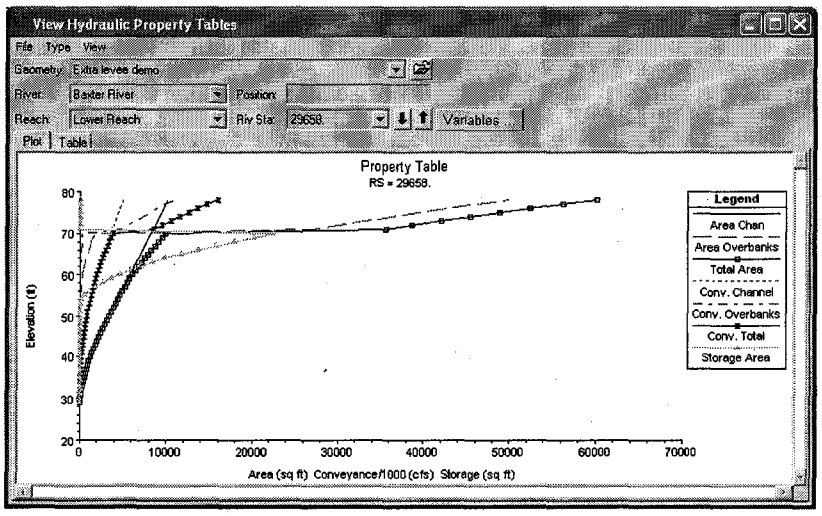


38

As seen in the figure, there is water in the right overbank. Because RAS is a one-dimensional model, the water surface elevation will be the same for the entire cross section. The user must determine if water can physically enter the right overbank at this cross section by examining upstream and downstream cross sections.

If water will enter the right overbank then the user must determine how to model this area.

Non-Permanent Ineffective Flow

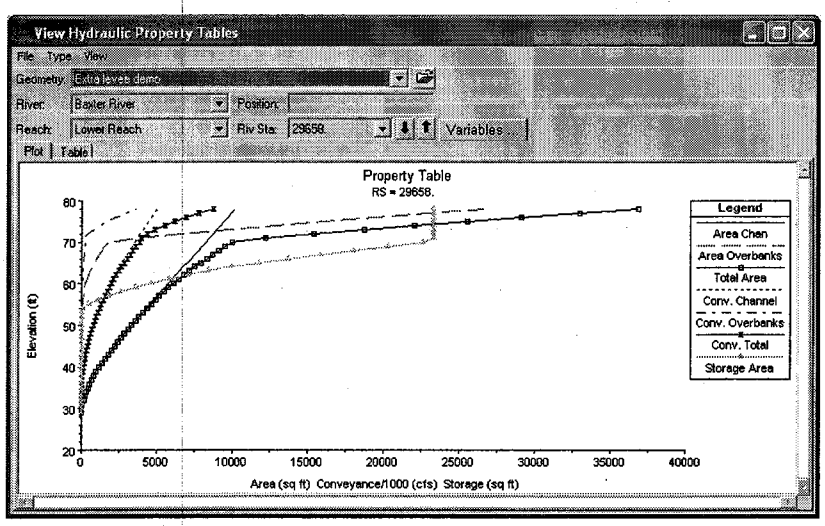


39

The figure above represents the previous cross section with the right overbank modeled as non-permanent ineffective flow. As seen on the figure, the storage in the cross section increases until reaching the trigger elevation where the ineffective flow area is turned off. By turning off the ineffective flow area the storage in the cross section returns to zero and the area of the cross section increases.

If the water level could possibly reach the trigger elevation then this is probably not the proper way to model this cross section. The change in volume of storage, and the change in area of the cross section, is too significant to model with a non-permanent ineffective flow area.

Permanent Ineffective Flow Area

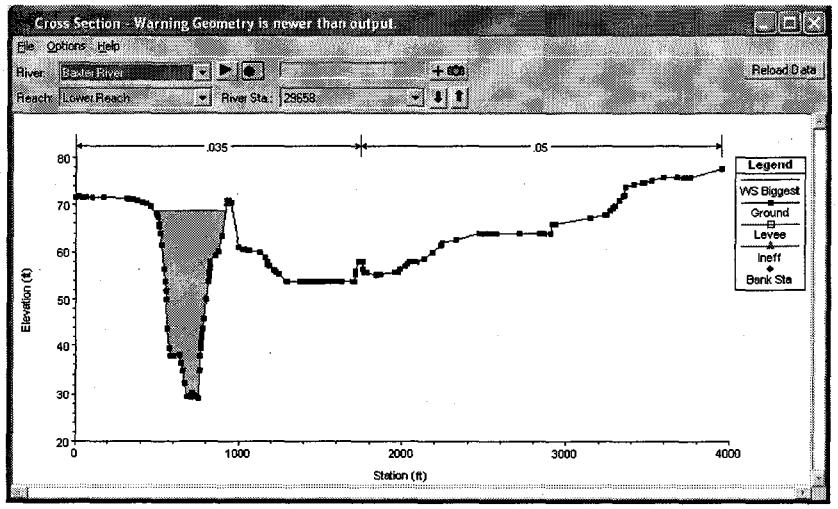


40

The figure above represents the previous cross section with the right overbank modeled as permanent ineffective flow. As seen on the figure, the storage in the cross section increases until reaching the top of the ineffective flow area where it remains constant.

If the water level could possibly reach the trigger elevation, while remaining relatively stationary, then this is probably a better representation compared to a non-permanent ineffective flow area.

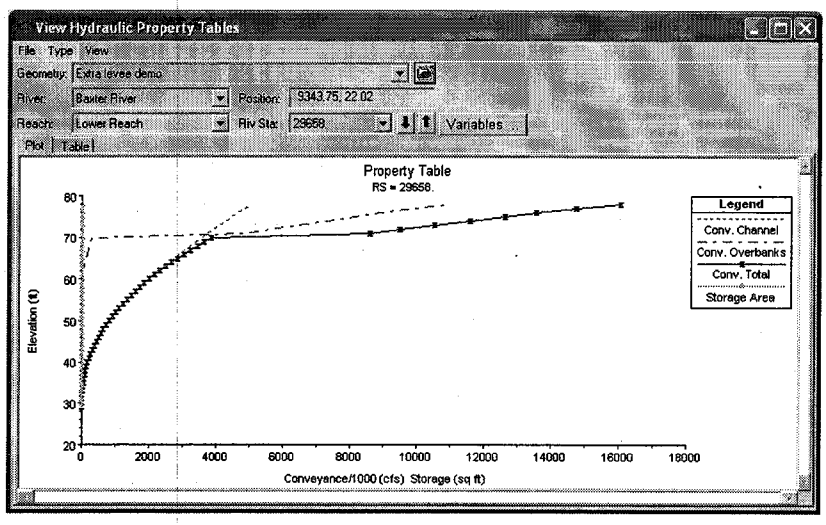
Levees



41

If it is determined that water will not enter the right overbank, then the area should be modeled with a levee. When the levee is established, no water can go to the right of the right levee station until the levee elevation is exceeded.

Levees



42

As seen on the figure, if the levee elevation is exceeded there will be an extremely large increase in conveyance with elevation for this geometry. This would not be the proper way to model the cross section if the levee is overtopped. Keep in mind that the magnitude of conveyance on the x-axis is divided by 1000.

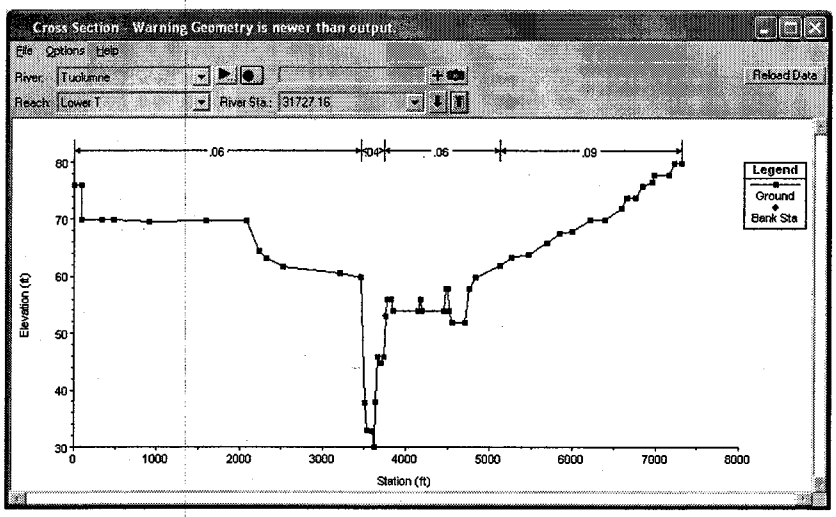
Storage Area Option

- What if adding a levee or ineffective flow area has a significant affect on the conveyance and storage relationships?
  - Alter the cross section by terminating it at the beginning of the problem area.
  - Model the area that was deleted with a lateral weir and off-stream storage area.
  - Storage areas will be discussed in impending lectures.

43

If the cross sectional geometry denotes a significant change in storage or conveyance when a levee is overtopped then this cross section should be altered. The cross section should be terminated at the levee and the area that was deleted should be modeled as a storage area. Abrupt and significant changes in storage and conveyance should be avoided because they can cause instabilities in the unsteady flow model.

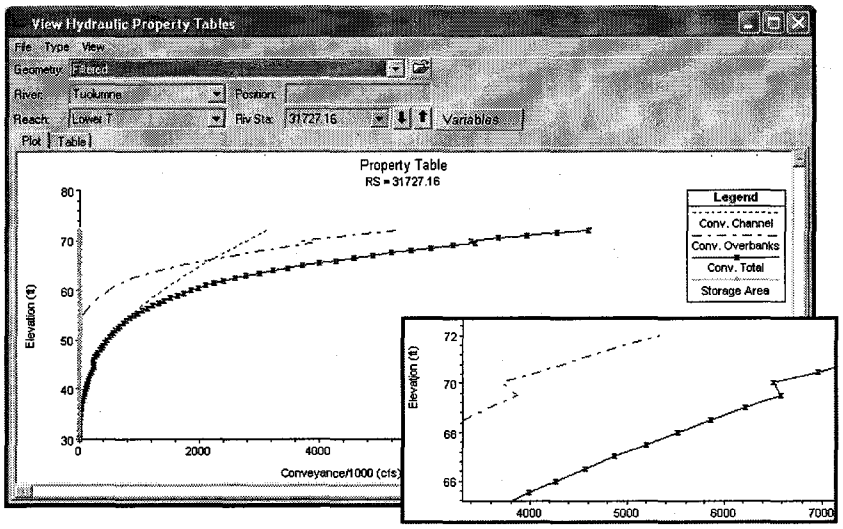
Conveyance Subdivisions



44

As shown in the figure above, there is a distinct increase in elevation located at station 2000. The entire left bank is being modeled with one Manning's n value. Remember that RAS computes the left and right overbank conveyances using subdivision of Manning's n.

Conveyance Subdivisions

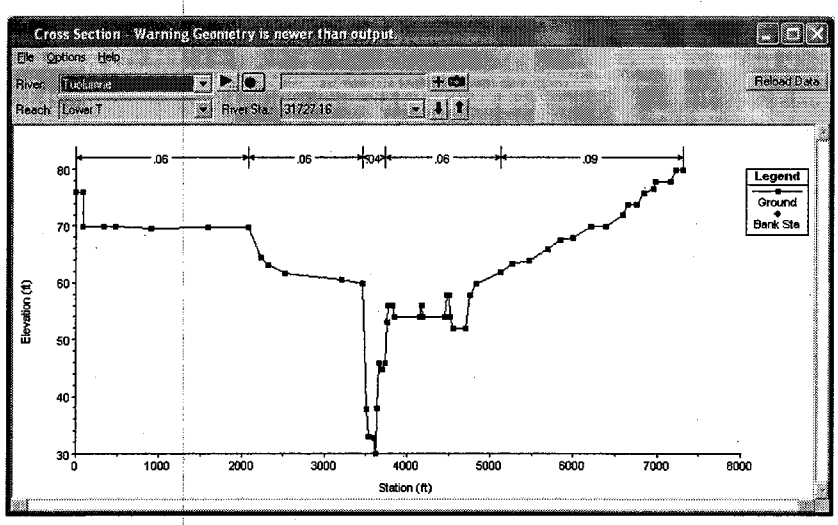


45

As seen in the figure, there is a hitch in the increase of conveyance with elevation at just above 75 feet. This corresponds to the steep increase in elevation located at station 2000 on the previous figure.

HEC-RAS uses Manning's equation to calculate conveyance. Imbedded in this equation are relationships for channel depth, channel area, and wetted perimeter. In this example, when the calculations go just above the elevation of 75 feet, Manning's equation sees an enormous increase in wetted perimeter for a minute increase in water depth. The conveyance answer found from this calculation is dramatically different from the conveyance answer found at the previous calculation below 75 feet.

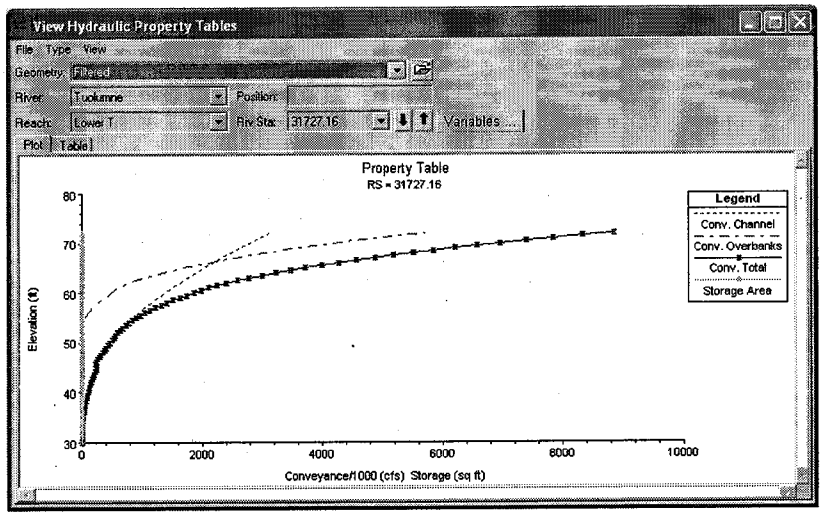
Conveyance Subdivisions



46

To account for this situation, a subdivision made by inserting another Manning's n value must be added at the point of discontinuity. The value does not need to be different, as long as the subdivision is present, RAS will calculate separate conveyances for the geometry.

Conveyance Subdivisions



47

As seen on the figure, this subdivision for the left overbank smoothed the relationship between conveyance and elevation.

Questions?

48

Workshop
Pre-Processing Geometric Data

Introduction

This workshop will help students learn how to use HEC-RAS to pre-process geometric data. Students will learn how to set cross-sectional table properties; pre-process the cross sections into tables of elevation versus conveyance, area, and storage; and review the output from the pre-processor. *While this data is from an actual study, the model and results of this workshop do not represent current or future conditions of the river.*

Background

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. The bridge crossing is located along State Highway 1049, near the middle of the river reach. However, the bridge will not be used for this workshop. The field data for this example was obtained from the United States Geological Survey (USGS) Hydrologic Atlas No. HA-601.

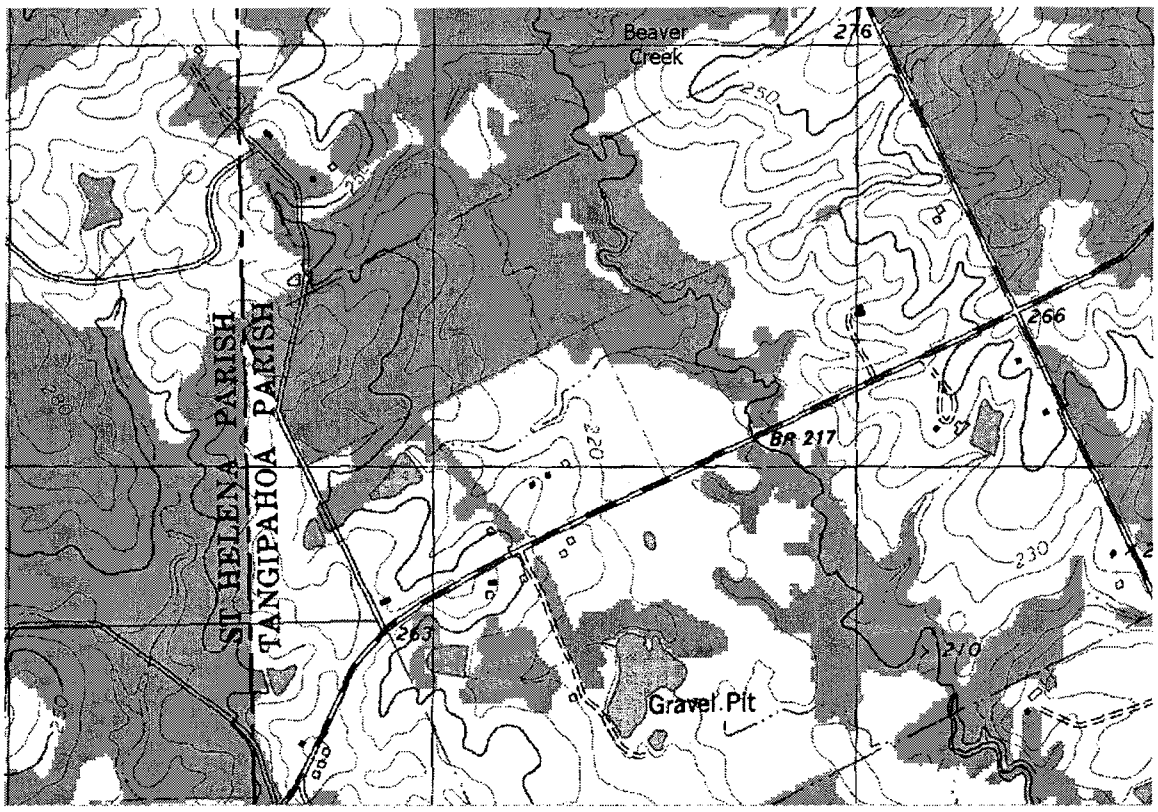


Figure 1 Beaver Creek near Kentwood, Louisiana

Problem Description

A project file (**Preprocessor.prj**) with the title "**Beaver Creek Geometric Data**" has been loaded on your workshop computers. This file contains all the data for this workshop. Only the pre-processor will be used in this workshop to examine the hydraulic property tables for cross sections.

Tasks

The following is a summary of the required tasks for each group:

1. **View HTAB Parameters and Analyze the Hydraulic Property Table**

   View the default **Cross Section Table Parameters** for this workshop in the **Geometric Data Editor** by selecting the **HTAB Parameters** button. Are the default parameters appropriate for this geometry? If not, change the parameters to appropriate values.

   Rerun the **Geometry Preprocessor** on the Unsteady Flow Analysis window.

   View the **Hydraulic Property Tables and Plots** for each cross section. Note the general relation between conveyance and elevation.

2. **Place Levees at Necessary Locations**

   Place levees at the following locations by either using the **Graphical Cross Section Editor** or the **Cross Section Editor**. (The placement doesn't need to be exact for the graphical method).

River Station	Station	Elevation
5.99	866	214.8
5.76	906	214.3
5.29	223	209.6
5.13	358	209
5.00	518	209

   Save the geometry as a new geometry file (*.g02) and then run the pre-processor (using the new geometry file). Next, view the **Hydraulic Property Tables** at each cross section where a levee was added. How did the conveyance vs. elevation relationship change because of the levee addition? Compare with the previous geometry file.

3. **Place Normal Permanent Ineffective Flow Areas**

   Add permanent ineffective flow areas at the same locations as the levees. (Place the permanent ineffective flow areas behind the levees - don't delete the levees). Save the geometry as a new geometry file (*.g03) and run the pre-processor using the new geometry file. Next, view the **Hydraulic Property Tables** at each cross section where a permanent ineffective flow area was added and compare the three geometry files. How did this change affect the conveyance vs. elevation relation compared to Task 2? What provides a better relation between conveyance and elevation, Task 2 or Task 3? Why?

Questions

1. Considering that HEC-RAS is a one-dimensional model, what problems will occur if a levee is overtopped during a simulation? What options in RAS can be used to model a levee being overtopped?

2. Are there any anomalies with the conveyance in the overbanks at River Station 5.29? If so, what is it caused by, and how can it be fixed?

3. How does a permanent ineffective area affect storage within the cross section? When should a cross section be modeled with non-permanent ineffective areas and when should it be modeled with permanent ineffective areas?

Boundary and Initial Conditions

Objectives:

- Know requirements and options for *boundary conditions*
- Know *initial condition* requirements
- Review sources of data for both

References:

HEC-RAS River Analysis System, User's Manual, Chapter 8, Performing an Unsteady Flow Analysis, November 2002.

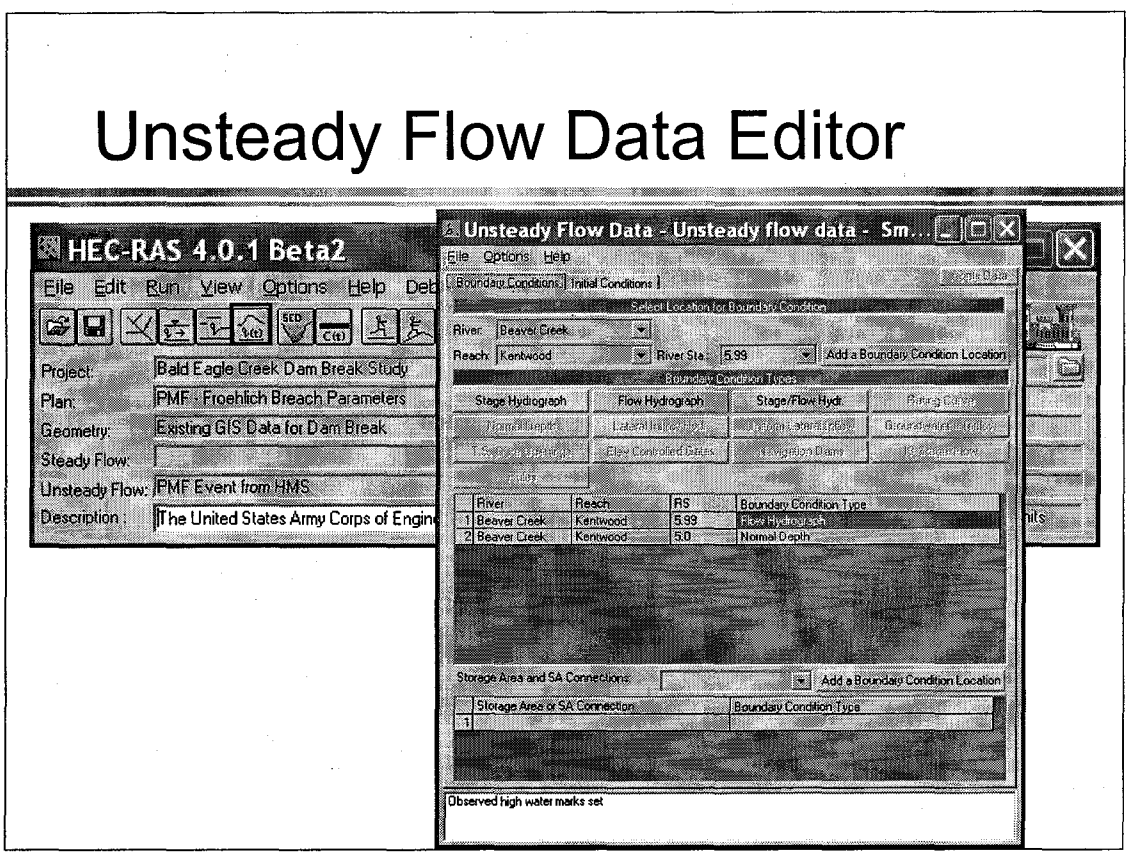
Unsteady Flow Data

- Initial Conditions – Required
  - Establish flow and stage at all nodes at the start of simulation
- External Boundaries - Required
  - Upstream and Downstream ends of the river
  - Typically flow or stage hydrograph upstream
  - Typically rating or "normal depth" downstream
- Internal Boundaries - Optional
  - Add flow within the river system
  - Define gate operation

As with steady-flow applications, the Unsteady Flow data define flow and starting conditions for the simulation. The model simulates a dynamic hydraulic process on an segment of a river system network. To perform the hydraulic calculations within the segment, the time-varying "Boundary Conditions" must be defined for the "External Boundaries."

The Unsteady Flow editor provides the options to defined:

- External boundary type and data
- Add internal boundary data
- Set the initial conditions for the start of the simulation



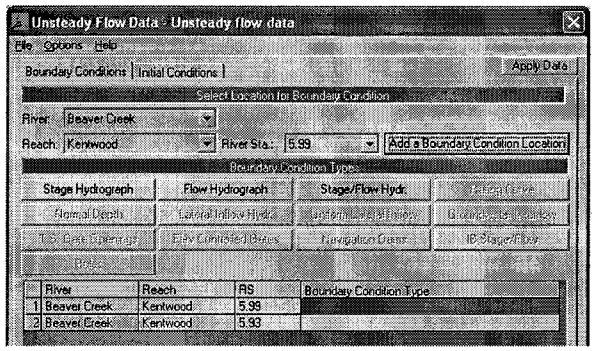
Once all of the geometric data are entered, the modeler can then enter any unsteady flow data that are required. To bring up the unsteady flow data editor, select **Unsteady Flow Data** from the **Edit** menu on the HEC-RAS main window. The Unsteady flow data editor should appear as shown above.

Unsteady Flow Data

The user is required to enter boundary conditions at all of the external boundaries of the system, as well as any desired internal locations, and set the initial flow and storage area conditions in the system at the beginning of the simulation period.

Boundary Conditions: Upstream

- Editor shows required external boundaries
- Boundary Type shows available options
- Upstream options:
  - Stage Hydrograph
  - Flow Hydrograph
  - Stage & Flow Hydrograph

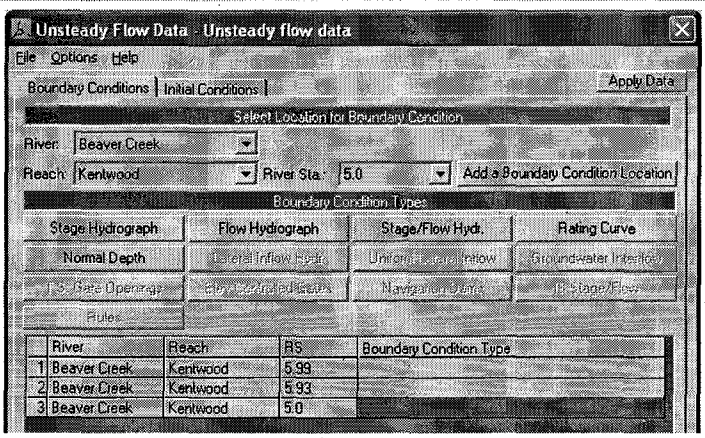


Boundary conditions are entered by first selecting the **Boundary Conditions** tab from the Unsteady Flow Data Editor. River, Reach, and River Station locations of the external bounds of the system will automatically be entered into the table.

Boundary conditions are entered by first selecting a cell in the table for a particular location, then selecting the boundary condition type that is desired at that location. Not all boundary condition types are available for use at all locations. The program will automatically gray-out the boundary condition types that are not relevant when the user highlights a particular location in the table.

Users can also add locations for entering internal boundary conditions. To add an additional boundary condition location, select the desired River, Reach, and River Station, then press the **Add a Boundary Condition Location** button.

Boundary Conditions: Downstream

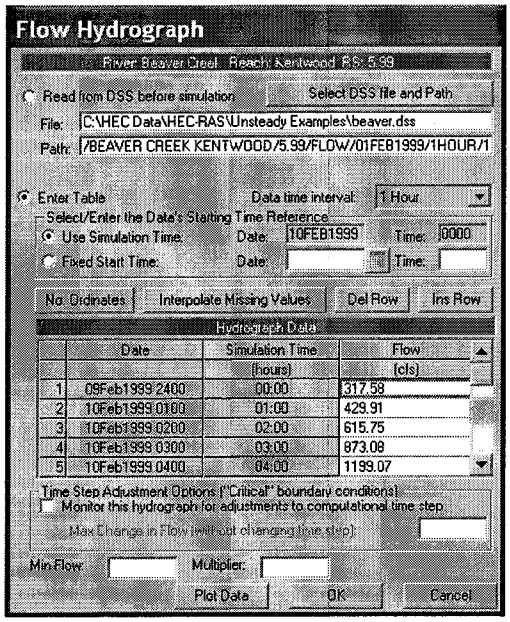


- Downstream Boundary Options:
  - Stage Hydrograph
  - Stage & Flow Hydrograph
  - Rating Curve
  - Normal Depth

The downstream boundary can be:

- Stage Hydrograph - e.g., gage data on the stream, or tidal cycle
- Flow Hydrograph - e.g., gage data converted to flow
- Stage & Flow - e.g., combined observed stage and forecasted flow
- Rating Curve - e.g., rating at a gauged location, or steady-flow rating
- Normal Depth - e.g., average slope of stream to estimate energy slope

Flow Hydrograph



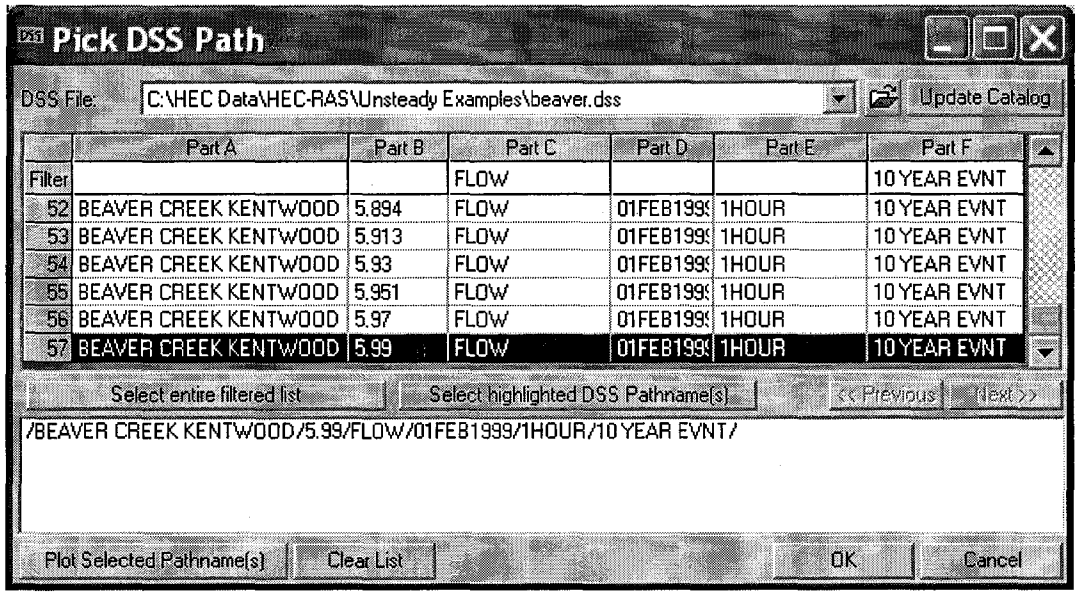
- Read from DSS
  - Select DSS file
  - Select Pathname
- Enter in Table
  - Select time interval
  - Select start date/time
  - Enter flow data - or cut & paste

Flow Hydrograph:

A flow hydrograph can be used as either an upstream boundary or downstream boundary condition, but is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown above will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table.

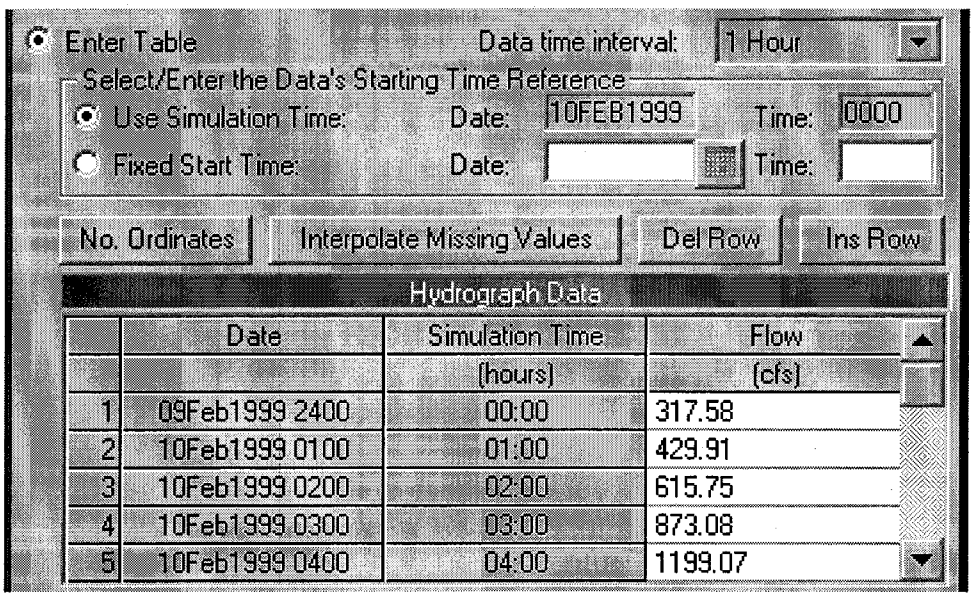
If the user selects the option to read the data from DSS, they must press the **Select DSS File and Path** button. When this button is pressed a DSS file and pathname selection screen will appear as shown on the next page. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table. The user can use the pathname filters to reduce the number of pathnames shown in the table. The last step is to select the desired DSS Pathname and to close the window.

Select DSS file and Path



When the **Select DSS File and Path** button is pressed a DSS file and pathname selection screen will appear as shown above. First select the desired DSS file by using the browser button at the top. Once a DSS file is selected, a list of all of the DSS pathnames within that file will show up in the table. The user can use the pathname filters to reduce the number of pathnames shown in the table. The last step is to select the desired DSS Pathname and to close the window.

Enter Flow Data

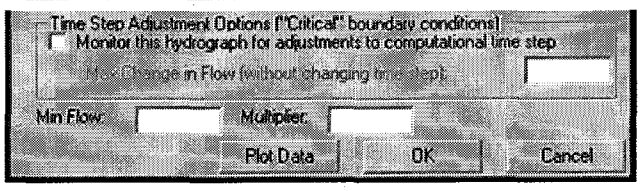


The user also has the option of entering a flow hydrograph directly into a table, as shown above. The first step is to enter a **Data time interval**. Currently the program only supports regular interval time series data. A list of allowable time intervals is shown in the drop down window of the data interval list box. To enter data into the table, the user is required to select either **Use Simulation Time** or **Fixed Start Time**.

If the user selects **Use Simulation Time**, then the entered hydrograph will always start at the beginning of the simulation time window. The simulation starting date and time is shown next to this box, but is grayed out.

If the user selects **Fixed Start Time** then the hydrograph is entered starting at a user specified time and date. Once a starting date and time is selected, the user can then begin entering the data.

Monitor Flow Rate-of-change



- Added option for Flow Hydrograph data
  - Set Maximum Flow Change in time step
  - If input data exceeds max, program will shorten computational time step
- Min Flow
- Multiplier

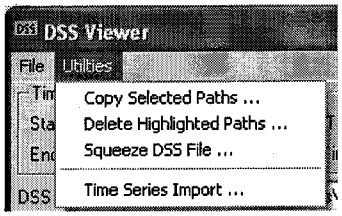
An additional option listed on the flow hydrograph boundary condition is to "**Monitor this hydrograph for adjustments to computational time step**." When you select this option, the program will monitor the inflow hydrograph to see if a change in flow rate from one time step to the next is exceeded. If the change in flow rate does exceed the user entered maximum, the program will automatically cut the time step in half until the change in flow rate does not exceed the user specified max. Large changes in flow can cause instabilities. The use of this feature can help to keep the solution of the program stable.

Also, the flow hydrograph for model boundaries can be monitored to ensure that the flow values do not drop below an input minimum value. If the value is less than the minimum, the value is set to the minimum.

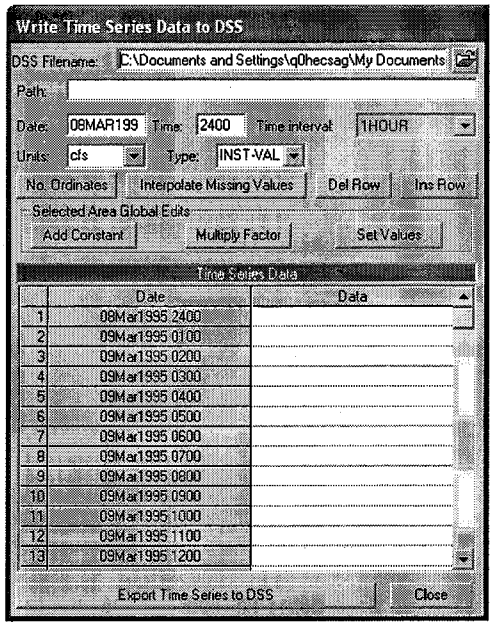
Additionally, the flow hydrograph for model boundaries can be easily modified by a constant ratio. This allows easy testing of model sensitivity to input flow values, or an analysis of model results with larger flood events.

These features appear on most flow input editors.

Enter Data to a DSS file



- Pathname
- Start date & time
- Time interval
- Units
- Data type
- Enter data, or
- Copy / Paste



Entering time-series data into a DSS file is an alternative to entering time-series data directly into the simulation. By entering the data into the file, it is available for use in other simulations. While DSS has DOS programs to accomplish data entry, this is the easiest Windows method at this time.

The function is a DSS Viewer Utility. The Write Time Series Data to DSS window provides for data entry of all the required pathname and header information. Then the data can be entered directly into the table. Or, if the data are available in a spreadsheet or similar table, the data can be copied and pasted into the input table.

Sources of Time-Series Data

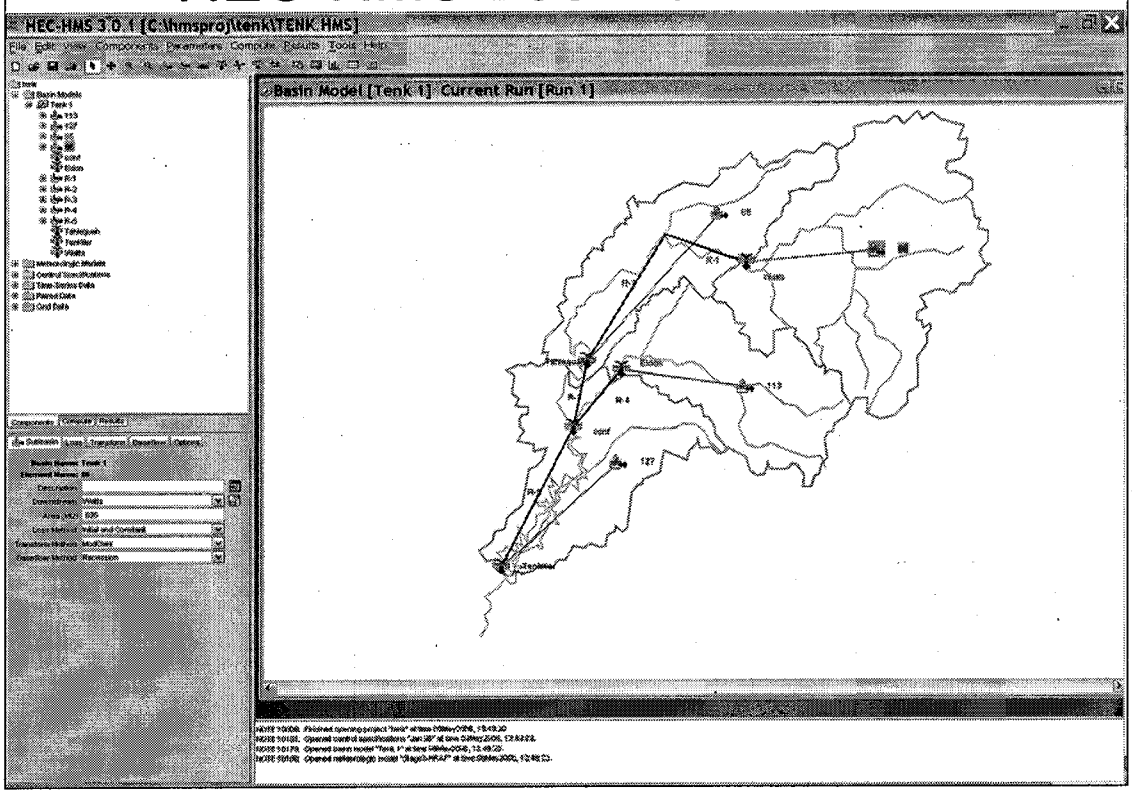
- Historic Records (USGS)
  - Stage Hydrographs
  - Flow Hydrographs
- Computed Synthetic Floods
- Peak Discharge with assumed time distribution
- Rainfall-runoff modeling

Unsteady-flow modeling provides a dynamic solution for stage and flow throughout the river network. The primary boundary data are time-series flow, typically for a flood event. Where do you get that flow data?

For most program user's, the simulation will route an upstream flow hydrograph, using a downstream rating curve or normal depth. The upstream hydrograph can be:

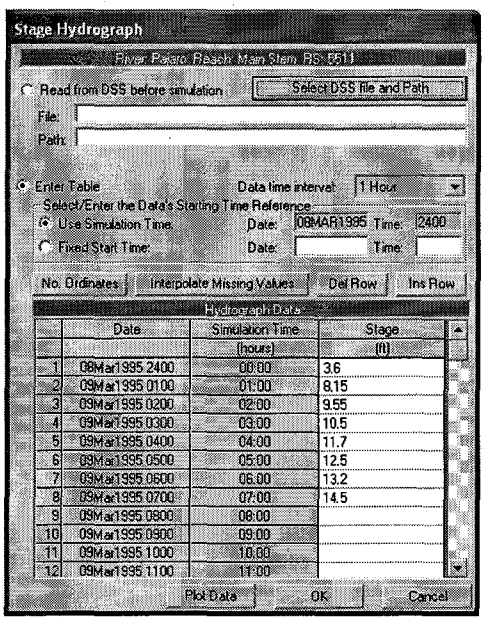
- Historic, like those obtained from the US Geologic Service.
- Synthetic, often, prior studies have computed and published synthetic floods like the Corps' Standard Project (SPF) or the Probable Maximum Flood (PMF).
- Modeling, using rainfall-runoff models like the Corps' Hydrologic Modeling System (HEC-HMS).
- Peak Discharge, and a time to peak estimate, can be used to provide a crude hydrograph using the HEC-RAS interpolation routine (Triangle hydrograph). If a Unit Hydrograph is available, that distribution can be used to develop a flow hydrograph.

HEC-HMS Basin Schematic



The HEC-HMS program can be used to model the rainfall-runoff process in the contributing watershed (catchment). Even when there are some gauged data in the watershed, the hydraulic model may require lateral inflow data and tributary flows where no gauged data exists. A basin model can be developed to provide the necessary data at all model locations. By careful coordination of study requirements and key information locations, the model nodes can be identified and modeled consistently among various computer programs. The data can be conveniently transferred through different models using the HEC-DSS file system.

Stage Hydrograph

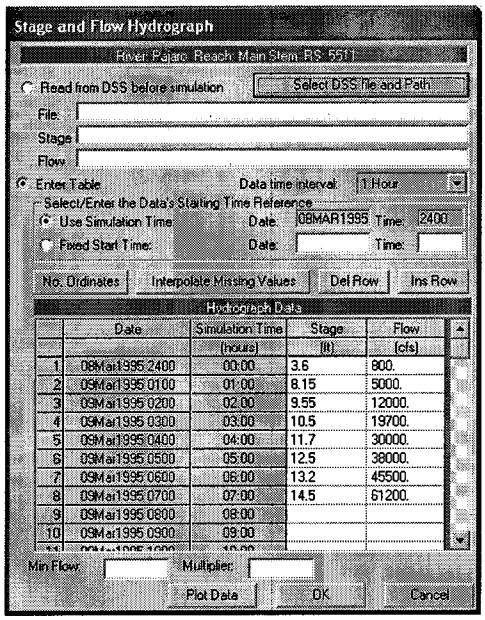


- Read from DSS
  - Select DSS file
  - Select Pathname
- Enter in Table
  - Select time interval
  - Select start date/time
    - Simulation time
    - Fixed starting time
  - Enter stage data - or cut & paste

Stage Hydrograph:

A stage hydrograph can be used as either an upstream or downstream boundary condition. The editor for a stage hydrograph is similar to the flow hydrograph editor shown above. The user has the choice of either attaching a HEC-DSS file and pathname or entering the data directly into a table.

Stage and Flow Hydrograph

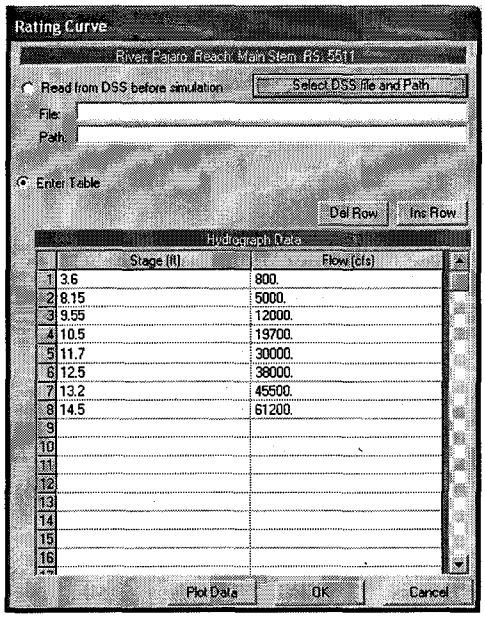


- Upstream Mixed
  - Stage used while defined
  - Flow used after
- Purpose:
  - Real-time use
  - Mix observed stage with
  - Forecasted flow

Stage and Flow Hydrograph:

The stage and flow hydrograph option can be used together as either an upstream or downstream boundary condition. <u>The upstream stage and flow hydrograph is a mixed boundary condition</u> where the stage hydrograph is inserted as the upstream boundary until the stage hydrograph runs out of data; at this point the program automatically switches to using the flow hydrograph as the boundary condition. The end of the stage data is identified by the HEC-DSS missing data code of "-901.0". This type of boundary condition is primarily used for forecast models where the stage is observed data up to the time of forecast, and the flow data is a forecasted hydrograph.

Rating Curve



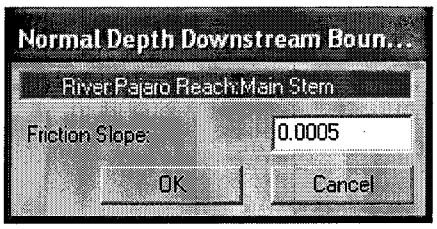
- Single value rating
- Downstream boundary
- Read from DSS
  - Select DSS file
  - Select Pathname
- Enter in Table
  - Enter stage- flow data - or
  - Cut & paste

Rating Curve:

The rating curve option can be used as a downstream boundary condition. The user can either read the rating curve from HEC-DSS or enter it by hand into the editor.

The downstream rating curve is a single valued relationship, and does not reflect a loop in the rating, which may occur during an event. This assumption may cause errors in the vicinity of the rating curve. The errors become a problem for streams with mild gradients where the slope of the water surface is not steep enough to dampen the errors over a relatively short distance.. When using a rating curve, make sure that the rating curve is a sufficient distance downstream of the study area, such that any errors introduced by the rating curve do not affect the study reach.

Normal Depth



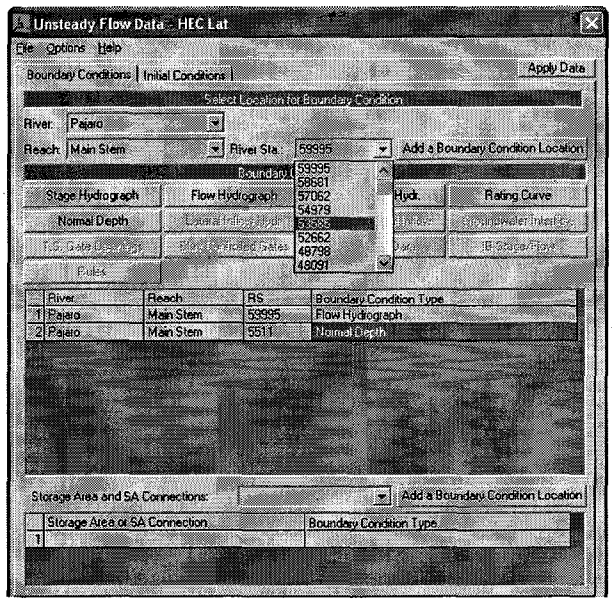
- Enter Friction (energy) Slope
- Program uses Manning's equation to compute stage
- Provides "Normal Depth" downstream boundary

Normal Depth:

The Normal Depth option can only be used as a downstream boundary condition for an open-ended reach. This option uses Manning's equation to estimate a stage for each computed flow. To use this method the user is required to enter a friction slope for the reach in the vicinity of the boundary condition. The slope of the water surface is often a good estimate of the friction slope.

As recommended with the rating curve option, when applying this type of boundary condition you should place it far enough downstream of the study reach, such that any errors it produces will not affect the results at the study reach.

Adding an Internal Boundary

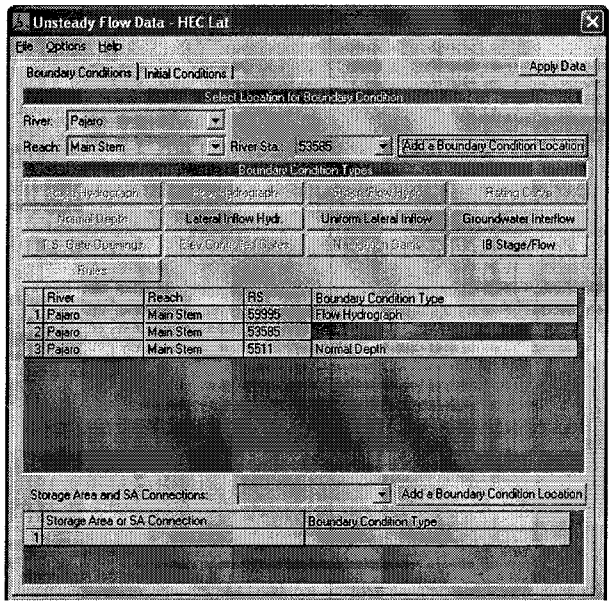


- Select:
  - River
  - Reach
  - River Station
- Add a Boundary Condition Location
- Location added to the list of Boundary Conditions

The Unsteady Flow Data editor automatically has the external boundaries defined, based on the geometric data model. The user can add internal boundary locations to define additional lateral flow. Also, when weirs with gates are added, the gate operation will be defined as an internal boundary.

Select the River, Reach and River Station where the data will be added. Then press **Add a Boundary Condition Location**. The location is added to the boundary condition table.

Internal Boundary Conditions



- Lateral Inflow Hydrograph
- Uniform Lateral Inflow
- Groundwater Interflow
- Also:
  - Gate openings
  - Navigation Dams
  - Internal Stage/Flow

Lateral Inflow Hydrograph: This internal boundary condition allows the user to bring in flow at a specific point along the stream.

Uniform Lateral Inflow Hydrograph: This internal boundary condition allows the user to bring in a flow hydrograph and distribute it uniformly along the river reach between two specified cross sections.

Groundwater Interflow: This option allows the user to identify a reach that will exchange water with a groundwater reservoir.

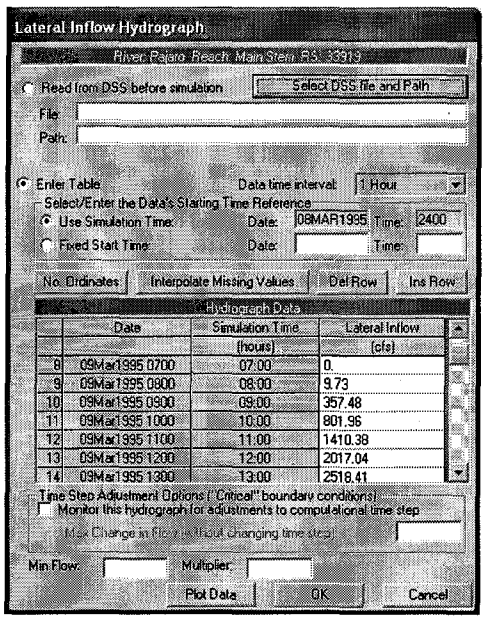
<u>THE FOLLOWING WILL BE PRESENTED IN THE GATE LECTURE</u>

Time Series of Gate Openings. This option allows the user to enter a time series of gate openings for an inline gated spillway, lateral gated spillway, or a spillway connecting two storage areas.

Elevation Controlled Gate. Define the opening and closing of gates based on the elevation of the water surface elevation upstream from the structure.

Navigation Dams: Simulate the gate operations for navigation dams attempting to maintain navigation depths.

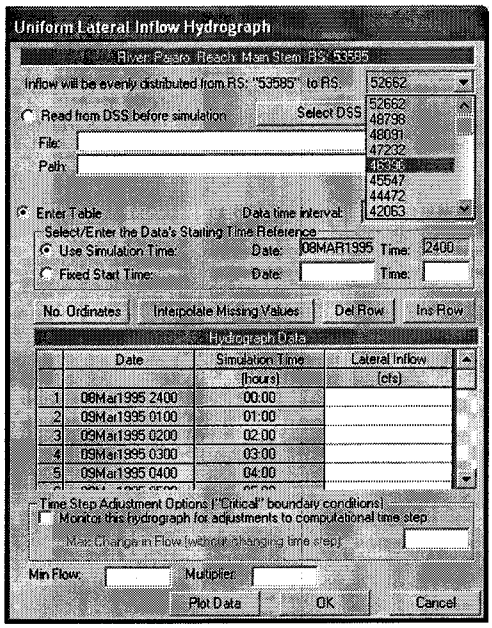
Lateral Inflow Hydrograph



- Lateral flow downstream from a Cross Section
- Read from DSS
  - Select DSS file
  - Select Pathname
- Enter in Table
  - Select time interval
  - Select start date/time
  - Enter flow data - or cut & paste

Lateral Inflow Hydrograph: This internal boundary condition allows the user to bring in flow at a specific point along the stream. The user attaches this boundary condition to the river station of the cross section just upstream of where the lateral inflow will come in. The actual change in flow will not show up until the next cross section downstream from this inflow hydrograph. The user can either read the hydrograph from DSS or enter it by hand.

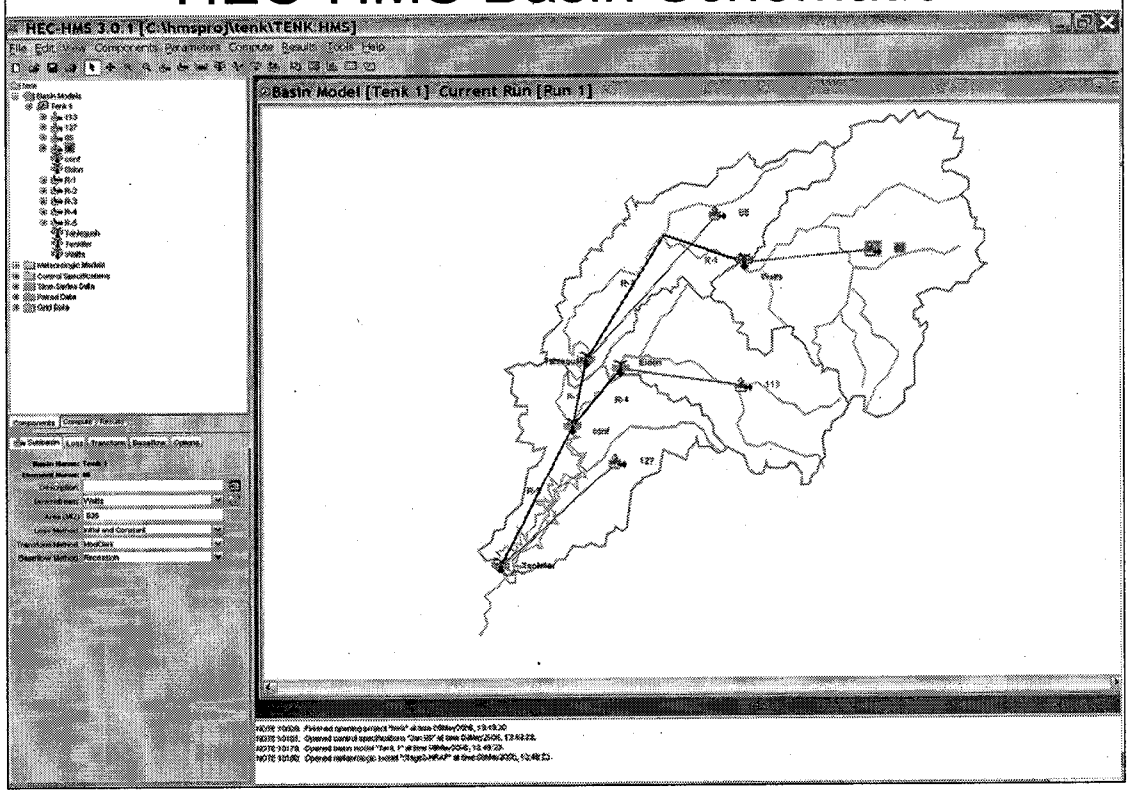
Uniform Lateral Flow Hydrograph



- Uniform Lateral flow to a reach element
- Add Internal Location at Upstream River RS
- Set Downstream RS
- Read from DSS
- Enter in Table

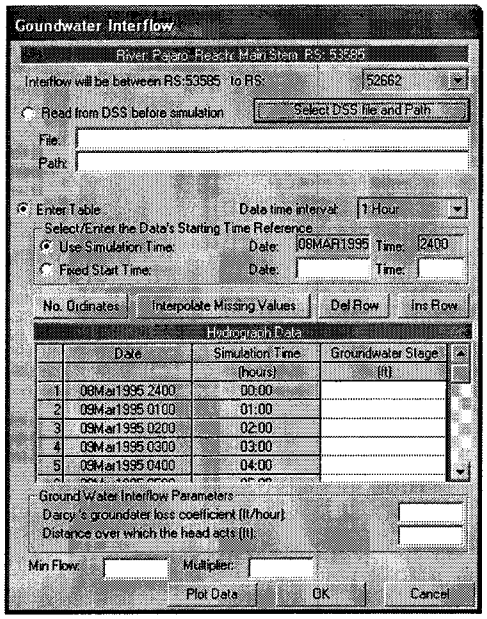
Uniform Lateral Inflow Hydrograph: This internal boundary condition allows the user to bring in a flow hydrograph and distribute it uniformly along the river reach between two specified cross sections. The hydrograph for this boundary condition type can be either read from DSS, or entered into a table.

HEC-HMS Basin Schematic



HMS provides individual watershed flows locations. If a reach flows through a watershed the hydrograph can be distributed along the reach with the distributed flow boundary condition.

Groundwater Interflow



- Add Internal Location at Upstream River RS
- Set Downstream RS
- Input Groundwater Stage
  - Read from DSS
  - Enter in Table
- Darcy's Loss Coefficient
- Distance between stages of river & groundwater

Groundwater Interflow: This option allows the user to identify a reach that will exchange water with a groundwater reservoir. The stage of the groundwater reservoir is assumed to be independent of the interflow from the river, and must be entered manually or read from DSS. The groundwater interflow is similar to a uniform lateral inflow in that the user enters an upstream and a downstream river station, in which the flow passes back and forth. The computed flow is proportional to the head between the river and the groundwater reservoir. The computation of the interflow is based on Darcy's equation. The user is required to enter Darcy's groundwater loss coefficient (hydraulic conductivity), as well as a time series of stages for the groundwater aquifer.

The groundwater algorithm is simplistic; simulating flow in only one direction, laterally, perpendicular to the river. The groundwater aquifer is assumed to be very large such the the interchange of water with the river has no impact on the groundwater level.

Darcy's Law

V = K S

V = Velocity (ft/hour)

K = Hydraulic Conductivity (ft/hour)

S = Slope (ft/ft) = Stage difference / Distance

Q = V A

Q = Flow (cfs)

A = Area, perpendicular to the flow (ft^2)

- Top width x length, if groundwater below channel
- Valley width x length plus channel depth x length, if groundwater is above channel invert

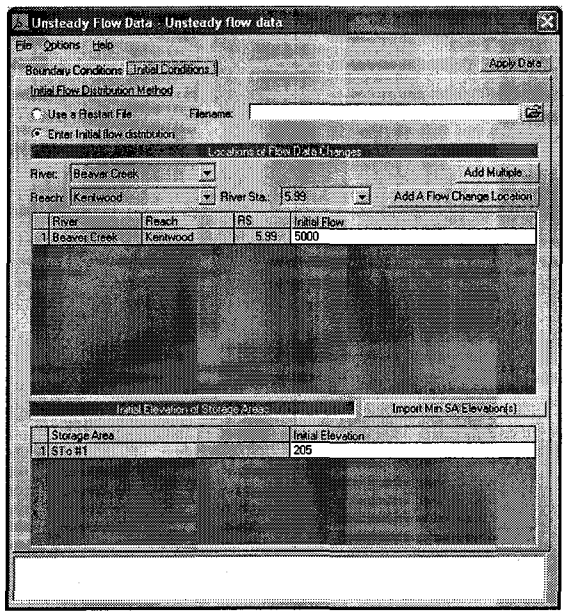
Darcy's Law (1856) applied the principles of fluid flow to the flow of water in a permeable media. The velocity of flow equals the product of a coefficient times the hydraulic gradient. The flow equals the velocity times the effective area = gross area times porosity of the media. (*Hydrology for Engineers, by LKP*)

- In RAS, the slope is computed by the difference between the water elevation at the cross section and the input groundwater elevation, divided by the input Distance.
- The area is length times the flow width.
- The length is the channel length for the reach element (between upstream and downstream River Stations).
- The flow width depends on the groundwater level.

  If the groundwater level is below the invert of the channel, the flow is down from the channel and the width is the average top width.

  If the groundwater level is above the invert, the width is the overbank width plus the channel depth, estimated by the average channel water elevation minus the groundwater elevation.

Initial Conditions



- Requires an initial flow for all reaches
- Enter steady-flow at upstream boundary
- Can add a flow-change location
- Pool elevation for storage areas
- Use system status from previous simulation (restart)

Initial Conditions In addition to the boundary conditions, the user must establish the initial conditions of the system at the beginning of the unsteady flow simulation. Initial conditions consist of flow and stage information at each of the cross sections, as well as elevations for any storage areas defined in the system. Select the **Initial Conditions** tab to bring up the data editor, shown above. There are two options for establishing the initial conditions.

The first option is to enter flow data for each reach and have the program perform a steady flow backwater run to compute the corresponding stages at each cross section. This option also requires the user to enter a starting elevation for any storage areas in the system. This is the most common method for establishing initial conditions.

A second method is to read a file of stages and flows written from a previous run, which is called a "Restart File". This is often used when running a long simulation time that must be divided into shorter periods. The output from the first period is used as the initial conditions for the next period, and so on. However, the RAS model has a dynamic allocation procedure to overcome the time-dimension limit.

Initial Conditions Data

- Initial Flow data
  - Should be nearly steady-state conditions
  - Starting flow from upstream hydrograph
  - Flow distribution from Steady Flow profile
- Initial Elevation in Storage areas
  - Stage from historic data
  - Initial early in event, dry conditions
- Hot Start file from previous simulation

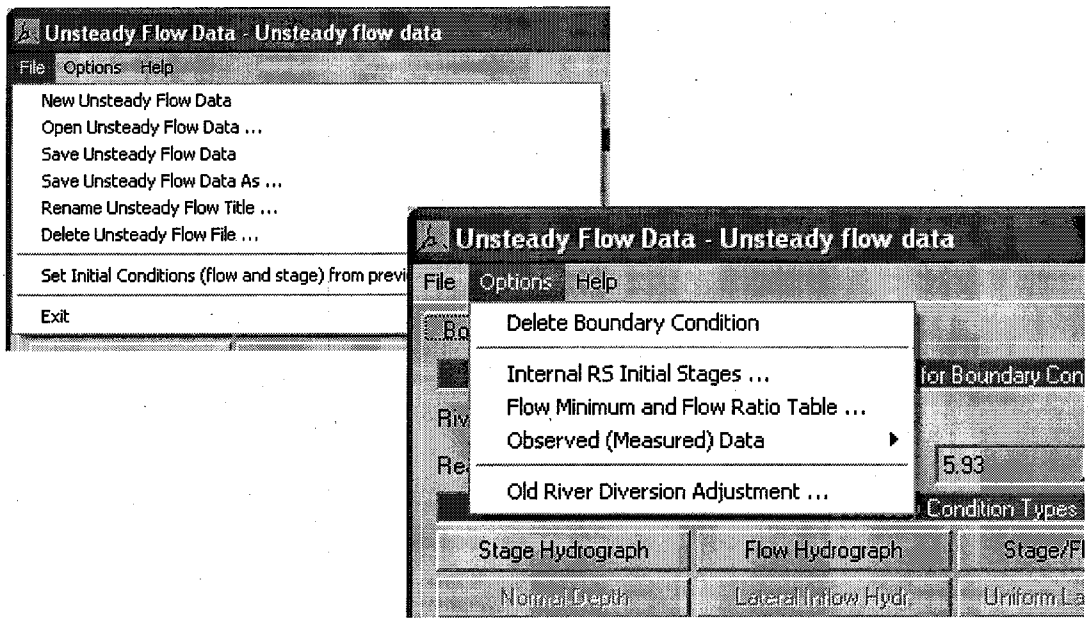
In most applications, the initial condition is a steady-flow water surface profile throughout the river system. If a constant flow is defined for each river reach, that flow should represent a condition when flows are fairly stable, i.e., a period prior to the start of the flood. The initial value should be the starting flow in the inflow hydrograph. You do not want a large change in flow between the initial flow and the first inflow value.

The flows in the river system should be consistent. For a complex river network, resolve the flow distribution using steady-flow analysis. The new option for optimizing the flow distribution could be used to help develop a consistent flow set.

If flow is not steady at the start of simulation, you can use hydrologic routing to estimate the flows through the river system. Flow changes can be input in the initial condition, just like steady flow data. An HMS model could be used for this purpose. Remember, the initial flow in the hydrologic model is steady flow.

The Hot Start file can be saved from a simulation. Additionally, this option may be used when the software is having stability problems at the beginning of a run. Run the model with all the inflow hydrographs set to a constant flow, and set the downstream boundaries to a high tailwater condition. Run the model with decreasing tailwater down to a normal stage over time. Once the tailwater is a reasonable value for the constant flow, those conditions can be written out to a file, and then used as the starting conditions.

File and Options Menus

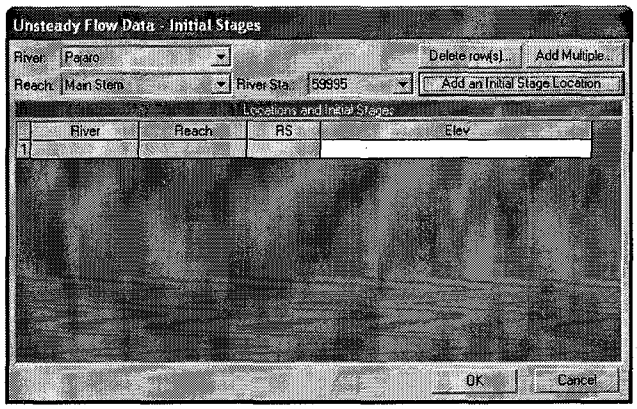


The Unsteady Flow Window has the standard **File Menu** options. It is a good idea to save the data before going on to the next step.

The **Options Menu** has four options:

- Delete an added Boundary Condition. You cannot delete external boundaries because they are required.
- Internal RS Initial Stages to set a stage at the start of simulation.
- Flow Minimum and Flow Ratio Table
- Add Observed Data - this option is helpful when calibrating model to simulate an historic flood event.

Internal RS Initial Stages

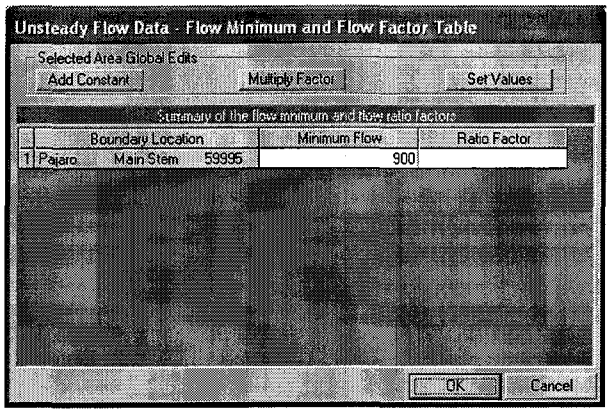


- Select:
  - River
  - Reach
  - River Station
- Add Location to table below
- Define Initial Elevation

The initial stage can be set for a river station. Normally, this would be defined by the computed water surface profile during the initial conditions stage of the computation process. This option is useful at locations upstream from a control structure where the initial stage is NOT the 'normal' backwater elevation.

Select the River, Reach and River Station where the initial stage is to be defined. Then 'click' the **Add an Initial Stage Location** button. The location will be added to the table below. Then you can enter the initial stage for that location.

Flow Minimum & Flow Ratio Table



- Global Editing to:
  - Add constant,
  - Multiply values, or
  - Set Values
- Minimum Flow value for boundary
- Multiplying factor to modify hydrograph for boundary

The boundary hydrographs can be modified with global editing to:

Add a constant value to the input

Multiply input values by a factor

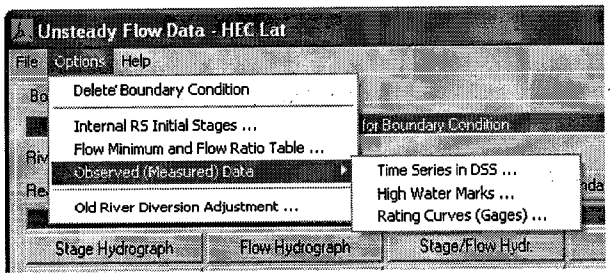
Set values to an input value

The following two options are also available on all flow hydrograph input windows.

1. The flow hydrograph for model boundaries can be monitored to ensure that the flow values do not drop below an input minimum value. If the value is less than the minimum, the value is set to the minimum.

2. The flow hydrograph for model boundaries can be easily modified by a constant ratio. This allows easy testing of model sensitivity to input flow values, or an analysis of model results with larger flood events.

Observed (Measured) Data

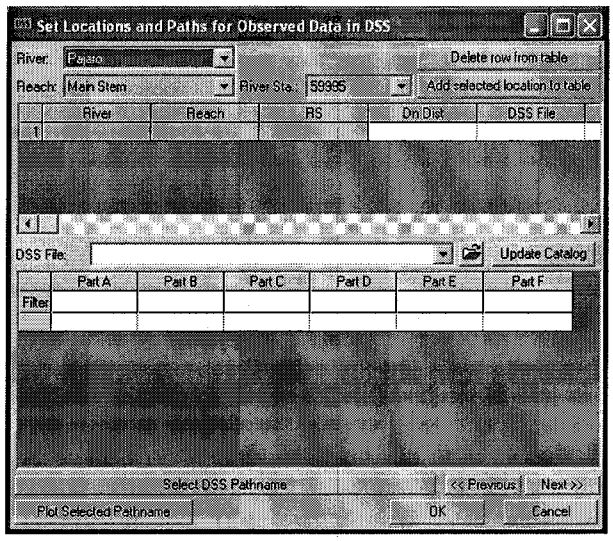


- Time Series in DSS
- High Water Marks
- Rating Curve (Gage)

There are 3 options available to enter observed data:

1. Time-series data can be entered to define observed stage or flow data. There data must be in a DSS file.
2. High-water marks can be defined
3. A Rating Curve can be defined

Observed Time Series Data

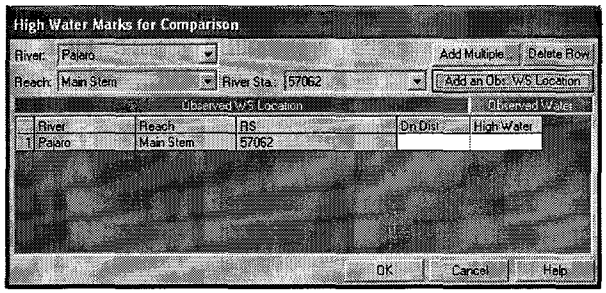


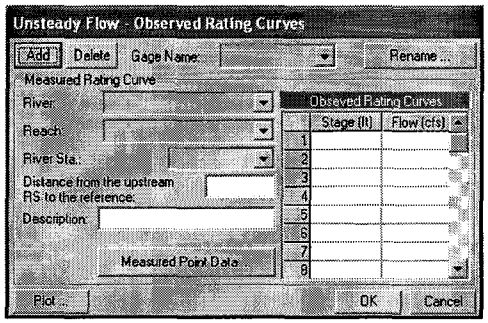
- Add Selected location
  - River
  - Reach
  - River Station
- Select DSS File
- Select Pathname to read Observed Data

Observed time-series data can be added to the unsteady flow data set. First select the model location where the observed data would apply. Select River, Reach, and River Station and press **Add selected location**. The location will appear in the table below. Several locations can be selected.

The second step is the selection of DSS records to read the observed data. The lower half of the window has the standard DSS file and record selection features. Filters can be selected for pathname parts to facilitate record selection. A DSS Record should be selected for each Location selected. Remember the time window set for the simulation will be used to extract the observed data, so be sure there is observed data for the time window.

High Water & Rating Curves





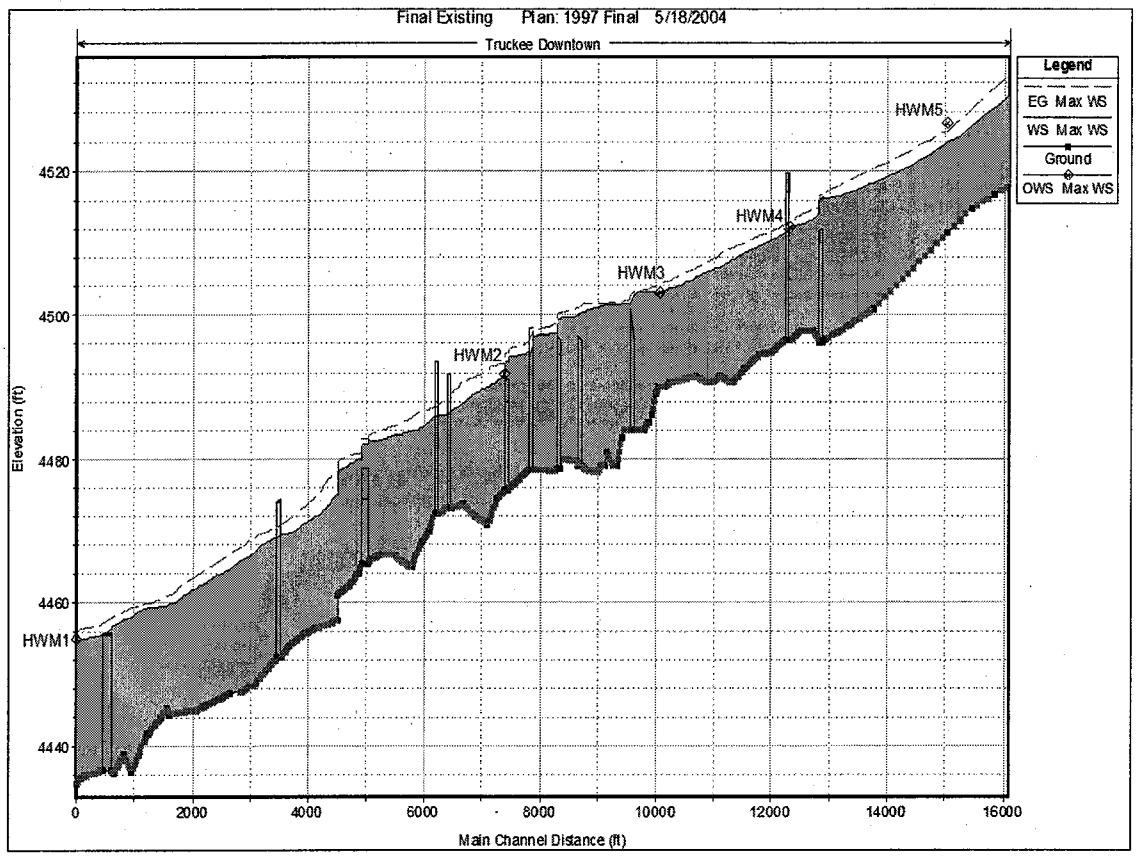
- High Water
  - Select location
  - Add an Obs. WS
  - Define Distance downstream & stage
- Observed Rating
  - Select location
  - Distance downstream
  - Stage – Flow data

High Water Marks can be defined for locations along the river reach.

First the nearest upstream cross section is selected and added to the Observed Water Surface Locations. The distance downstream from the section and the observed stage are then entered in the table.

Observed Rating provides for a Rating Curve or Measured Flow Data to be entered.

1. The Rating curve is defined in typical Stage – Flow relationship, starting from the lowest stage and sequentially increasing.
2. The Measured Point Data is entered by pressing that button and entering the observed stage and flow, along with optional description. These data do not have to define a complete rating.



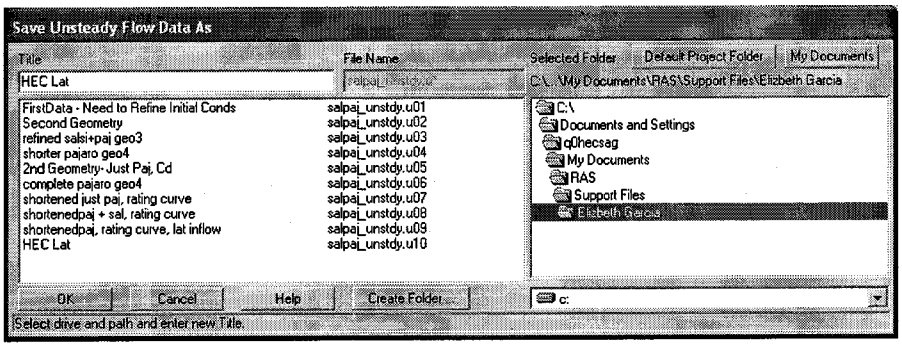
High Water Marks can be defined for locations along the river reach.

First the nearest upstream cross section is selected and added to the Observed Water Surface Locations. The distance downstream from the section and the observed stage are then entered in the table.

Observed Rating provides for a Rating Curve or Measured Flow Data to be entered.

1. The Rating curve is defined in typical Stage – Flow relationship, starting from the lowest stage and sequentially increasing.
2. The Measured Point Data is entered by pressing that button and entering the observed stage and flow, along with optional description. These data do not have to define a complete rating.

File Unsteady Flow Data



- Save Unsteady Flow Data
- Provide Descriptive Title
- Ready for Unsteady Simulation

After entering all required data, save the file to permanently write the data to disk. Once complete, the next step is to set up an unsteady flow simulation. As with steady-flow simulation, a Plan is established using a Geometry and Unsteady Flow file.

WORKSHOP 2:

Basic Unsteady Flow Modeling Using HEC-RAS

Problem Description

Beaver Creek data will be used for this workshop. The data comes from the USGS bridge data developed for the hydraulic analysis of bridges. The model data are part of the steady-flow data distributed with HECRAS.

In this problem, the geometric data processed in Workshop 1 will be used to route a flood event. Unsteady flow data will be defined, a simulation performed and output will be reviewed.

Workshop Tasks:

1. Open Wrk2.prj, which represents the data developed in Workshop 1.

2. Open the Geometry file. Check the HTAB settings and adjust them, if necessary based on your Workshop 1 analyses. Save the geometry file.

3. Open the Unsteady Flow data editor and enter the required data for processing.

   a. Upstream Boundary is a flow hydrograph. The data for the flood is in an excel file, and is listed in Hydrograph Table, on the next page.

   b. Downstream Boundary is a rating curve. The data are listed in the Rating Table, on the next page.

   c. Initial Flow should start with the first discharge value from the flood hydrograph.

Beaver Creek Flood Hydrograph

Date	Time	Flow
21-May-74	2400	500.00
22-May-74	0100	730.00
	0200	1404.33
	0300	2477.03
	0400	3875.00
	0500	5502.97
	0600	7250.00
	0700	8997.03
	0800	10625.00
	0900	12022.97
	1000	13095.67
	1100	13770.00
	1200	14000.00
	1300	13770.00
	1400	13095.67
	1500	12022.97
	1600	10625.00
	1700	8997.03
	1800	7250.00
	1900	5502.97
	2000	3875.00
	2100	2477.03
	2200	1404.33
	2300	730.00
	2400	500.00

Section 5. Rating Curve

Elevation	Flow
204.14	200
206.09	500
207.26	1000
208.47	2000
209.30	3000
209.83	4000
210.25	5000
210.68	6000
210.99	7000
211.27	8000
211.53	9000
211.78	10000
212.02	11000
212.25	12000
212.47	13000
212.68	14000
213.00	16000
214.00	21000
215.00	26000
216.00	30000

4. Set up the Unsteady Flow Analysis. Establish a new Plan, check Programs to Run, and set the times. What Computational Interval would be appropriate? Don't forget to select the cross sections for output.

5. Evaluate the results.

   a. Are there model results that suggest problems? What could be done to improve the solution?

   b. If time permits, make adjustments and run again.

   c. What was the attenuation in peak discharge?

6. Add a lateral flow to River Station 5.29. The inflow starts at 25 cfs, peaks at 250 cfs at noon and subsides to 25 cfs at midnight. Save the Unsteady Flow file as a new name and run the simulation as a New Plan. How much and how far did the inflow change the peak profile?

Modeling Bridge and Culverts

Cameron Ackerman, P.E.

1

Objectives: Review bridge & culvert input, Know added input for processing bridge and culvert data, Know how to review processed data

References:

HEC-RAS River Analysis System, *User's Manual*, Chapter 6, January 2001.

HEC-RAS River Analysis System, *Hydraulic Reference Manual*, January 2001.

Chapter 5 Modeling Bridges

Chapter 6 Modeling Culverts

Appendix B on Flow Transitions, and

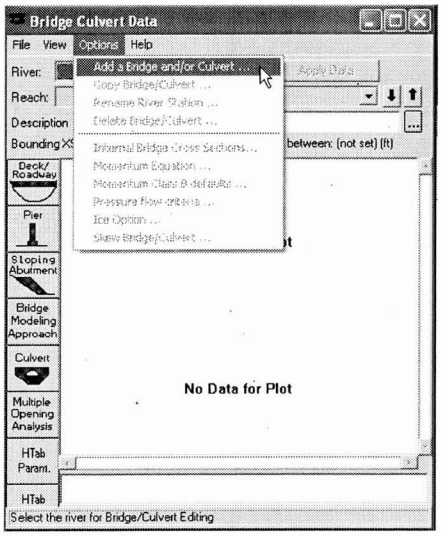
Appendix D on WSPRO Coefficients

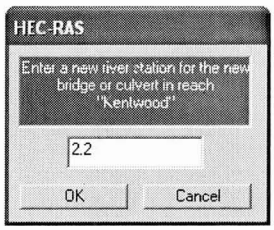
Flow Transitions in Bridge Backwater Analysis, HEC Research Doc. 42, September 1995.

Once all of the necessary cross-section data have been entered, the modeler can then add any bridges or culverts that are required. The bridge routines in HEC-RAS allow the modeler to analyze a bridge with several different methods without changing the bridge geometry. The bridge routines have the ability to model low flow (Class A, B, and C), low flow and weir flow (with adjustments for submergence), pressure flow (orifice and sluice gate equations), pressure and weir flow, and high flows with the energy equation only.

The culvert routines provide for complete analysis for both inlet and outlet control. Multiple culverts, both identical and different, can be modeled. The program also can model multiple bridge and/or culvert openings at a single location.

Adding a Bridge or Culvert





- Select River - Reach
- Option - Add a Culvert
- Enter River Station
- OK

2

Culvert data are entered in the same manner as bridge data. To enter culvert data the user presses the **Bridge/Culvert** button on the Geometric Data window. Once this button is pressed, the Bridge/Culvert Data Editor will appear

To add a culvert group to the model the user must then do the following:

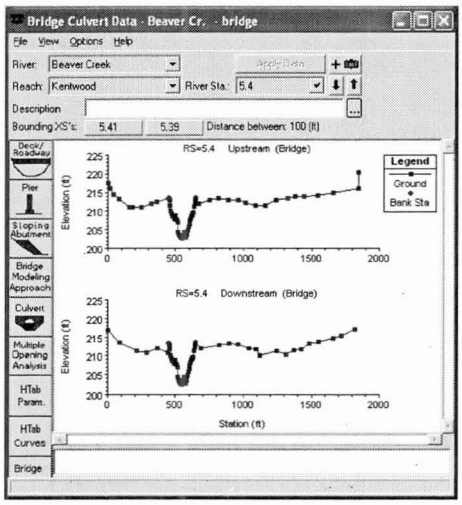
1. Select the river and reach that you would like to place the culvert in. This is accomplished by pressing the down arrow on the reach box, and then selecting the reach of choice.

2. Go to the **Options** menu of the Bridge/Culvert editor and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new culvert group. After entering the river station, press the **OK** button and the cross sections that bound the new culvert group will appear in the editor.

3. Enter all of the required data for the culvert group. This includes the culvert deck information and the culvert specific data. The culvert deck and roadway information is entered in the same manner as a bridge (using the deck/roadway editor). To enter culvert specific data, press the **Culvert** button on the Bridge/Culvert Data editor.

4. Once all of the culvert data are entered, press the **OK** button in order for the interface to accept the information.

Bridge - Culvert Data Window



Bridge Application

- Deck/Roadway
  - Top of Road
  - Approach Fill area
  - Bridge Low chord
- Pier Data
- Sloping Abutments
- Bridge Modeling Approach

3

3. Enter all of the required data for the new bridge. This may include:
   - Bridge Deck
   - Abutments
   - Piers
   - Bridge modeling approach information
4. Enter any desired optional information. Optional bridge information is found under the **Options** menu at the top of the window.
5. Press the **Apply Data** button for the interface to accept the data.

During the hydraulic computations, the program automatically formulates two additional cross sections inside of a bridge structure. The geometry inside of the bridge is a combination of the bounding cross sections (2 and 3) and the bridge geometry.

The bridge deck editor is used to describe the area that will be blocked due to the bridge deck, road embankment and vertical abutments. To enter bridge deck information the user presses the **Deck** button on the Bridge/Culvert Data Editor. Once the deck button is pressed, the Deck Editor will appear, as shown on the next page (except yours will be blank).

Bridge Modeling Approach

- Low Flow Methods
  - Energy
  - Momentum
  - Yarnell
  - WSPRO
- Use or Compute
- Highest Energy option
- High Flow Methods
  - Energy Only
  - Pressure & Weir

4

The Bridge Modeling Approach editor is used to define how the bridge will be modeled and to enter required coefficients. Press the **Bridge Modeling Approach** button on the Bridge/Culvert Data editor. The editor will appear as shown (except yours will only have the default methods selected). **Low Flows Options**. To use any or all of the low flow methods, click the buttons under the **Compute** column. If either the Momentum or Yarnell method are selected, the user must enter a value for the coefficients.

Only one of the methods can be selected as the answer to "**Use**". An alternative to selecting a single method is to use the **Highest Energy Answer.**

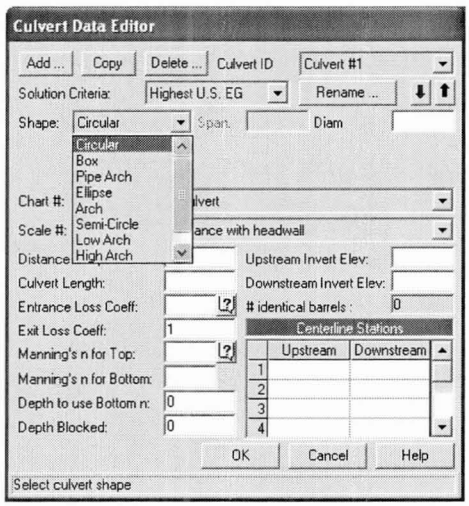
High Flow Options. You can choose between Energy-based calculations or pressure and weir flow calculations. Energy calculations will be based on standard-step calculations through the bridge.

If pressure and weir flow is the selected, the user must enter coefficients. The first coefficient applies to the equation that is used when only the upstream side (inlet) of the bridge is submerged. If this coefficient is left blank, the program selects a coefficient based on the submergence. The second coefficient applies to the pressure equation when both the upstream and downstream end of the bridge is submerged. Generally this coefficient is around 0.8. For more information on pressure flow coefficients see Hydraulics of Bridge Waterways (FHWA, 1978).

Max Low Chord - Sets the elevation at which pressure flow begins to be calculated. If this field is left blank, then the highest low chord elevation is used.

Once all of the bridge modeling approach information is entered, press the **OK.**

Culvert Data



- Culvert #1 (can Rename)
- Shape - 9 available
- Size reflects shape
- Chart - culvert type
- Scale - entrance type
- Distance to upstream section
- Culvert Length

5

Culvert Data. To enter culvert specific information, press the **Culvert** button on the Bridge/Culvert Data Editor window. When this button is pressed, the Culvert Data Editor will appear as shown above.

Shape - The shape selection box allows the user to select from one of the eight available shapes. This is accomplished by pressing the down arrow on the side of the box, and then selecting one of the eight available shapes.

Rise - The rise field describes the maximum height inside of the culvert.

Span - The span field is used to define the maximum width inside of the culvert. he span is left blank for circular culverts.

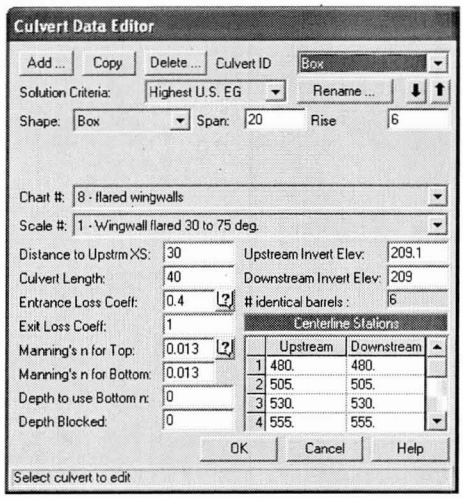
Chart #. - This field is used to select the Federal Highway Administration Chart number that corresponds to the type and shape of culvert being modeled. Once the user has selected a culvert shape, the corresponding FHWA chart numbers will show up in the chart # selection box. See Hydraulics Reference manual.

Scale. - This field is used to select the Federal Highway Administration Scale number that corresponds to the type of culvert entrance. Once the user has selected a culvert shape and chart #, the corresponding FHWA scale numbers will show up in the scale selection box.

Distance to Upstream XS - The distance from the upstream cross section to the upstream end of the culvert. This data is only used to plot the culverts.

Length - The length field describes the length of the culvert along the centerline of the barrel.

Culvert Data - *continued*



- Entrance Loss
- Exit Loss
- Manning *n* culvert
- Manning *n* Bottom
- Depth for bottom *n*
- Depth Blocked
- Invert Elevations
- Centerline Stations
- OK

6

Entrance Loss Coefficient - This coefficient will be multiplied by the velocity head inside the culvert at the upstream end. This value represents the entrance energy loss.

Exit Loss Coefficient - This coefficient will be multiplied by the change in velocity head from inside the culvert to outside the culvert at the downstream end. This value represents the energy loss that occurs as water exits the culvert.

Manning's n for Top – This is the Manning's n value of the top of the culvert barrel and the sides above the depth to use bottom n value.

Manning's n for Bottom - This is the Manning's n value of the bottom of the culvert barrel and the sides up to the depth to use bottom n value.

Depth to use Bottom n – This is the depth for using the bottom n value.

Depth Blocked – The depth blocked filed defaults to zero but can be set to fill a culvert to a input depth. The fill applies to the entire culvert length.

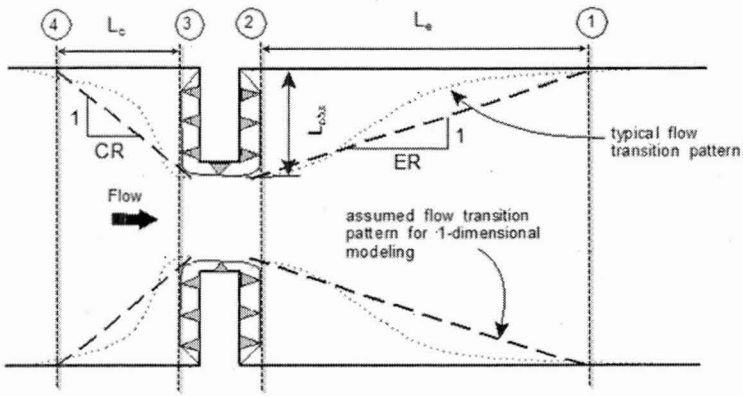
Number of Identical Barrels in Culvert - This field counts the number of identical barrels. The number of identical barrels is limited 25.

Upstream and Downstream Invert Elevations - These two fields set the culvert invert and define slope.

Upstream and Downstream Centerline Stations - These two fields set the location of the culverts within the cross section.

Once all of the culvert information is entered, the user should press the **OK.**

Flow Transitions



See Appendix B "Flow Transitions in Bridge Backwater Analysis" in *Hydraulics Reference*, a summary from HEC RD #42, September 1995.

7

The bridge and culvert routines utilize four user defined cross sections in the computations of energy losses due to the structure. A plan view of the basic cross section layout is shown above.

Cross section 1 is located sufficiently downstream from the structure so that the flow is not affected by the structure (i.e., the flow has fully expanded). This distance should generally be determined by field investigation during high flows.

Cross sections 2 and 3 are the bounding sections for the bridge. Both sections must be defined before the bridge or culvert data are input. They will be used to define the bridge interior for the upstream and downstream bridge sections. Both cross sections will usually have ineffective flow areas to either side of the bridge opening during low flow and pressure flow profiles. To model only the bridge opening these two sections, you should use the ineffective flow area option. This option is selected from the cross-section data editor.

Cross section 4 is an upstream cross section where the flow lines are approximately parallel and the cross section is fully effective. In general, flow contractions occur over a shorter distance than flow expansions.

Research Doc 42, presents results examining the flow transitions. These results are summarized in Appendix B, in the Hydraulic Reference Manual.

Ineffective Area Option

- Bounding sections provide full conveyance
- When all flow is under bridge, Overbanks cannot convey flow through highway fill
- Ineffective area option indicates overbank storage instead of conveyance
- Station should be near abutments
- Elevation is set to define when overbanks become effective - when water flows over the roadway

8

The effective area option is used to define the ineffective flow portions of the two cross sections bounding roadway crossings for bridges or culverts. If all the flow is going under the crossing, then the active flow zone immediately upstream and downstream from the crossing must be limited to the zone (area) flowing under the roadway.

Stations. If the bounding cross sections are some distance (say 5 feet or more) away from the roadway fill, the the active flow area should be wider than the width of the opening under the roadway. Modelers often use a 1 to 1 transition rate in the short distance upstream and downstream from the opening due to the more rapid flow transition near the opening. This approach applies to both bridges and culverts.

Elevations. The overbanks may be considered effective, once there is sufficient roadway overflow. Because we do not know exactly what water surface elevations will be appropriate and estimate must be made initially and then carefully reviewed after the model is run. A reasonable initial estimate is the top of roadway elevation for both upstream and downstream sections.

Ineffective-flow Zone at Bounding cross sections

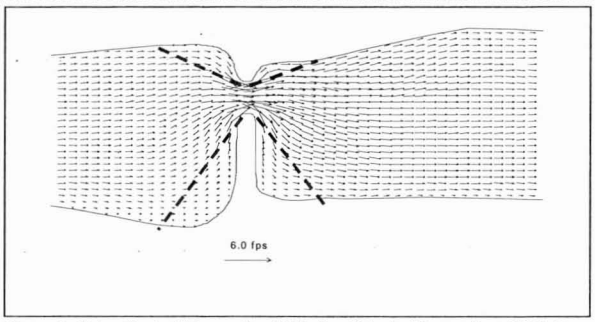


Figure 7. Poley Creek Alabama, Q = 4600 cfs. Velocity Vectors from RMA-2.

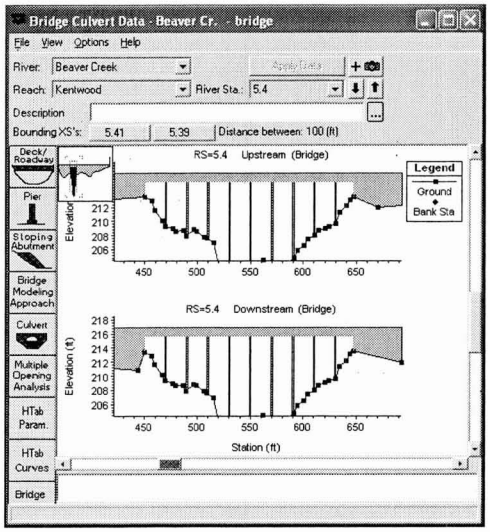
Note: Contraction - Expansion coefficients DO NOT apply to unsteady flow solution for contraction & expansion reaches

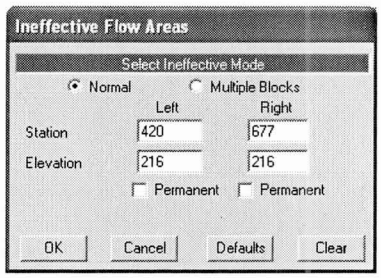
9

The ineffective area option is used at sections 2 and 3 to keep all the active flow in the area of the bridge opening until the elevations associated with the left and/or right ineffective flow areas are exceeded by the computed water surface elevation. The program allows the stations and controlling elevations of the left and right ineffective flow areas to be specified by the user. Also, the stations of the ineffective flow areas do not have to coincide with stations of the ground profile, the program will interpolate the ground station.

The ineffective flow areas should be set at stations that will adequately describe the active flow area at cross sections 2 and 3. In general, these stations should be placed outside the edges of the bridge opening to allow for the contraction and expansion of flow that occurs in the immediate vicinity of the bridge.

Ineffective Area Option





- Normal
- Multiple Blocks
- Permanent

10

The ineffective area options can be accessed from the bridge editor. Pressing the section number box invokes the cross-section editor for that section and then the Ineffective Flow editor. Two options are available: Normal or Blocked.

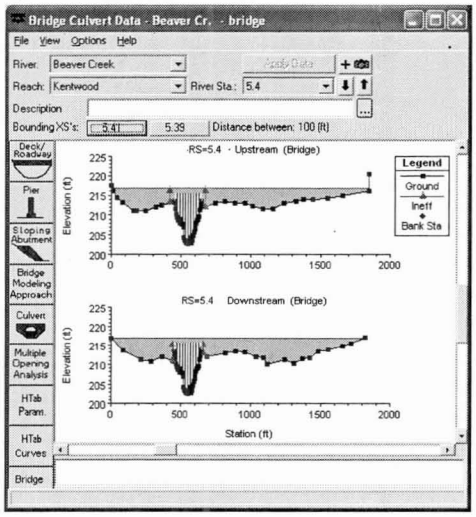
Normal makes the area to the left of Left and right of Right stations ineffective.

Blocked makes a zone, from beginning to ending station, ineffective.

Elevation defines the upper limit of the option. When the water surface elevation exceeds the input Elevation, the option "disappears." That is, the entire area is now effective.

Permanent is a new option in Version 3. If this option is selected, the ineffective area remains, even when the water surface exceeds the Elevation. The zone will remain as storage only and conveyance will be computed above the zone. When that happens, a wetted perimeter is included along the "top" of the ineffective area zone.

Hydraulic Parameters

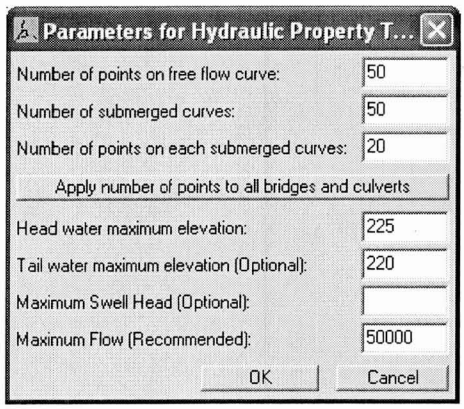


- Fix Bridge Solution first
  - Use steady flow analysis
  - Run range of flow profiles
  - Set bridge solution options
- HTab Param button provides data window
- Data entered for each bridge and culvert, or applied to all

11

Hydraulic structures, such as bridges and culverts, are converted into families of rating curves that describe the structure as a function of tailwater, flow and headwater. The user can set several parameters that can be used in defining the curves. To set the parameters for the family of rating curves, the user can select the **HTab Parameter** button from the Bridge and Culvert editor or from the Hydraulic connection editor. When this button is pressed the following window will appear:

Hydraulic Property Tables



- Default setting shown
  - Max curves = 50
  - Max points = 20
- Apply to all?
- Max head water elev.
- Optional
  - Max tailwater elevation
  - Max swell head (head - tail water difference)
  - Max flow

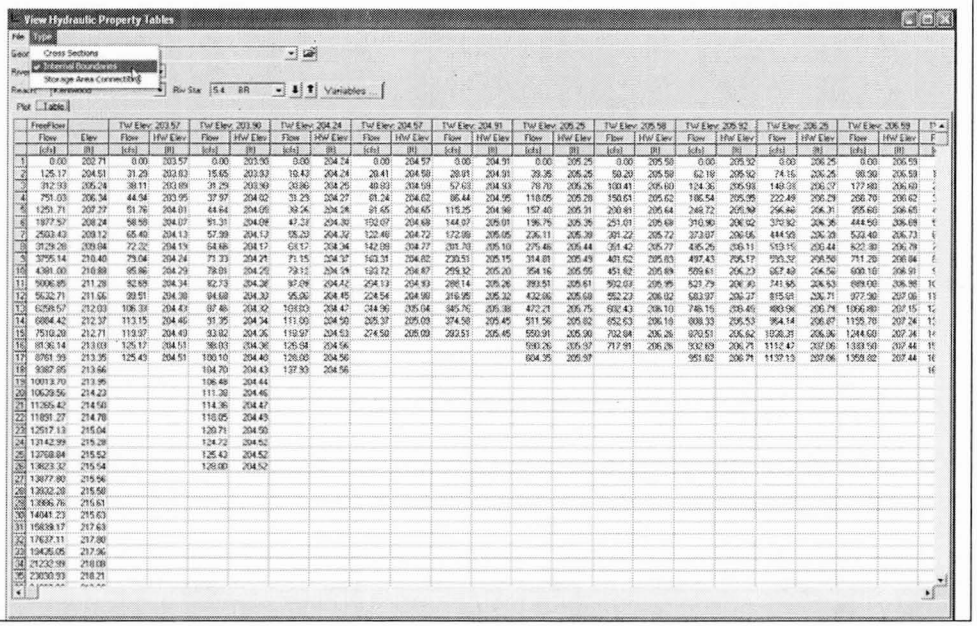
12

As shown in Figure above, the user can set the number of points to be computed on the free-flow rating curve (maximum of 50 points); the number of submerged curves to be computed (maximum of 50); and the number of points on the submerged curves (maximum of 20). The default values for these parameters are 40, 40, and 10 respectively.

The Maximum Head Water Elevation is required in Version 3.1 or later. Additionally, the user can refine the curves by putting limits on the extents of the curves. This can be accomplished by entering the tail water maximum elevation; maximum swell head (difference between the head water and tailwater); and the maximum possible flow.

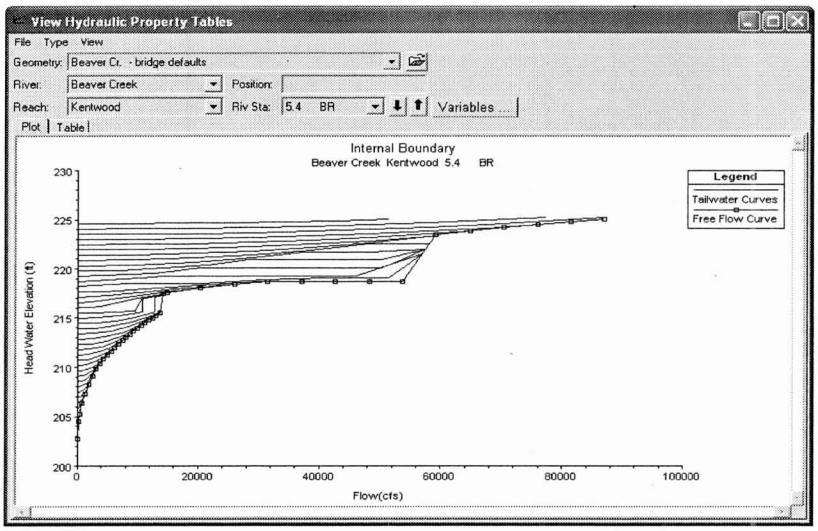
The Maximum Flow expected is good to define. This will ensure the rating curves are processed to a flow high enough, but not too high.

Hydraulic Property Table



The results for bridge and culvert processing is a family of curves. The Free Flow curve provides the Head Water-Flow relationship without Tail Water influence. Then, the family of Tail Water-Head Water relationships are shown. The curves define the change in water-surface elevations between the bounding sections.

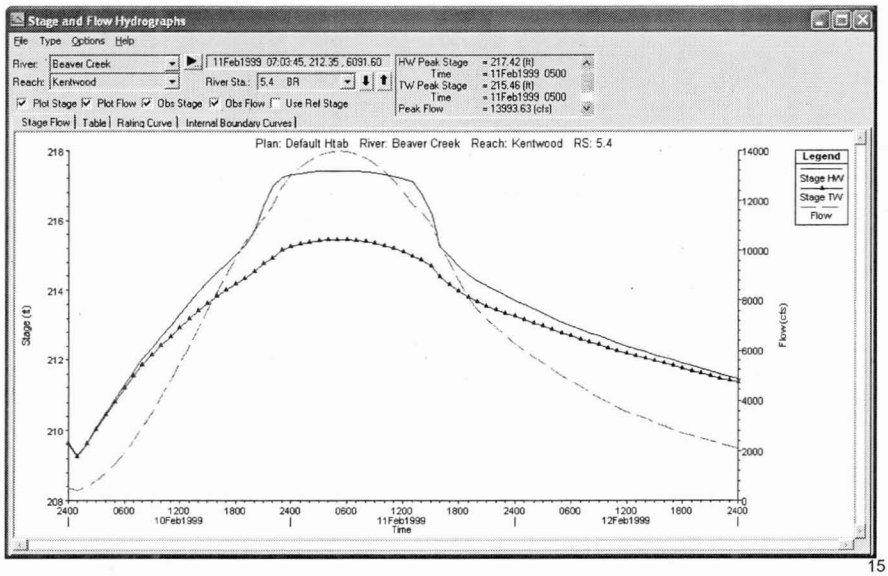
Internal Boundary Plot - Default Bridge Settings



14

The Internal Boundary curves can be displayed as a graphic or a table by selecting the appropriate Tab. The Filename provides access to HTAB results for all processed Geometry files in the Project. Only one can be displayed at a time.

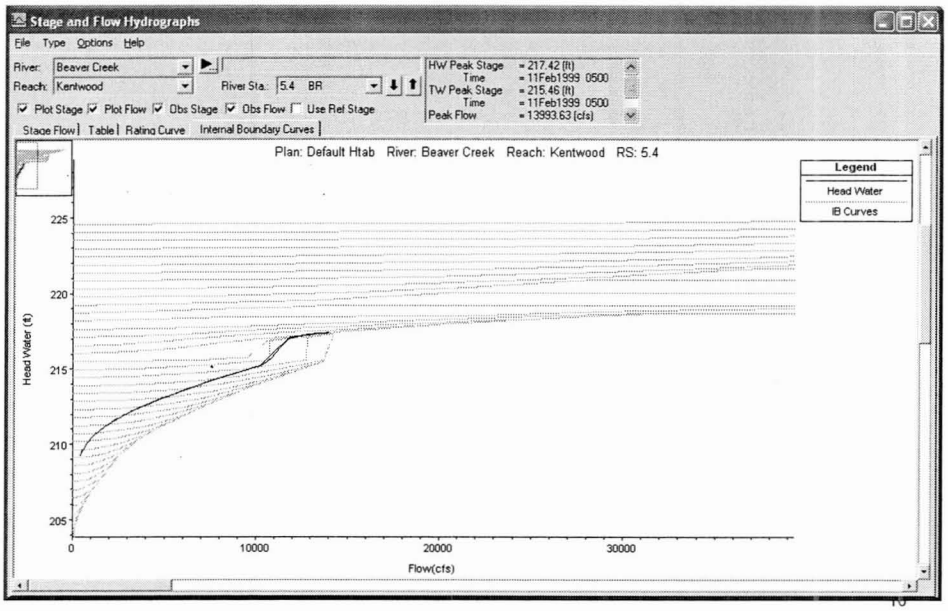
Stage & Flow Hydrograph - Bridge with Default Settings



15

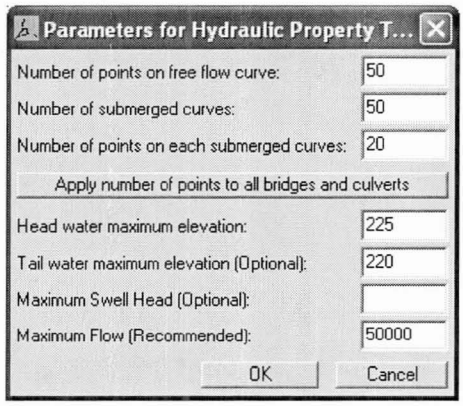
Stage and Flow hydrographs for bridges are accessed by clicking the Type menu and selecting Bridges/Culverts. The display shows the Head Water and Tail Water Stages along with Flow through the bridge. Check-boxes provide options for Stage, Flow, Observed Stage, Observed Flow and Use Reference Stage.

Internal Boundary Rating Curve



The Rating Curve was new to Version 3.1. The curve shows the Bridge HTAB Rating in light grey with a sequential plot of the Head Water Stage – Flow for the solution. This plot shows where the solution lies on the family of curves. If the solution lies in empty space in the bridge rating, an improved rating with more data in the region should improve the solution.

HTAB - Set Optional Settings



- Increase number of curves & points
- Shorten the range to provide better definition
  - Reduced headwater
  - Set Max Tailwater
  - Set Max Flow

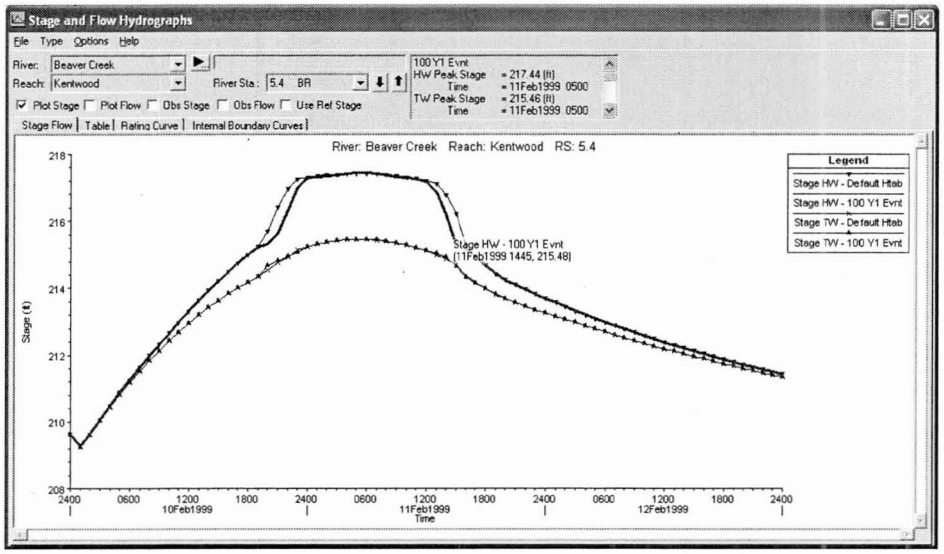
  (default = Flow at Max Free-flow Headwater)
- Or set optional Max Swell Head

17

The first demonstration solution is performed with the Default settings, plus the required Headwater elevation. The second example has HTAB settings to increase the data within the range of expected solutions.

The number of curves was increased to 50 and number of points on the submerged curves to 20. The Maximum Headwater elevation was reduced one foot, the tail water was set, and the Maximum Flow was set to a value slightly higher that the expected values.

Stage & Flow Hydrographs
Default (red) & Optional HTAB

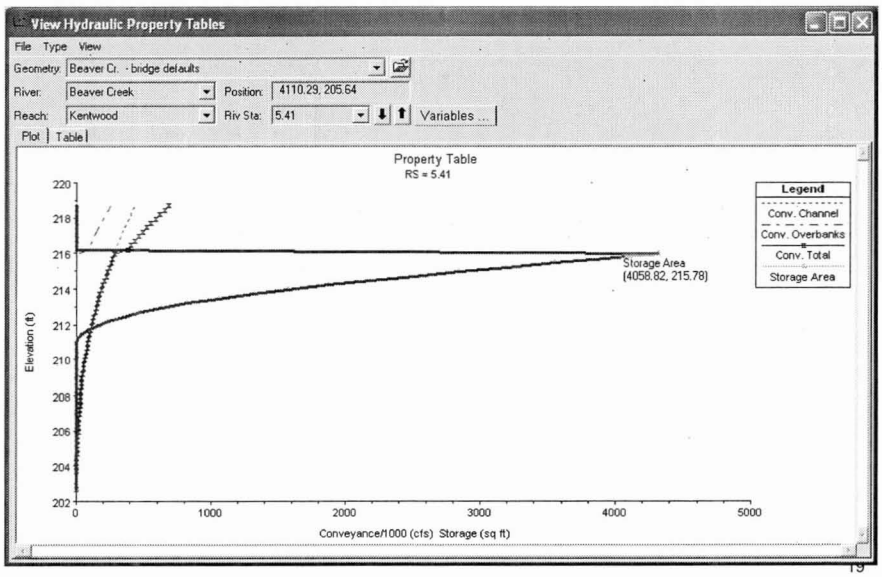


18

The Stage hydrographs are shown for the two solutions at the bridge. The curve in red (dark) is the created using "optional" values for the HTAB processor. The results are essentially the same except during the transition into and out of pressure flow. The transition into and through pressure flow is often the most difficult portion of the solution.

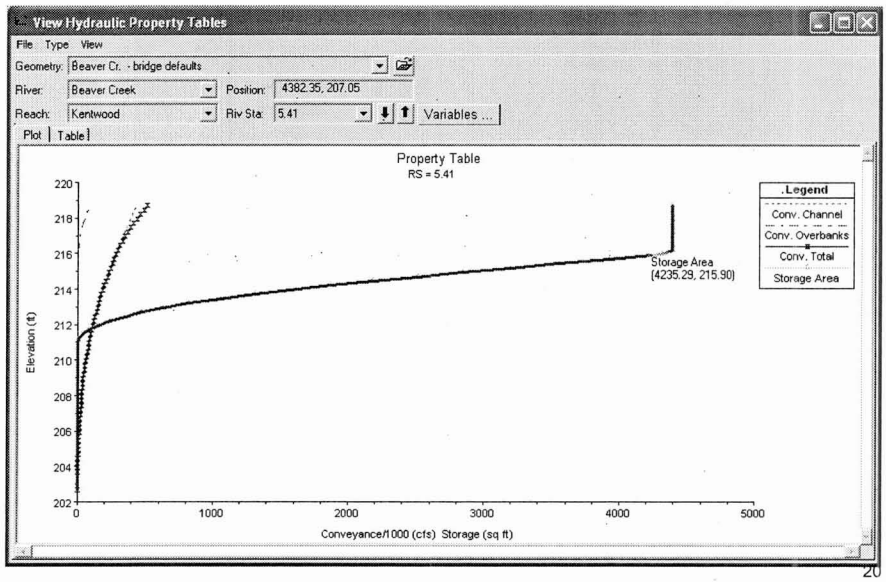
Here you see that using the more "refined" hydraulic properties results in a more abrubt transition through pressure flow. The answer may be more accurate but result in a less stable model.

Cross Section Properties with Normal Effective Area Option



The effective area option also impacts on the transition through pressure flow and into weir flow. The red (bold) curve shows the storage for the Normal Effective Area option. Once the controlling elevation is exceeded, the option disappears, the storage zone becomes conveyance, and there is a sharp increase in conveyance. The transition between the option being on and off is a zone of potential instability, similar to exceeding the Levee Option elevation. One solution to this problem is to raise the Manning's n values in the ineffective area zone, so when they turn off the transition is not so abrupt. This also gives more reasonable flow conveyance for flow going over the bridge embankment in the overbank areas.

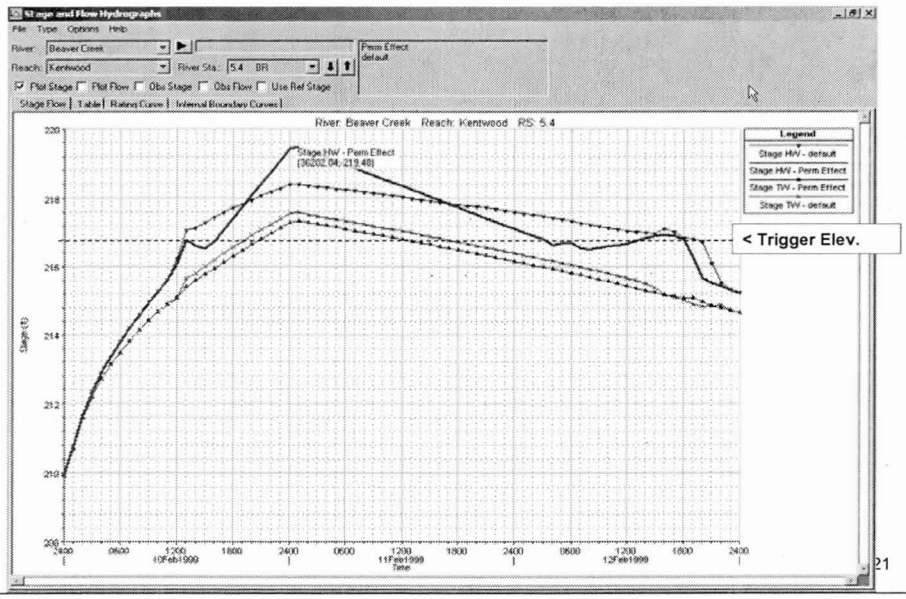
Cross Section Properties
Permanent Ineffective Area Option



20

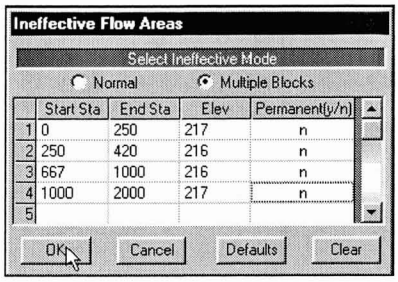
The Permanent Option keeps the Storage Area for all elevations exceeding the control elevation. Below the controlling elevation, the two options are the same. When the elevation is exceeded, Conveyance is only computed above the storage area. This option reduces the shock from the storage area suddenly disappearing. The following slide shows the application of Permanent Storage compared to the Normal option with the Default settings.

Headwater - Tailwater Stage
Normal & Permanent Ineffective Area

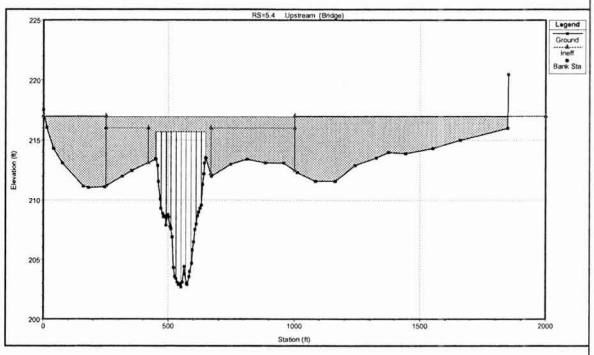


While the Permanent Option would seem to reduce the transition problem, it often does not. The problem comes from the limited conveyance available with the over-bank area treated as permanent storage. The upstream energy is higher for the high flows, but the increased upstream energy goes to Velocity Head. The water surface does not smoothly increase. The result, above, shows an upstream water surface dropping and rising as the solution progresses. Also, the upstream water surface around the peak is much higher because the cross section has less conveyance with the permanent storage.

Multiple Blocks Ineffective Area



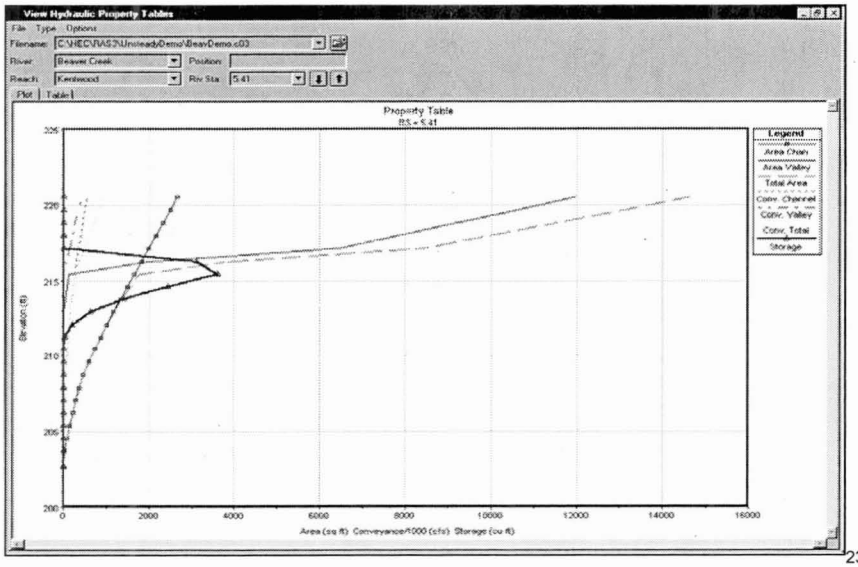
	Start Sta	End Sta	Elev	Permanent(y/n)
1	0	250	217	n
2	250	420	216	n
3	667	1000	216	n
4	1000	2000	217	n
5				



22

A second approach could set the effective area option at several elevations. The zone near the channel would disappear first providing a limited increase in conveyance. Then, as the flow-stage goes higher, the second block would become conveyance. This should reduced the shock of the storage changing to conveyance and it seems reasonable that the area near the channel would become active conveyance before the fringe area would. This approach is demonstrated in the next slide.

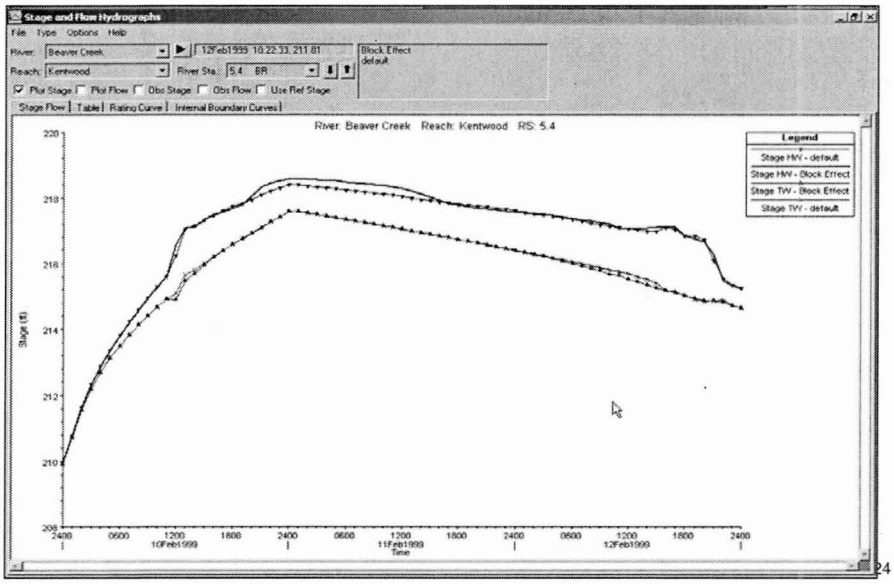
Cross Section Properties
Blocked Ineffective Area Option



23

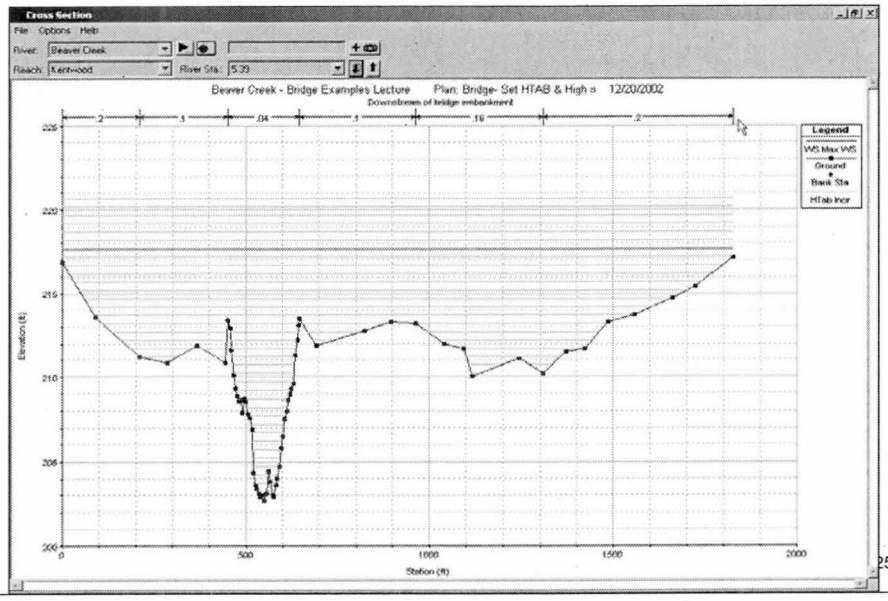
The storage for the multiple block option is shown above for the downstream bounding cross section. The storage decreases in two elevation steps, instead of one.

Headwater - Tailwater Stage
Normal & Blocked Ineffective Area



The bridge stage hydrographs are shown for the Default and the Multiple-blocked effective area option. The results do not significantly vary for this example. However, the approach has potential application for wide cross-sections where total over-bank conveyance does not make sense..

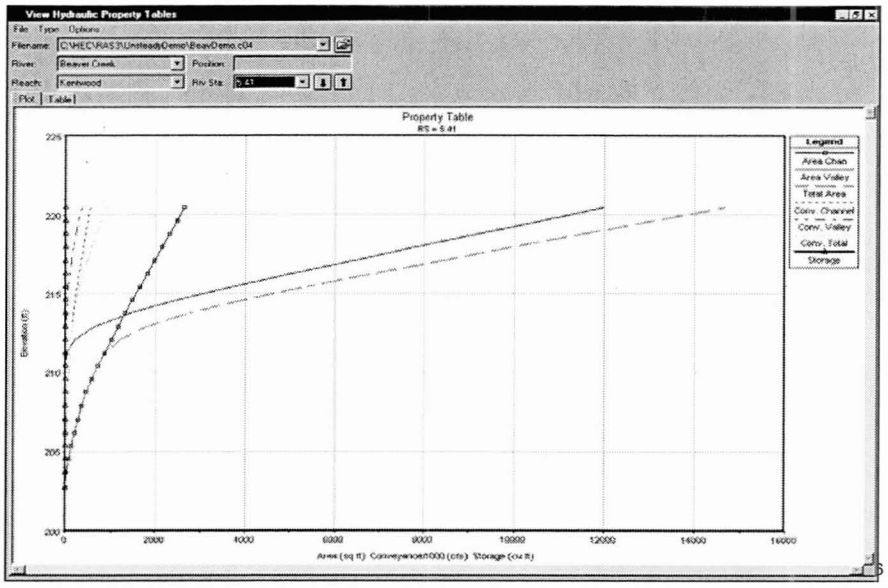
High *n* values for Ineffective Area



25

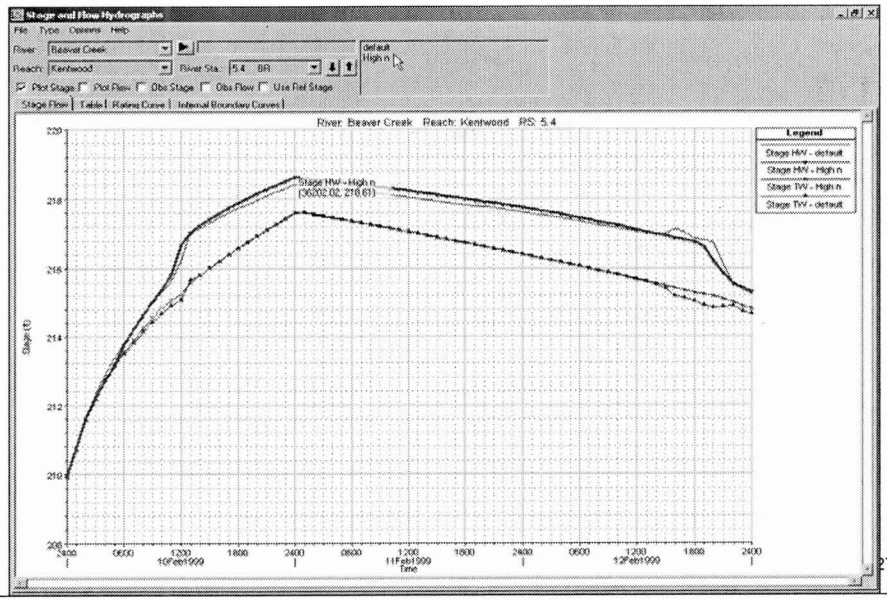
At times, the effective-area option may create computational instability that cannot be resolved. An alternative is to use high *n* values to reduce conveyance. This approach is often used in the bounding sections, with the effective-area option, to better match over-bank flow with the roadway overflow from the bridge solution. <u>In this example, the effective-area option is not used</u>. Therefore, in this approach the high *n* values replace the effective-area option.

Cross Section Properties
High *n* for Ineffective Area



The HTAB results show no Storage. The storage area is all conveyance, but at a lower value due to the high *n*.

Headwater - Tailwater Stage
Normal Ineffective Area & High *n*



27

The red (bold) Stage Hydrograph for the High *n* and the Default solution is shown above. The transition zone is much smoother for the High *n* solution. Also, the peak stage is higher due to the high *n* values. With this approach, care must be taken to ensure that the high *n* values do not unreasonable raise the peak stage.

Ineffective Area Recommendations

- Set the Ineffective Area option to eliminate overbank flow in the bounding sections where the flow is blocked by the roadway, until significant roadway overflow
- Use higher overbank *n* to "balance" overbank flow in bounding sections with computed overflow
  - Expect transition problems near controlling elevations
  - Higher overbank *n* will tend to reduce compuational shock
- If the ineffective Area option creates instability, try Higher *n* only
- Review bridge solutions with Bridge Tables
- Document adjustments in Bridge Description Box

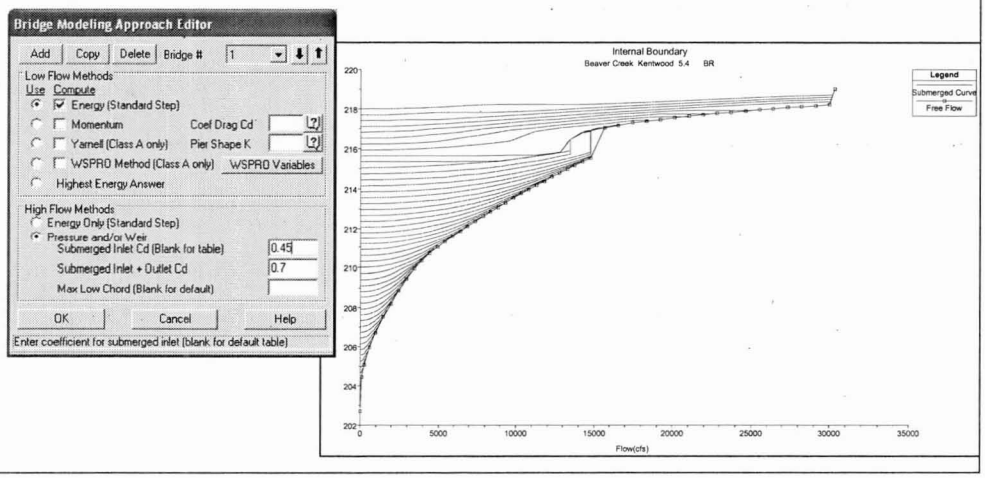
28

Generally, the bridge or culvert solution defines how flow moves through and over the roadway crossing. The modeler must manipulate the bounding cross-section characteristics to reasonably match the bridge/culvert solution. The degree these adjustments are made depend on the relative importance of the bridge solution to study objectives.

The Bridge/Culvert hydraulics should be initially resolved with Steady-flow solutions. However, there are usually additional adjustments required because the unsteady-flow solution processes through the entire range of flow from low to very high. The general approach listed above is the same as steady-flow applications. Due to the complexity of the unsteady flow solution, there may be need for additional adjustments to provide a stable solution.

Internal Boundary for Bridge with Fixed Sluice Gate Coefficient

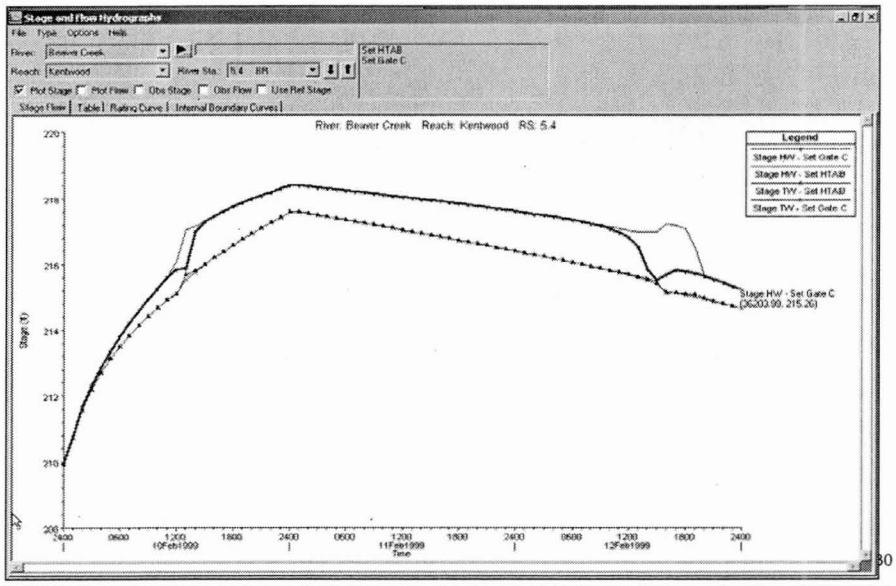
- If Pressure & Weir Rating creates a problem:



The variable gate-equation coefficient sometimes creates Head water shocks at the start of pressure flow. Hydraulic texts often show the use of a constant coefficient around 0.45 for the gate equation. In RAS, the Submerged Inlet Cd can be defined, as shown in the example above. The results for this option is shown in the next slide.

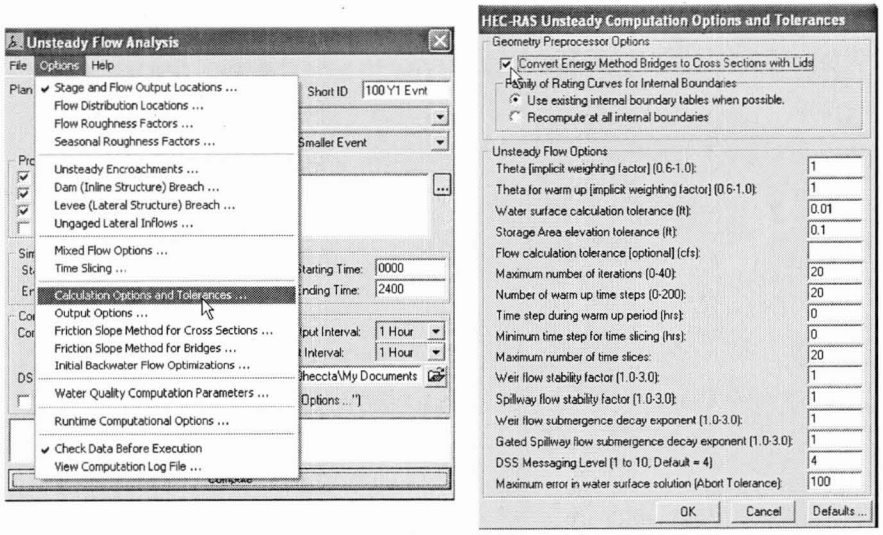
The sluice gate discharge coefficient computed by RAS is illustrated in Figure 5.5 of the Hydraulic Reference Manual.

Default & Set Sluice Gate Coefficient



The red (bold) stage hydrograph shows the fixed coefficient solution compared to the default solution using the Coefficient Curve. While neither solution looks ideal, the fixed coefficient shows a lower headwater for much of the transition through pressure flow.

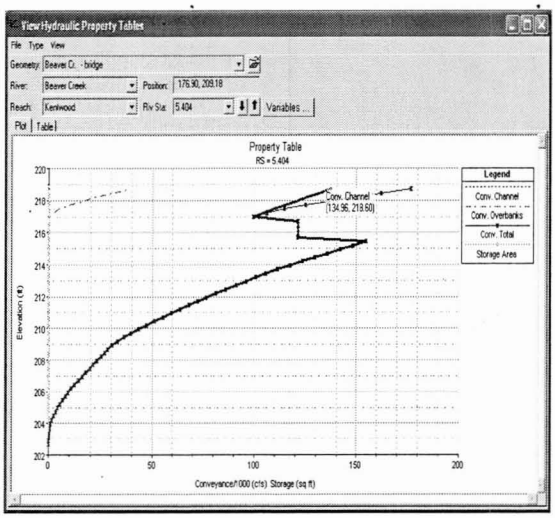
Convert Energy Method Bridge



Bridges modeled using the Energy Method for both high and low flow can be processed as cross sections with a lid. RAS will create interior sections with RS numbers between the bounding sections. These cross sections will be represented with Conveyance and Area Curves like regular cross sections.

You can process Energy Bridges as cross sections by selecting the Calculation Options from the Options Menu in the Unsteady Flow Window. At the top of the Calculation Options Menu is **Convert Energy Method Bridges to Cross Sections with Lids**. This option will only affect the Energy Only bridges.

Hydraulic Bridge Properties Energy Only processed as section

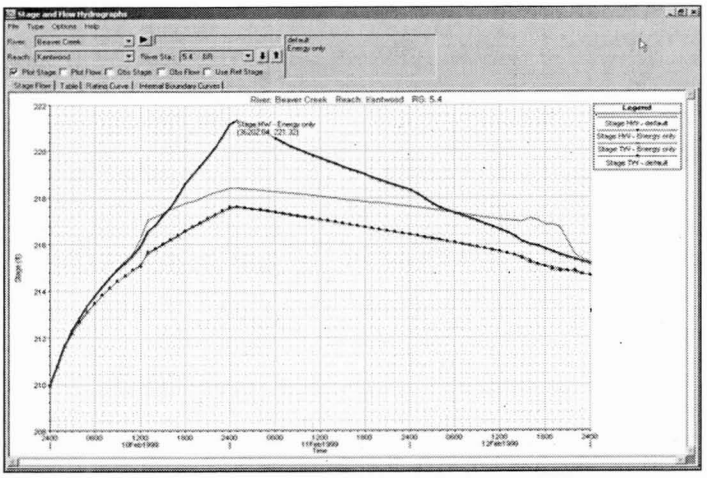


- Check results carefully
- Conveyance can quickly decrease at the bridge deck
- Better to process as internal boundary if conveyance solution is a problem

32

The Hydraulic Properties display shows the Bridge results as cross sections. The red (bold) line shows the channel conveyance. One problem with this approach is the sharp reversal as the water hits the low chord. The reversal of the functions tend to cause computational problems in the transitions zone.

Headwater - Tailwater Stage Energy Only vs. Pressure & Weir

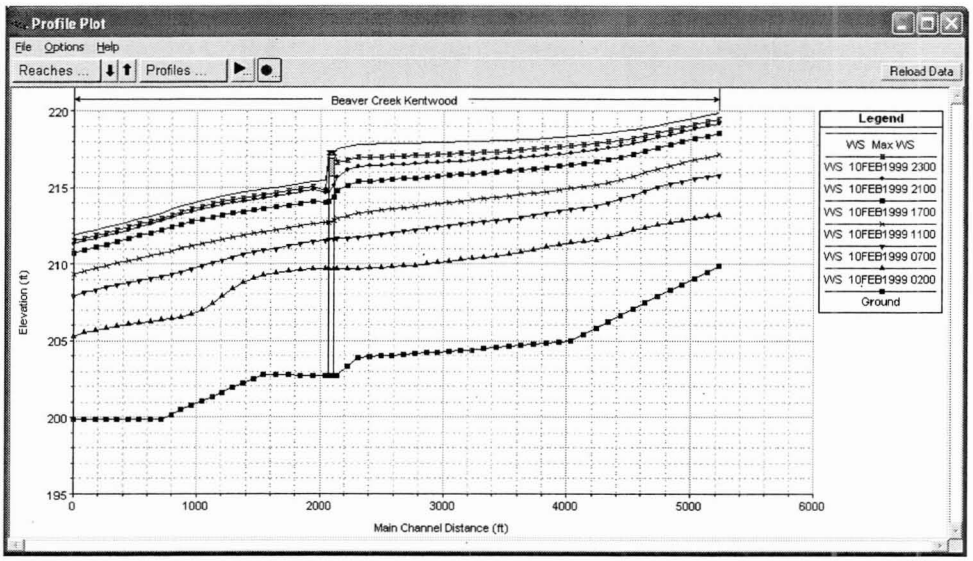


- Note that energy method will use overbank and channel *n* for high flows

33

The red (bold) stage hydrograph shows the Energy Only solution compared to the Default options. Note the smoother transition through the pressure-flow zone. However, the energy solution shows a much higher stage for roadway overflow. The energy method is using the cross-section *n* values for computing the roadway flow. The use of higher *n* values at the bounding sections will affect this solution. The interior bridge sections can be edited to modify those *n* values.

Profile Plots with Post-processing



34

With post-processing, all the standard output options are available. The profile plots can show the flow through the bridge by selecting rising, or falling stage profiles. The animation option is a convenient way to look at the profiles, one at a time.

Profile Output - Bridge Only

Profile Output Table - Bridge Only

File Options Std. Tables Locations Help

HEC-RAS Plan: 100 Y1 Evnt River: Beaver Creek Reach: Kentwood [Reload Data]

Reach	River Sta	Profile	E.G. US. (ft)	Min El Prs (ft)	BR Open Area (sq ft)	Prs O WS (ft)	Q Total (cfs)	Min El Weir Flow (ft)	Q Weir (cfs)	Delta EG (ft)
Kentwood	5.4	10FEB1999 1700	215.29	215.70	1600.36		9021.07	216.94		0.50
Kentwood	5.4	10FEB1999 1800	215.59	215.70	1600.36		9701.97	216.94		0.54
Kentwood	5.4	10FEB1999 1900	215.86	215.70	1600.36		10347.70	216.94		0.58
Kentwood	5.4	10FEB1999 2000	216.12	215.70	1600.36		10962.00	216.94		0.62
Kentwood	5.4	10FEB1999 2100	216.36	215.70	1600.36		11532.66	216.94		0.65
Kentwood	5.4	10FEB1999 2200	216.52	215.70	1600.36	218.94	12081.20	216.94	1403.04	1.66
Kentwood	5.4	10FEB1999 2300	216.88	215.70	1600.36	219.27	12328.34	216.94	1601.72	1.63
Kentwood	5.4	10FEB1999 2400	217.38	215.70	1600.36	219.73	12675.01	216.94	1904.32	1.58
Kentwood	5.4	11FEB1999 0100	217.75	215.70	1600.36	220.40	13165.47	216.94	2330.28	1.52
Kentwood	5.4	11FEB1999 0200	217.58	215.70	1600.36	221.06	13616.86	216.94	2733.54	1.78
Kentwood	5.4	11FEB1999 0300	217.61	215.70	1600.36	221.30	13794.45	216.94	2880.60	1.77
Kentwood	5.4	11FEB1999 0400	217.66	215.70	1600.36	221.48	13921.05	216.94	2985.44	1.76
Kentwood	5.4	11FEB1999 0500	217.68	215.70	1600.36	221.59	13985.18	216.94	3044.60	1.75
Kentwood	5.4	11FEB1999 0600	217.68	215.70	1600.36	221.58	13974.81	216.94	3042.34	1.75
Kentwood	5.4	11FEB1999 0700	217.65	215.70	1600.36	221.45	13889.65	216.94	2962.04	1.76
Kentwood	5.4	11FEB1999 0800	217.59	215.70	1600.36	221.22	13723.07	216.94	2818.77	1.77
Kentwood	5.4	11FEB1999 0900	217.55	215.70	1600.36	220.81	13454.25	216.94	2591.77	1.79
Kentwood	5.4	11FEB1999 1000	217.51	215.70	1600.36	220.35	13141.16	216.94	2313.67	1.82
Kentwood	5.4	11FEB1999 1100	217.46	215.70	1600.36	219.84	12762.67	216.94	1985.70	1.86
Kentwood	5.4	11FEB1999 1200	217.30	215.70	1600.36	219.58	12559.60	216.94	1801.93	1.58

35

The Bridge Only Summary Table provides all bridge solutions at the post-processing interval. This table is useful to see bridge flow and weir flow. Note when weir flow begins and overflow magnitude. This information will be compared to the flow distribution in the bounding cross sections.

Profile Output - 6 Sections

Profile Output Table - Six XS Bridge

File Options Std. Tables Locations Help

HEC-RAS Plan: 100YT Evnt River: Beaver Creek Reach: Kentwood

Reload Data

Reach	River Sta	Profile	E.G. Elev (ft)	W.S. Elev (ft)	Crit W.S. (ft)	Frctn Loss (ft)	C & E Loss (ft)	Top Width (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Vel Chnl (ft/s)
Kentwood	5.425	10FEB1999 2100	216.27	215.94	213.02	0.16		1778.38	1509.06	7659.69	2447.15	5.64
Kentwood	5.425	10FEB1999 2200	216.38	216.04	213.24	0.17		1796.63	1592.12	7915.16	2625.90	5.77
Kentwood	5.425	10FEB1999 2300	216.65	216.35	213.14	0.14		1838.84	1680.35	7811.56	2948.55	5.51
Kentwood	5.41	10FEB1999 2100	216.18	215.42	211.48	0.10	0.06	1716.89	24.79	11545.78	38.87	6.99
Kentwood	5.41	10FEB1999 2200	216.31	215.50	211.64			1733.08	27.06	12054.18	41.84	7.23
Kentwood	5.41	10FEB1999 2300	216.51	216.03	211.74			1834.67	723.45	10631.51	1042.97	6.00
Kentwood	5.4 BR U	10FEB1999 2100	216.16	215.25	211.76	0.26	0.03	185.75		11609.44		7.65
Kentwood	5.4 BR U	10FEB1999 2200	217.37	217.06	211.92			1843.77	342.41	10831.64	921.24	6.42
Kentwood	5.4 BR U	10FEB1999 2300	217.43	217.10	212.01			1844.15	405.53	10908.17	1090.70	6.42
Kentwood	5.4 BR D	10FEB1999 2100	215.87	214.88	211.75	0.12	0.06	185.75		11609.44		8.02
Kentwood	5.4 BR D	10FEB1999 2200	217.37	217.06	211.92			1821.51	342.41	10831.64	921.24	6.42
Kentwood	5.4 BR D	10FEB1999 2300	217.43	217.10	212.02			1824.00	405.53	10908.17	1090.70	6.42
Kentwood	5.39	10FEB1999 2100	215.69	214.81	211.49	0.20		1614.36	28.92	11572.22	8.30	7.56
Kentwood	5.39	10FEB1999 2200	215.55	214.97	211.66	0.17		1633.69	602.38	10308.06	1212.64	6.59
Kentwood	5.39	10FEB1999 2300	215.64	215.06	211.75	0.17		1643.48	628.74	10497.12	1272.07	6.64
Kentwood	5.37	10FEB1999 2100	215.12	214.81	212.70	0.16		1601.55	2099.86	7195.89	2307.14	5.55
Kentwood	5.37	10FEB1999 2200	215.26	214.95	212.79	0.16		1619.48	2225.58	7409.82	2481.84	5.62
Kentwood	5.37	10FEB1999 2300	215.33	215.02	212.84	0.16		1629.12	2293.65	7523.08	2577.29	5.65

36

The 6-Section Summary Table shows the flow distribution upstream and downstream from the bridge. Also, the components of energy loss are shown for the steps through the bridge and transition sections.

Profile Output - Bridge Comparison

Profile Output Table - Bridge Comparison

File Options Std. Tables Locations Help

HEC-RAS Plan: 100 Yr Event River: Beaver Creek Reach: Kentwood Reload Data

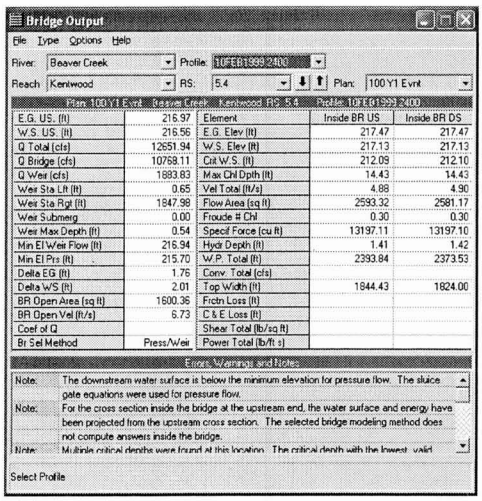
Reach	River Sta	Profile	E.G. US.	W.S. US.	Br Sel Method	Energy EG	Momen. EG	Yarnell EG	WSPRO EG	Prs O EG	Prs/Wr EG	Ener
			(ft)	(ft)		(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	
Kentwood	5.4	10FEB1999 1300	213.98	213.63	Energy only	213.93						
Kentwood	5.4	10FEB1999 1400	214.33	213.93	Energy only	214.27						
Kentwood	5.4	10FEB1999 1500	214.66	214.22	Energy only	214.59						
Kentwood	5.4	10FEB1999 1600	214.98	214.49	Energy only	214.90						
Kentwood	5.4	10FEB1999 1700	215.29	214.76	Energy only	215.19						
Kentwood	5.4	10FEB1999 1800	215.59	215.01	Energy only	215.46						
Kentwood	5.4	10FEB1999 1900	215.86	215.24	Energy only	215.72						
Kentwood	5.4	10FEB1999 2000	216.12	215.46	Energy only	215.96						
Kentwood	5.4	10FEB1999 2100	216.36	215.67	Energy only	216.19						
Kentwood	5.4	10FEB1999 2200	216.52	216.21	Press/Weir	216.49				219.05	217.36	
Kentwood	5.4	10FEB1999 2300	216.88	216.61	Press/Weir	216.59				219.36	217.42	
Kentwood	5.4	10FEB1999 2400	217.38	217.16	Press/Weir	216.73				219.82	217.48	
Kentwood	5.4	11FEB1999 0100	217.75	217.55	Press/Weir	216.91				220.47	217.56	
Kentwood	5.4	11FEB1999 0200	217.58	217.34	Press/Weir	217.06				221.13	217.61	
Kentwood	5.4	11FEB1999 0300	217.61	217.38	Press/Weir	217.12				221.37	217.64	
Kentwood	5.4	11FEB1999 0400	217.66	217.42	Press/Weir	217.17				221.55	217.67	
Kentwood	5.4	11FEB1999 0500	217.68	217.44	Press/Weir	217.19				221.66	217.67	
Kentwood	5.4	11FEB1999 0600	217.68	217.44	Press/Weir	217.19				221.65	217.66	
Kentwood	5.4	11FEB1999 0700	217.65	217.41	Press/Weir	217.16				221.51	217.66	
Kentwood	5.4	11FEB1999 0800	217.59	217.35	Press/Weir	217.10				221.29	217.63	
Kentwood	5.4	11FEB1999 0900	217.55	217.32	Press/Weir	217.00				220.88	217.59	
Kentwood	5.4	11FEB1999 1000	217.51	217.28	Press/Weir	216.88				220.43	217.55	
Kentwood	5.4	11FEB1999 1100	217.46	217.24	Press/Weir	216.74				219.92	217.50	

Energy grade elevation upstream of bridge for pressure and/or weir method.

37

The Bridge Comparison Summary Table shows the basis for each solution. While the table is usually used to compare different solution methods, the unsteady flow model should not be run using multiple methods. The decision should be made on the appropriate method and that should be fixed.

Detailed Bridge Output



Bridge Output

File Type Options Help

River: Beaver Creek Profile: 10FEB1999 2400

Reach: Kentwood RS: 5.4 Plan: 100Y1 Evnt

Plan: 100Y1 Evnt Beaver Creek Kentwood RS: 5.4 Profile: 10FEB1999 2400

			Inside BR US	Inside BR DS
E.G. US. (ft)	216.97	Element		
W.S. US. (ft)	216.56	E.G. Elev (ft)	217.47	217.47
Q Total (cfs)	12651.94	W.S. Elev (ft)	217.13	217.13
Q Bridge (cfs)	10768.11	Crit W.S. (ft)	212.09	212.10
Q Weir (cfs)	1883.83	Max Chl Dpth (ft)	14.43	14.43
Weir Sta Lft (ft)	0.65	Vel Total (ft/s)	4.88	4.90
Weir Sta Rgt (ft)	1847.98	Flow Area (sq ft)	2593.32	2581.17
Weir Submerg	0.00	Froude # Chl	0.30	0.30
Weir Max Depth (ft)	0.54	Specif Force (cu ft)	13197.11	13197.10
Min El Weir Flow (ft)	216.94	Hydr Depth (ft)	1.41	1.42
Min El Prs (ft)	215.70	W.P. Total (ft)	2393.84	2373.53
Delta EG (ft)	1.76	Conv. Total (cfs)		
Delta WS (ft)	2.01	Top Width (ft)	1844.43	1824.00
BR Open Area (sq ft)	1600.36	Frctn Loss (ft)		
BR Open Vel (ft/s)	6.73	C & E Loss (ft)		
Coef of Q		Shear Total (lb/sq ft)		
Br Sel Method	Press/Weir	Power Total (lb/ft s)		

Errors, Warnings and Notes:

Note: The downstream water surface is below the minimum elevation for pressure flow. The sluice gate equations were used for pressure flow.

Note: For the cross section inside the bridge at the upstream end, the water surface and energy have been projected from the upstream cross section. The selected bridge modeling method does not compute answers inside the bridge.

Note: Multiple critical depths were found at this location. The critical depth with the lowest valid

Select Profile

- Post processing provides Interior Bridge Data and Energies
- Contraction & Expansion is turned off
- Notes & Warnings give solution information
- Use summary tables to get overview
- Detailed for problems

38

Graphics and Summary Tables provide a good overview of the solutions. When there appears to be some difficulties in a solution, the Detailed Output Table provides the solution for a specific time. Also, the Notes section can be helpful describing the assumptions and steps of the solution.

Workshop
Modeling Bridges with Unsteady Flow

Introduction

This workshop will help students learn how to use HEC-RAS to: model bridges using unsteady flow, and to review and understand pertinent bridge output. Sensitivity of simulation results to model changes will be reviewed.

Background

Figure 1 shows the study region, Beaver Creek near Kentwood, Louisiana. Shown in Figure 2 is a photograph of the river reach, which is looking upstream at the bridge crossing. These two figures and the data for this study were taken from the USGS study "Backwater at Bridges and Densely Wooded Flood Plains, Beaver Creek Near Kentwood, Louisiana" by George J. Arcement, B.E. Colson, and C.O. Ming.

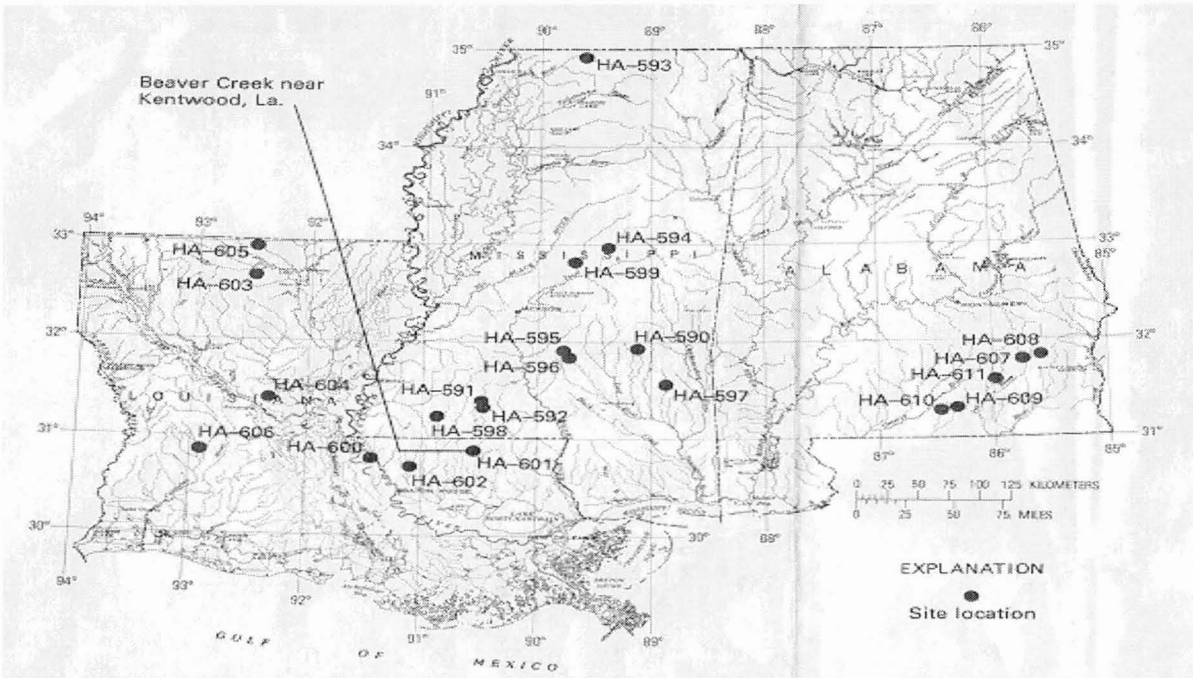


FIGURE 1.—INDEX MAP OF STUDY SITES IN THE BRIDGE BACKWATER INVESTIGATION PROJECT, ALABAMA, LOUISIANA, AND MISSISSIPPI.

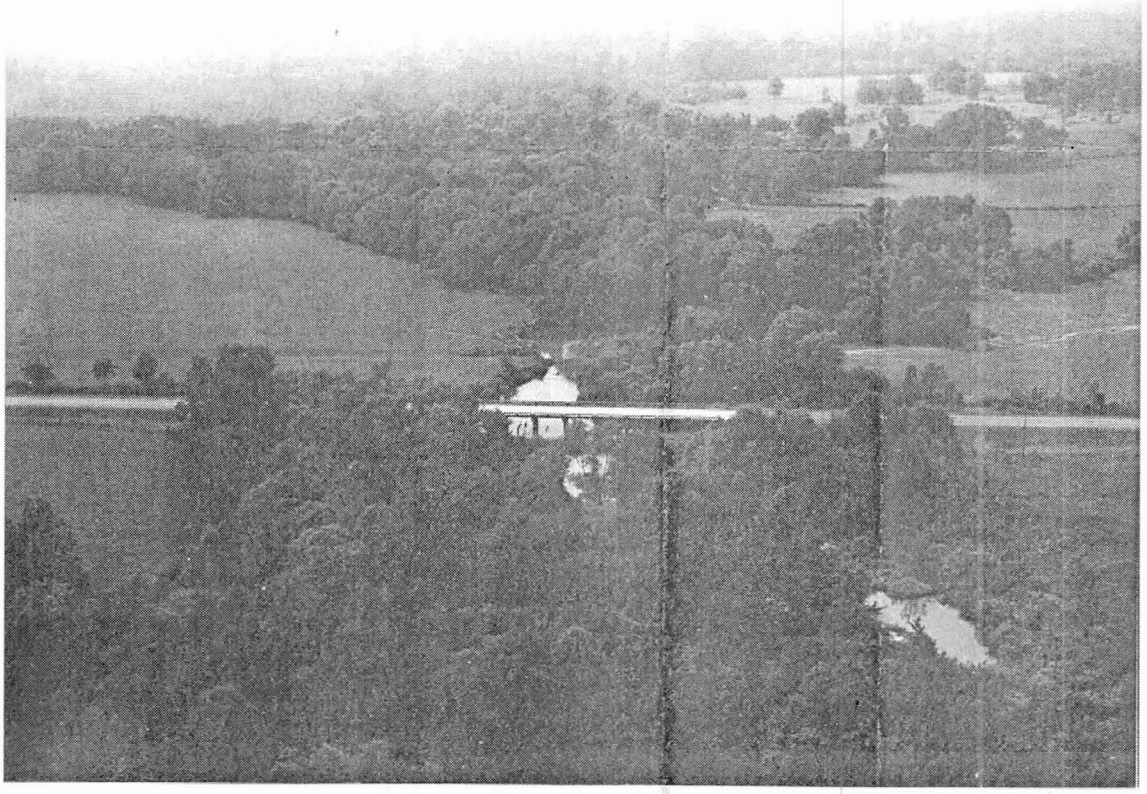


FIGURE 2—AERIAL VIEW LOOKING UPSTREAM AT BRIDGE ON STATE HIGHWAY 1049 NEAR KENTWOOD, LOUISIANA

Problem

A project file (**BridgeModeling.prj**) with the title "**Bridge Modeling Workshop - Unsteady Flow**" has been loaded on your workshop computers. This file contains all of the data for this workshop, except the bridge modeling data. The following is a summary of the required tasks for each group:

1. Open the project and enter all of the necessary information to model the bridge. Then save the data.

2. Create a plan file and compute steady-flow water surface profiles.

3. Review the results and make necessary adjustments.

4. Compare the computed results from the third profile with the observed high water marks.

5. Complete any required model calibration. Select a single method for low flow and the appropriate high-flow method.

6. Review the steady-flow results and determine the appropriate settings for processing the bridge data (HTAB).

7. Set up the model for an unsteady flow simulation with the 1974 flood hydrograph.

8. Perform the unsteady-flow simulation and evaluate the results.

Tasks

1. Enter the Bridge Information

The bridge is located at river station 5.4 of Beaver Creek. A cross section of the bridge is shown in Figure 3. The **Bridge Design Editor** can be used to enter the deck and road embankment and the pier data.

- Road embankment is at a constant elevation of 216.93 feet.
- The bridge low chord is at elevation of 215.7 feet.
- The bridge opening has vertical walls at cross section stationing 450 and 647 feet. *(A zoomed in view of the bridge opening is shown in Figure 4.)*
- The bridge deck is 40 feet wide, and the upstream side of the bridge deck is 30 feet from cross section immediately upstream of the bridge (section 5.41). The upstream and downstream sides of the bridge are exactly the same.
- As shown in Figure 4, the bridge has 9 piers. The piers are 1.25 ft. wide each and have a square nose. The piers are spaced 20.0 ft. apart on center, starting with the first pier at station 470.0 ft.

Select a bridge modeling approach for both low flow and high flow. Enter the necessary coefficients for the selected modeling approaches. Also, remember to add ineffective flow areas at the cross sections that bound the bridge. **SAVE THE GEOMETRIC DATA**.

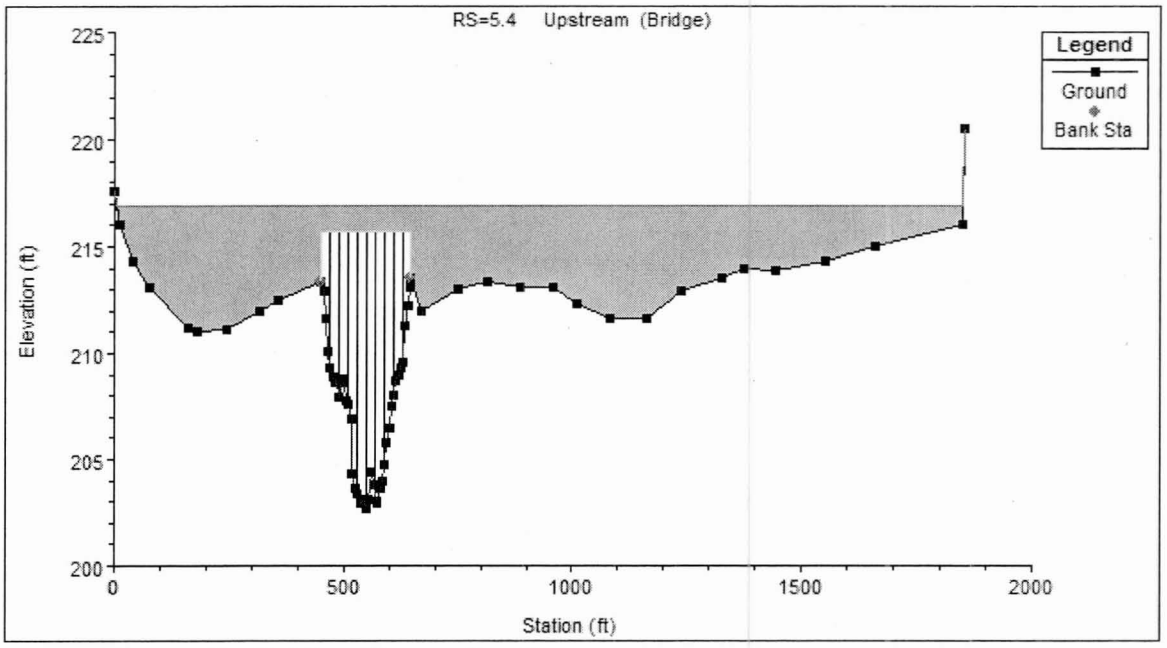


Figure 3. Bridge Cross Section

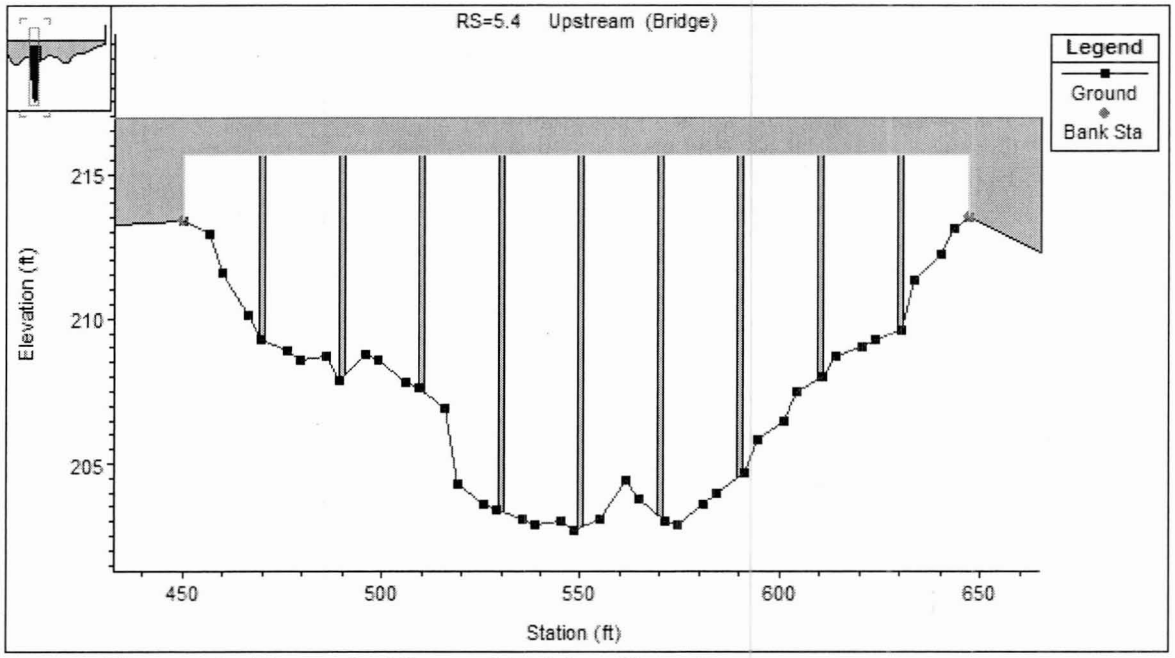


Figure 4. Zoom in View of Bridge Opening

2. Compute Steady-flow Water Surface Profiles

Once you have saved the geometric data, create a new steady flow plan. The steady-flow file is set for three profiles: 5000, 10000, and 14000 cfs. Give the plan a title and remember to enter a short ID for the plan, and save the data.

3. **Review The Results**

Review the results to ensure that the computed water surface profiles are reasonable. Make necessary adjustments to the model.

4. **Compare the Computed Results With Observed Data**

The flow rate used in the third profile (Q=14,000 cfs), is the same as an observed event that occurred on May 22, 1974. The USGS obtained high water marks and average values were estimated from the high water marks.

Table 1. Observed High Water Marks

Cross Section River Stationing	Observed WS Elevation in Feet
5.99	220.0
5.76	218.4
5.61	218.1
5.44	217.8
5.41	217.8*
Bridge 5.40	
5.39	215.2*
5.29	214.9
5.13	213.6
5.00	211.8

*Note: The observed high water mark at this location is questionable.

5. **Set up an Unsteady Flow Simulation.**

Based on your review of the steady-flow simulation, determine the appropriate settings for the bridge processing (HTAB). Set the HTAB values and save the geometry file. You don't have to rename it.

The flood hydrograph used in the previous workshop will be applied here, along with the downstream boundary and initial flow. The hydrograph represents the May 1974 flood. An unsteady flow file has been saved with your project, you will need to load it. The flow file is titled "**May 1974 Flood**" and is the "**BridgeModeling.u01**" file

Set up an unsteady flow simulation. Decide the appropriate time interval and set any required parameters. Save the plan as a new name and run the simulation.

6. **Review the Results for the Unsteady-flow Simulation.**

Review the results to ensure that they are reasonable. Compare the results for the maximum stage to those from the third profile with the observed data. Make any necessary changes and run the simulation again, until results look reasonable. Keep track of the changes you make.

Questions

1. How do your steady-flow results compare to the observed data? What methods did you use?

2. What settings did you use for HTAB on bridge model processing?

3. What changes did you make to the unsteady flow model, after you were satisfied with the steady flow solution?

4. How did the unsteady-flow simulation compare to the steady-flow profile with 14,000 cfs?

Inline and Lateral Hydraulic Structures

Gary Brunner

Overview

- Introduction
- Inline Structures
- Entering Inline Structure Data
- Lateral Structures
- Entering Lateral Structure Data
- Controlling Inline and Lateral Gate Settings
- Inline and Lateral Structure Output

Introduction

- Inline or lateral gated spillways, uncontrolled over-flow weirs, lateral culverts, and lateral rating curve.
- Can be used to model the following:
  - Large to Small Dams
  - Run of the river Structures
  - Detention ponds
  - Drop Structures or natural drops
  - Lateral storage facilities
  - Natural overflows to ponding areas

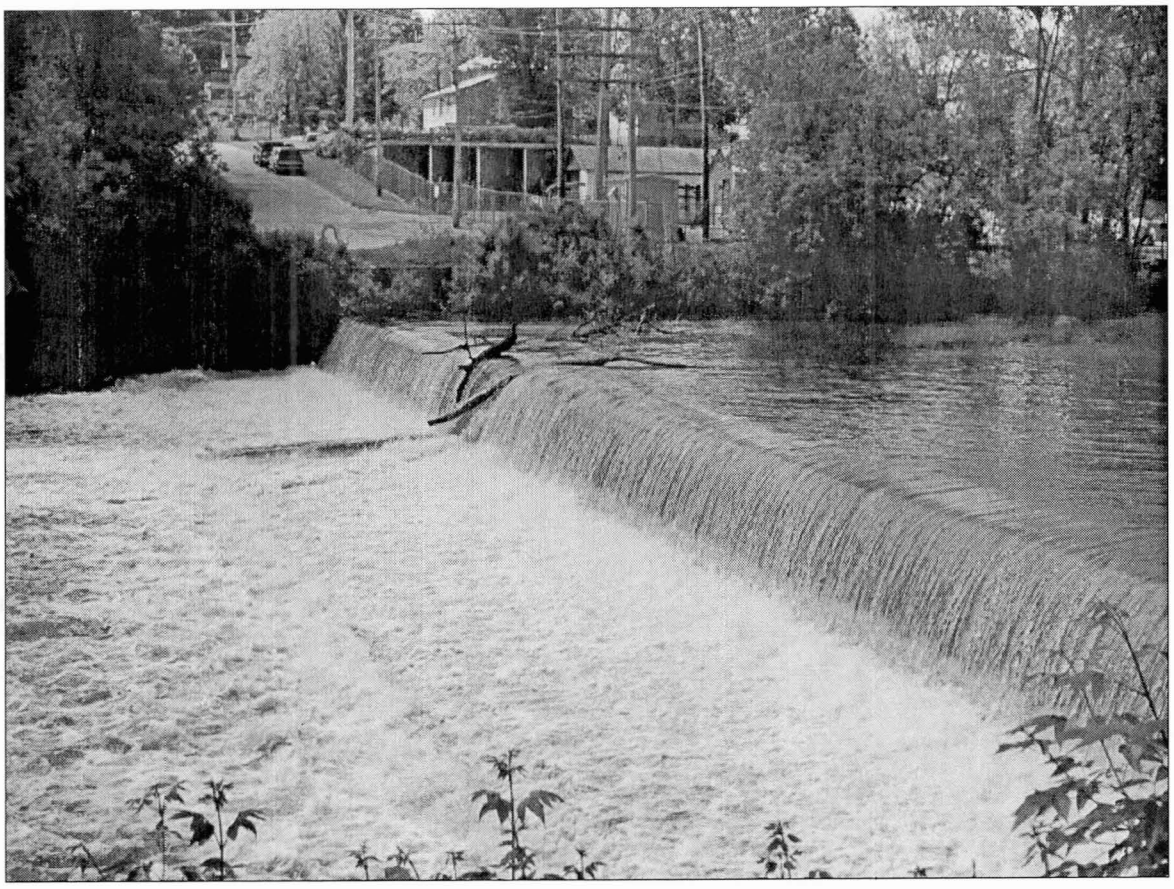
This version of HEC-RAS allows the user to model inline gated spillways, overflow weirs, drop structures, lateral gated spillways, lateral weirs, lateral culverts, and lateral rating curves. HEC-RAS has the ability to model radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape. In addition to the gate openings, the user can also define a separate uncontrolled overflow weir.

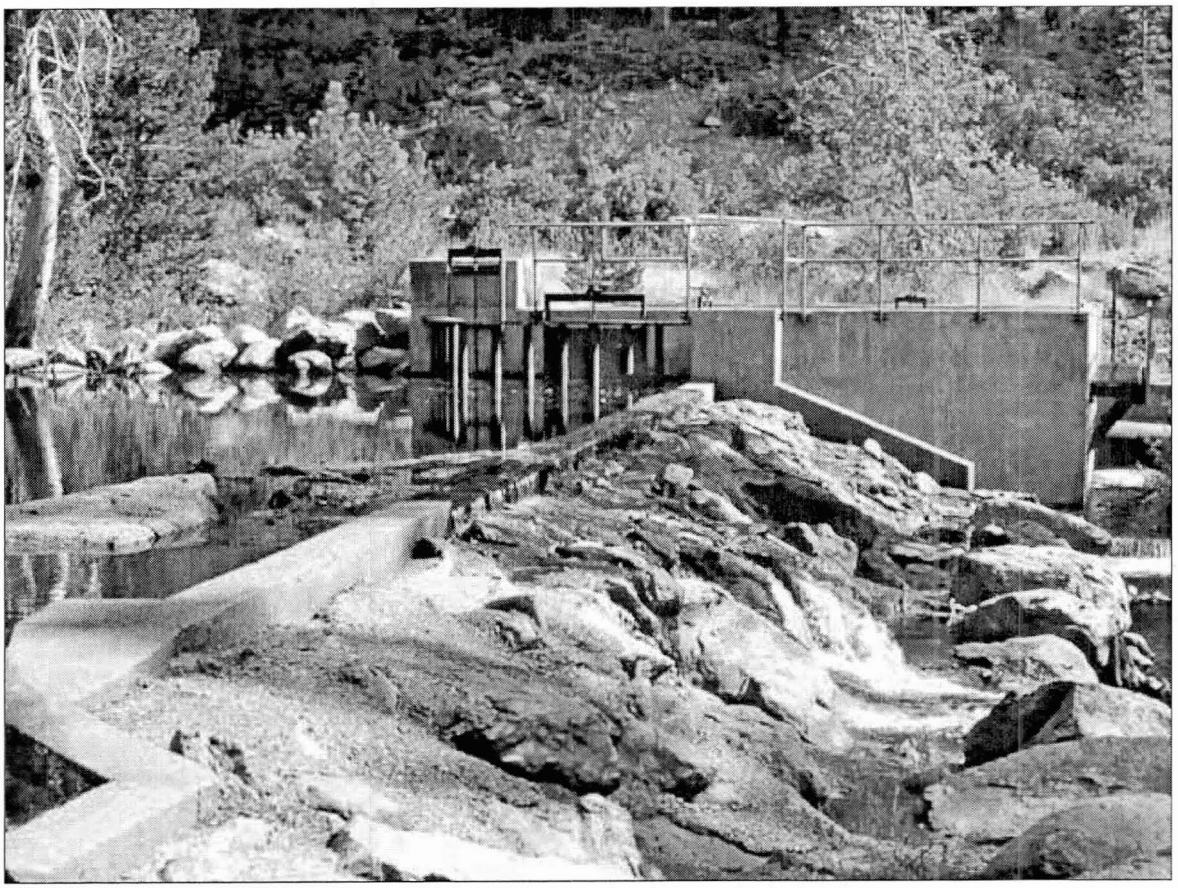
Inline Hydraulic Structures

- Radial gates (often called Tainter gates)
- Vertical lift gates (Sluice gates)
- Overflow Gates (Open air or closed top))
- Ogee, Broad Crested, or sharp crested Weir shapes for both gated spillways and overflow weirs
- Gate equations can handle low flow, normal gate flow (upstream submerged) or fully submerged gate flow (both ends submerged)
- Up to 10 gate groups can be defined at any river crossing. Up to 25 identical gates per group.

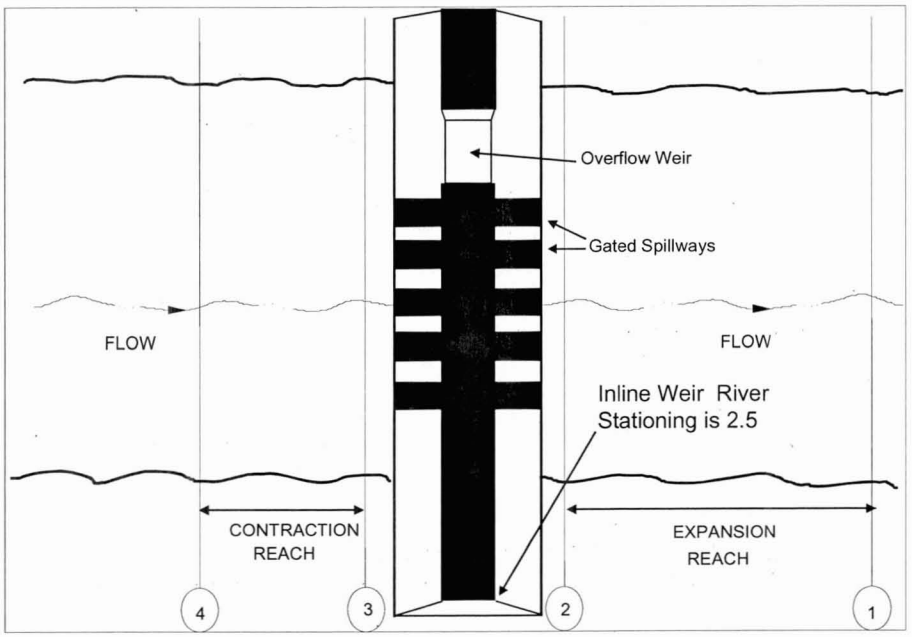
Gated Spillways within HEC-RAS can be modeled as radial gates (often called tainter gates) or vertical lift gates (sluice gates). The equations used to model the gate openings can handle both submerged and unsubmerged conditions at the inlet and outlet of the gates. If the gates are opened far enough, such that unsubmerged conditions exist at the upstream end, the program automatically switches to a weir flow equation to calculate the hydraulics of the flow. The spillway crest through the gate openings can be specified as either an ogee crest shape or a broad crested weir. The program has the ability to calculate both free flowing and submerged weir flow through the gate openings.

Up to 10 gate groups can be entered into the program at any one river crossing. Each gate group can have up to 25 identical gate openings. Identical gate openings must be the same gate type; size; elevation; and have identical gate coefficients. If anything about the gates is different, except their physical location across the stream, the gates must be entered as separate gate groups.



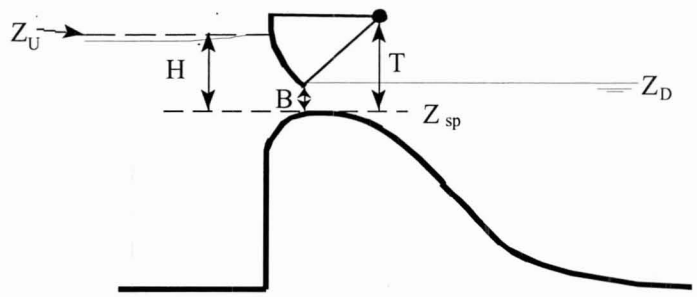


Cross Section Locations



The inline hydraulic structure routines in HEC-RAS require the same cross sections as the bridge and culvert routines. Four cross sections in the vicinity of the hydraulic structure are required for a complete model, two upstream and two downstream. In general, there should always be additional cross sections downstream from any structure (bridge, culvert, weir, etc...), such that the user entered downstream boundary condition does not affect the hydraulics of flow through the structure. In order to simplify the discussion of cross sections around the inline hydraulic structure, only the four cross sections in the vicinity will be discussed. These four cross sections include: one cross section sufficiently downstream such that the flow is fully expanded; one at the downstream end of the structure (representing the tailwater location); one at the upstream end of the structure (representing the headwater location); and one cross section located far enough upstream at the point in which the flow begins to contract. Note, the cross sections that bound the structure represent the channel geometry outside of the embankment

Radial Gates



$$Q = C\sqrt{2g}\,W\,T^{TE}\,B^{BE}\,H^{HE}$$

Where: $H = Z_U - Z_{sp}$

C = Discharge Coefficient, typically 0.6 – 0.8

The flow through the gate is considered to be "Free Flow" when the downstream tailwater elevation (Z_D) is not high enough to cause an increase in the upstream headwater elevation for a given flow rate. The equation used for a Radial gate under free flow conditions is as follows:

Where:

Q = Flow rate in cfs

C = Discharge coefficient (typically ranges from 0.6 - 0.8)

W = Width of the gated spillway in feet

T = Trunnion height (from spillway crest to trunnion pivot point)
TE = Trunnion height exponent (default 0.0)

B = Height of gate opening in feet

BE = Gate opening exponent (default 1.0)

H = Upstream Energy Head above the spillway crest $Z_U - Z_{sp}$

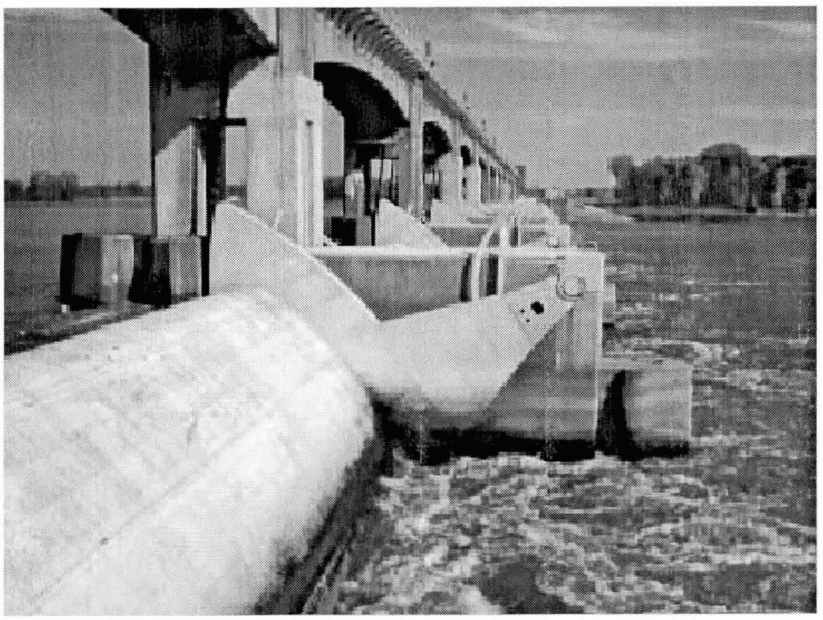
HE = Head exponent (default 0.5)

Z_U = Elevation of the upstream energy grade line

Z_D = Elevation of the downstream water surface

Z_{sp} = Elevation of the spillway crest through the gate

Radial Gates



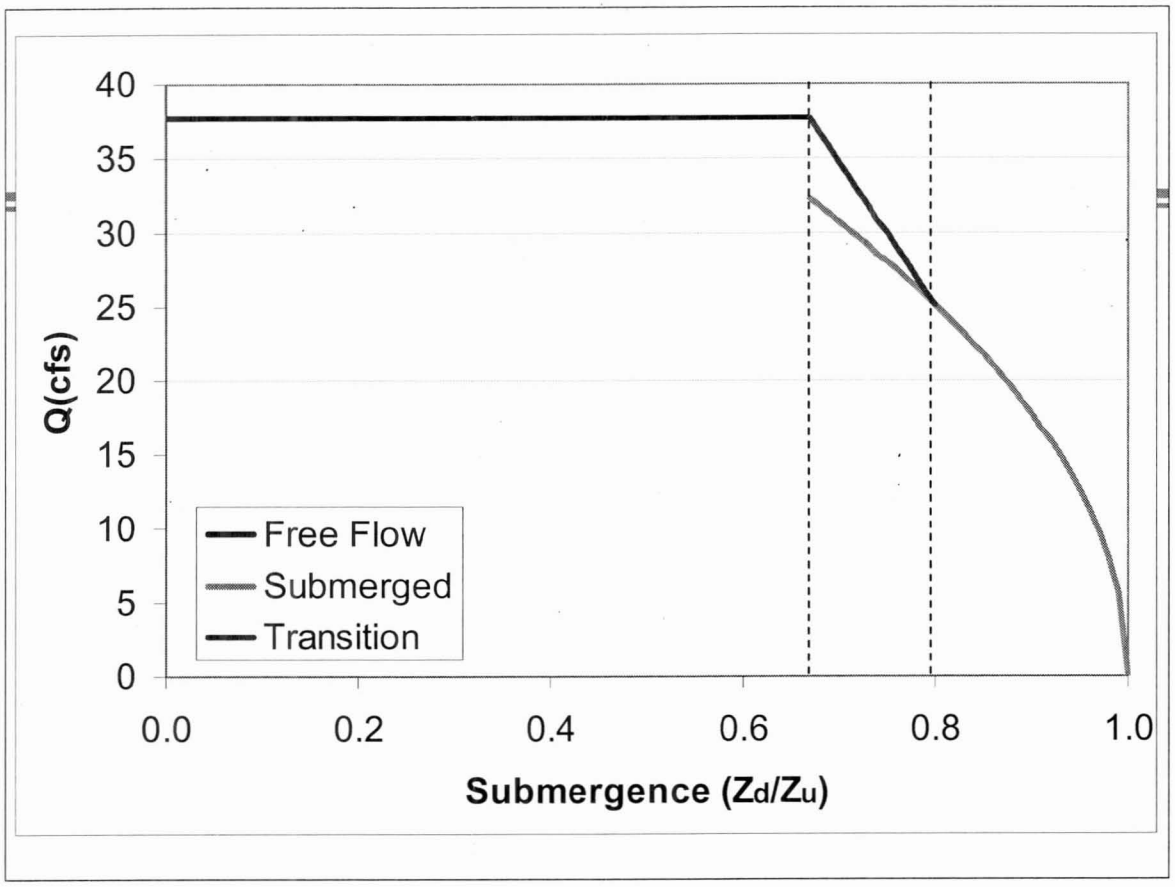
Fully Submerged Gate Flow

Orifice Equation

$$Q = CA\sqrt{2gH}$$

Where: $H = Z_U - Z_D$

C = Discharge Coefficient, typically 0.8


40
35
30
25
20
15
10
5
0
Q(cfs)
Free Flow
Submerged
Transition
0.0
0.2
0.4
0.6
0.8
1.0
Submergence (Zd/Zu)

Submerged Radial Gates

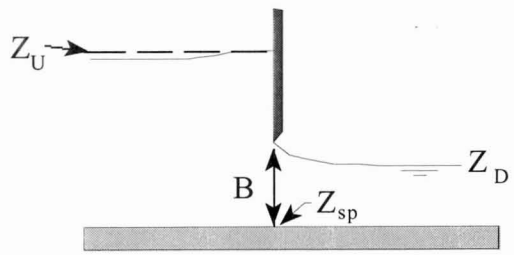
When the downstream tailwater increases to the point at which the upstream headwater is affected (downstream submergence is causing a greater upstream headwater for a given flow), the program switches to the submerged form of the equation. This is assumed to occur at 67 percent submergence.

$$Q = C\sqrt{2g}\, W\, T^{TE}\, B^{BE}\, (3H)^{HE}$$

Where: $H = Z_U - Z_D$

Submergence begins to occur when the tailwater depth above the spillway divided by the headwater energy depth above the spillway, is greater than 0.67. The Equation above is used to transition between free flow and fully submerged flow. This transition is set up so the program will gradually change to the fully submerged Orifice equation when the gates reach a submergence of 0.80

Sluice Gates

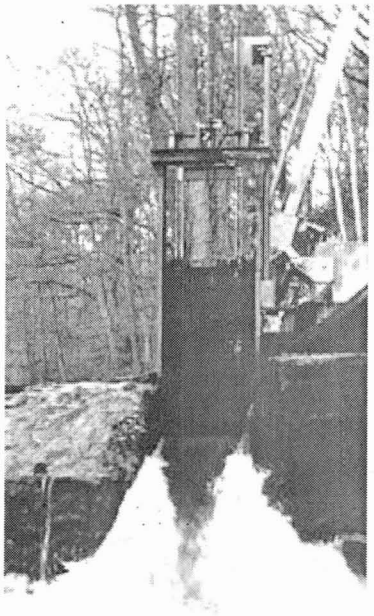


$$Q = C W B \sqrt{2gH}$$

Where: H = Upstream energy head ($Z_U - Z_{sp}$)

C = Coefficient of discharge, 0.5 to 0.7

Sluice Gates



Submerged Sluice Gates

$$Q = CWB\sqrt{2g3H}$$

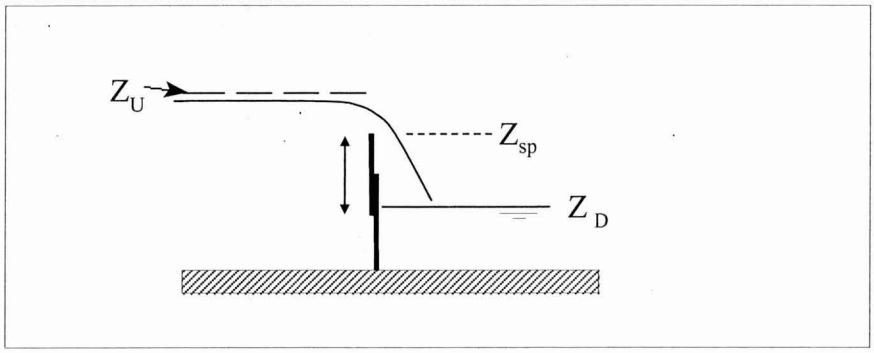
Where: $H = Z_U - Z_D$

Transitions to fully submerged Orifice flow at 0.8 submergence:

$$Q = CA\sqrt{2gH}$$

Submergence begins to occur when the tailwater depth above the spillway divided by the headwater energy above the spillway, is greater than 0.67. The above Equation is used to transition between free flow and fully submerged flow. This transition is set up so the program will gradually change to the fully submerged Orifice equation when the gates reach a submergence of 0.80.

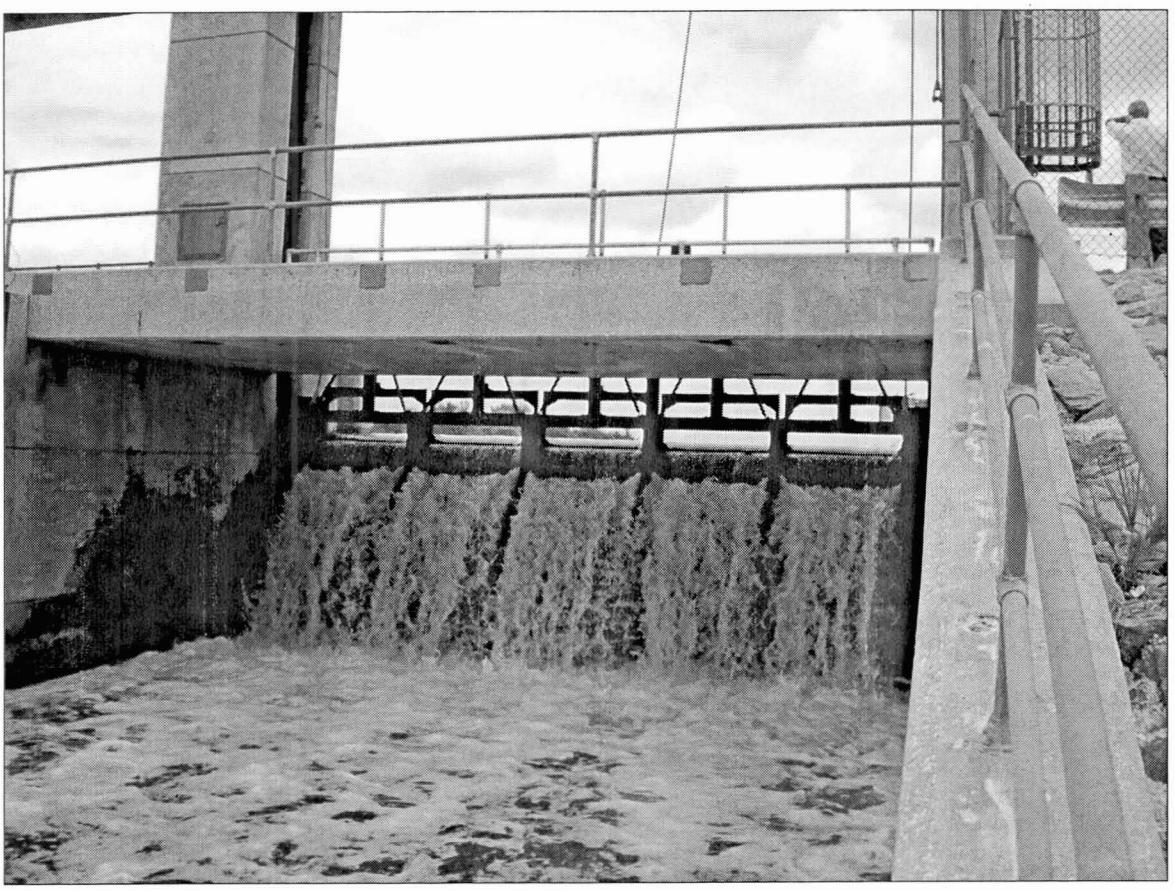
Overflow Gates



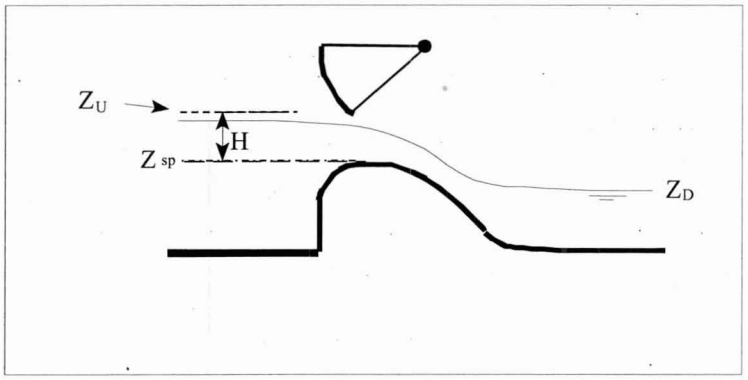
$$Q = CLH^{3/2}$$

Where: H = Upstream energy head ($Z_U - Z_{sp}$)

C = Coefficient of discharge, 3.1 – 3.3



Low Flow Through Gates



$$Q = C\,L\,H^{3/2}$$

When the upstream water surface is equal to or less than the top of the gate opening, the program calculates the flow through the gates as weir flow.

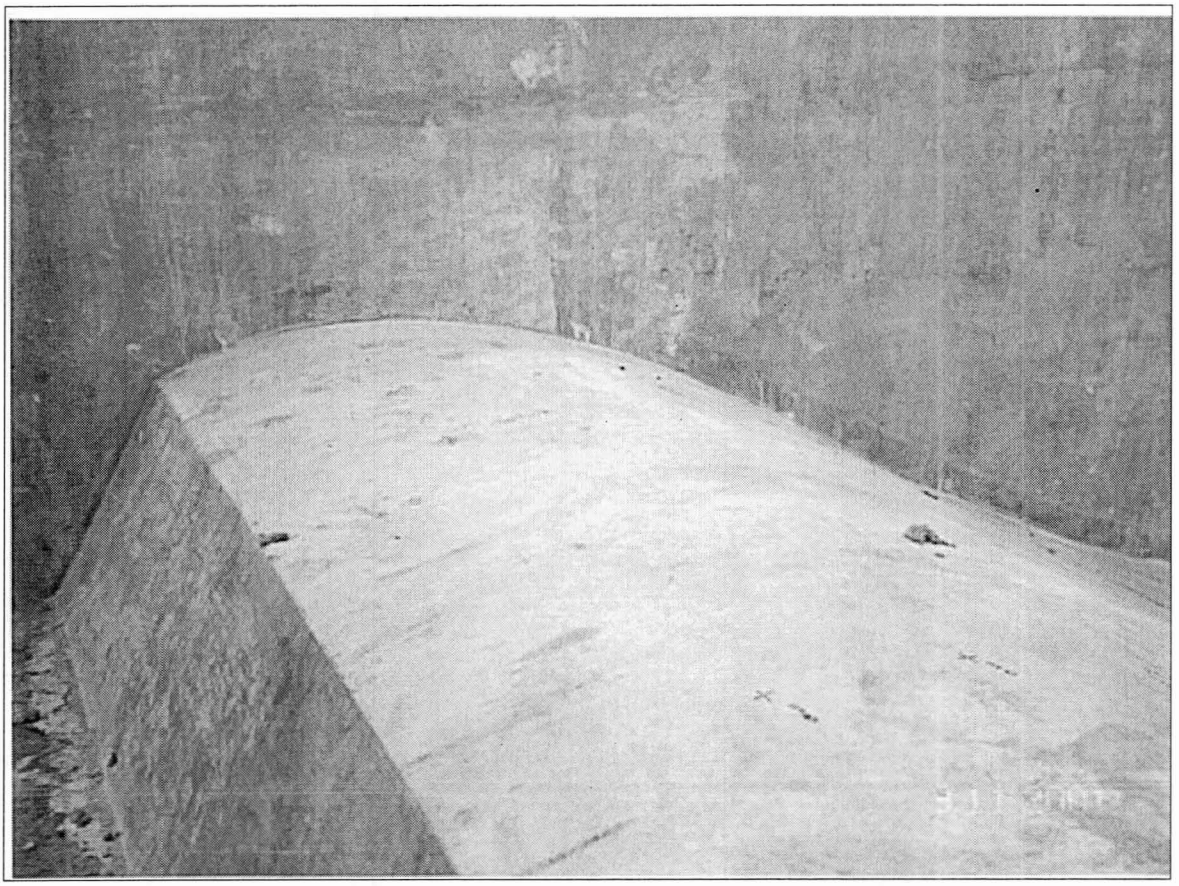
The user can specify either a broad crested or ogee weir shape for the spillway crest of the gate. If the crest of the spillway is ogee shaped, the weir coefficient will be automatically adjusted when the upstream energy head is higher or lower than a user specified design head. The adjustment is based on a curve (Bureau of Reclamation, 1977). The curve provides ratios for the discharge coefficient, based on the ratio of the actual head to the design head of the spillway.

The program automatically accounts for submergence on the weir when the tailwater is high enough to slow down the flow. Submergence is defined as the depth of water above the weir on the downstream side divided by the headwater energy depth of water above the weir on the upstream side. As the degree of submergence increases, the program reduces the weir flow coefficient. Submergence corrections are based on a trapezoidal (broad crested) or ogee shaped weir.

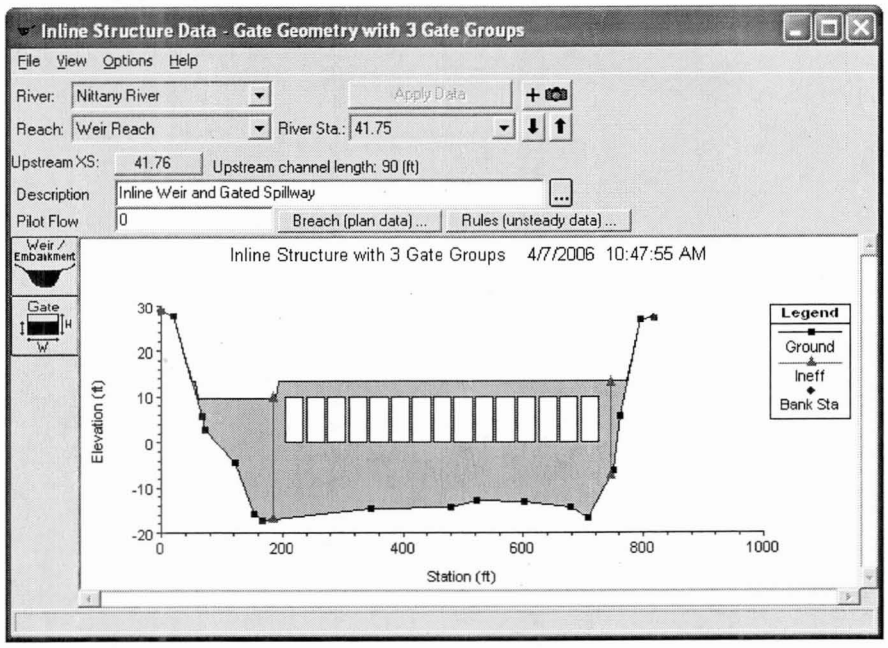
Uncontrolled Over-Flow Weirs

- Can represent Emergency Spillway or flow over entire Embankment.
- Uses Standard Weir Equation
- Can have Ogee, Broad Crested, or Sharp Crested weir shape
- Weir flow submergence is calculated

In addition to the gate openings, the user can define an uncontrolled overflow weir at the same river crossing. The weir could represent an emergency spillway or the entire top of the structure and embankment. Weir flow is computed using the standard weir equation. The uncontrolled overflow weir can be specified as either a broad crested or ogee shaped weir. If the weir is ogee shaped, the program will allow for fluctuations in the discharge coefficient to account for upstream energy heads that are either higher or lower than the design head. The program will automatically account for any submergence of the downstream tailwater on the weir, and reduce the flow over the weir. The modeler is referred to Chapter 5 of the Hydraulic Reference Manual for additional discussions concerning uncontrolled overflow weirs, including submergence criteria and selection of weir coefficients.

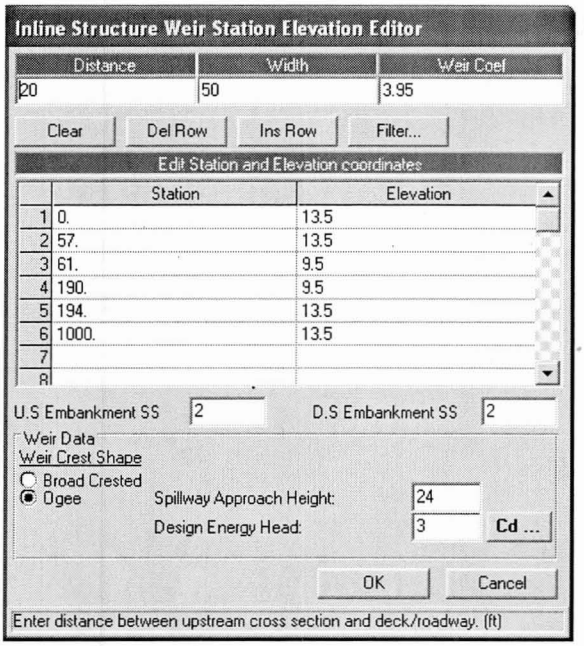


Entering Inline Structure Data



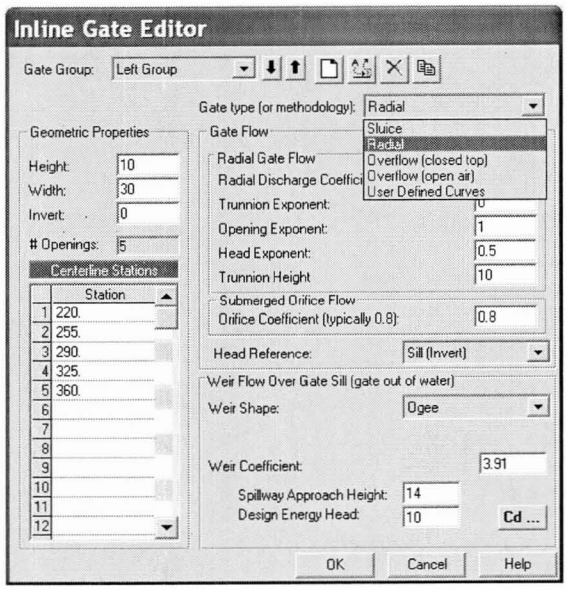
Inline structure data are entered in a similar manner as bridge and culvert data. To enter an structure press the **Inline Structure** button from the Geometric Data window.

Weir and Embankment Profile



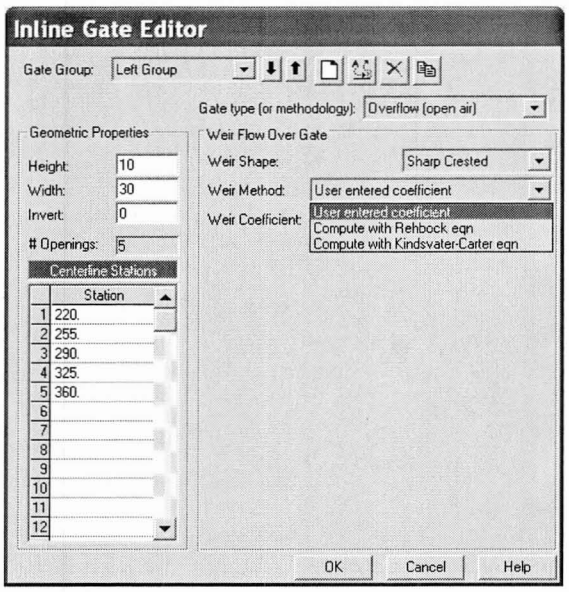
The Embankment and Weir data are entered together, and are used to describe the embankment blocking the stream as well as any uncontrolled weirs. To enter the weir and embankment data, press the **Weir/Embankment** button and the editor will appear.

Gate Editor



To add gated spillways to the structure, press the **Gate** button on the Inline structure data editor. Once this button is pressed, the gated editor will appear. The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. The functionality of the gates is controlled from the Unsteady Flow Data editor. Please see Chapter 6 of the User's Manual for a complete description of the data contained in this editor.

Overflow Gates Data



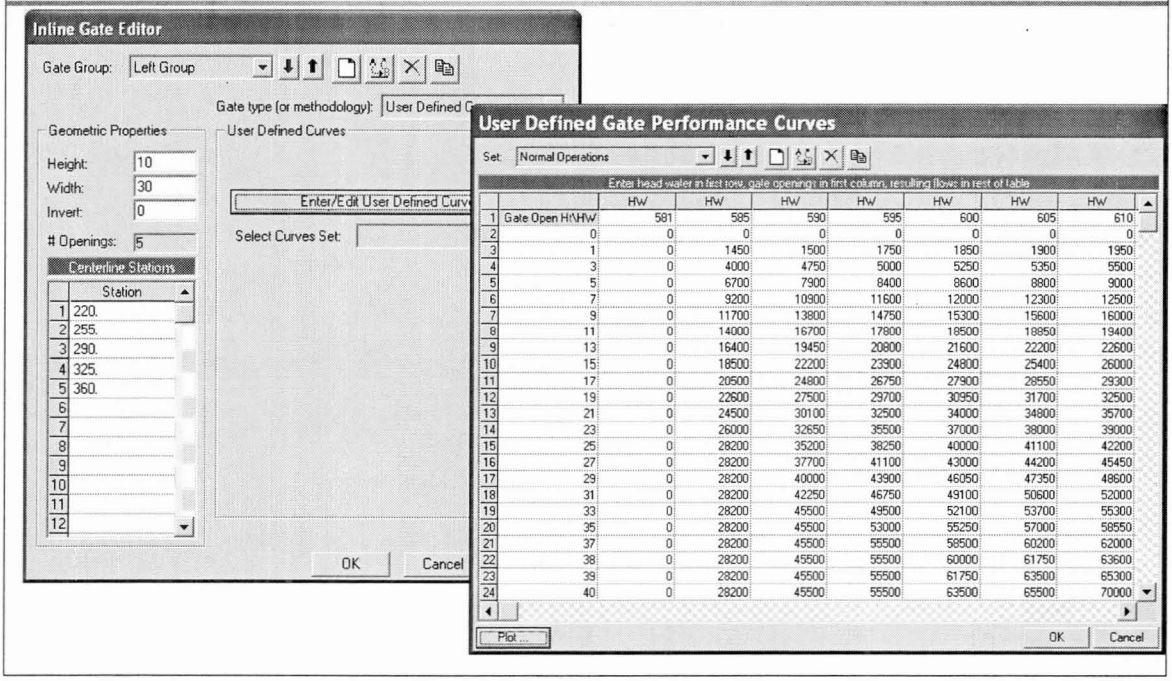
Rises and falls from the bottom rather than the top (as the sluice and radial options do)

The invert varies rather than the high chord

Can also be used for inflatable weirs and slat gates

Can use standard weir equation, Rehbock equation, or Kindsvater – Carter equation.

User Defined Curves

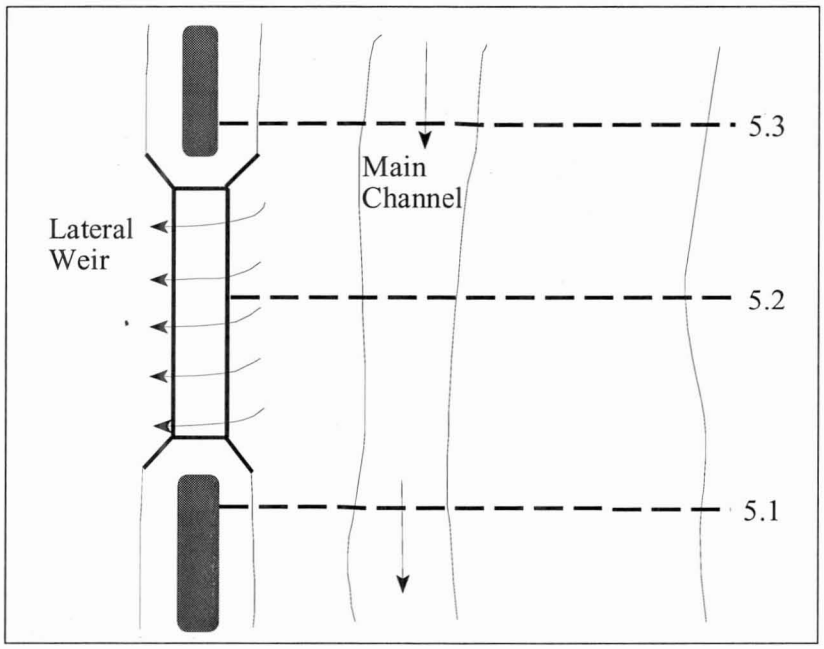


Lateral Structures

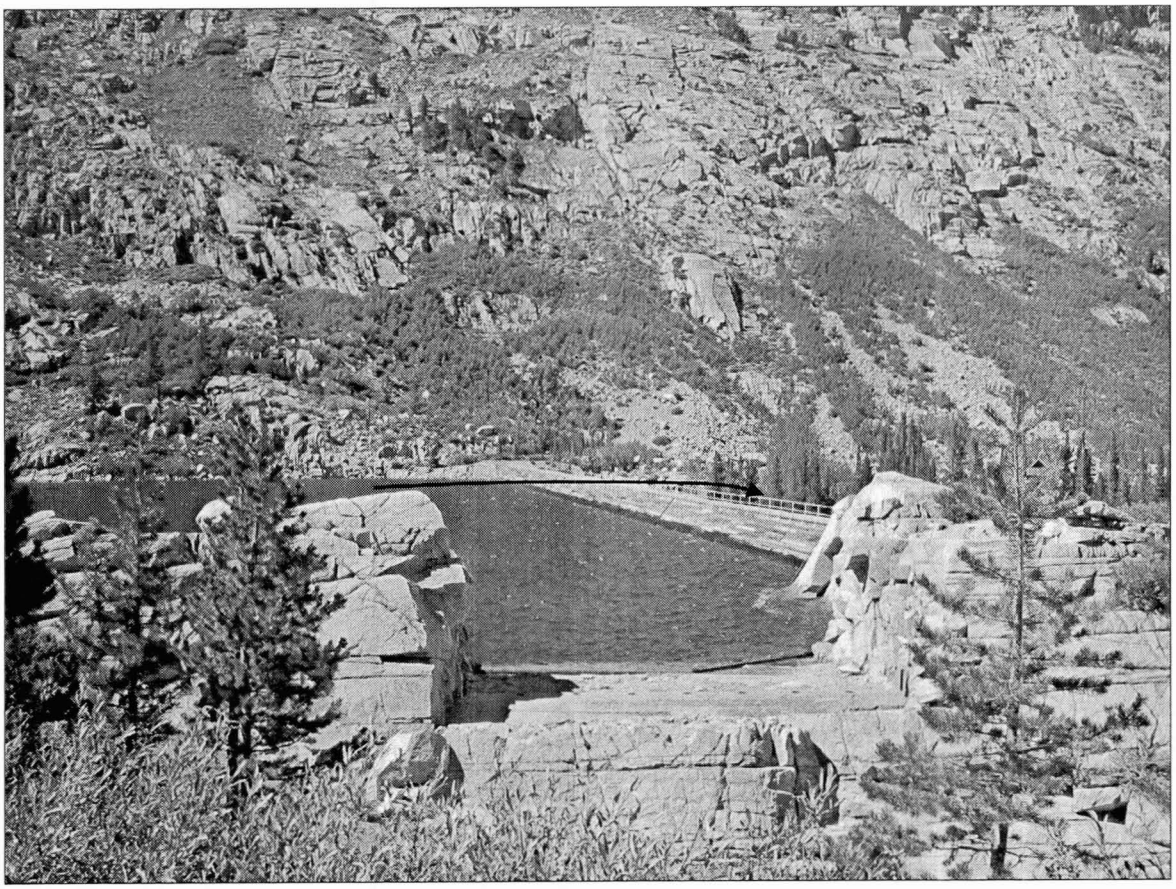
- Radial, Sluice, and Overflow Gates
- Culvert groups
- Uncontrolled Overflow Weir
- Rating Curves
- Same Hydraulic Equations for Gates as with Inline Structures
- Lateral structure Can handle a sloping water surface as well as irregular weir profile.

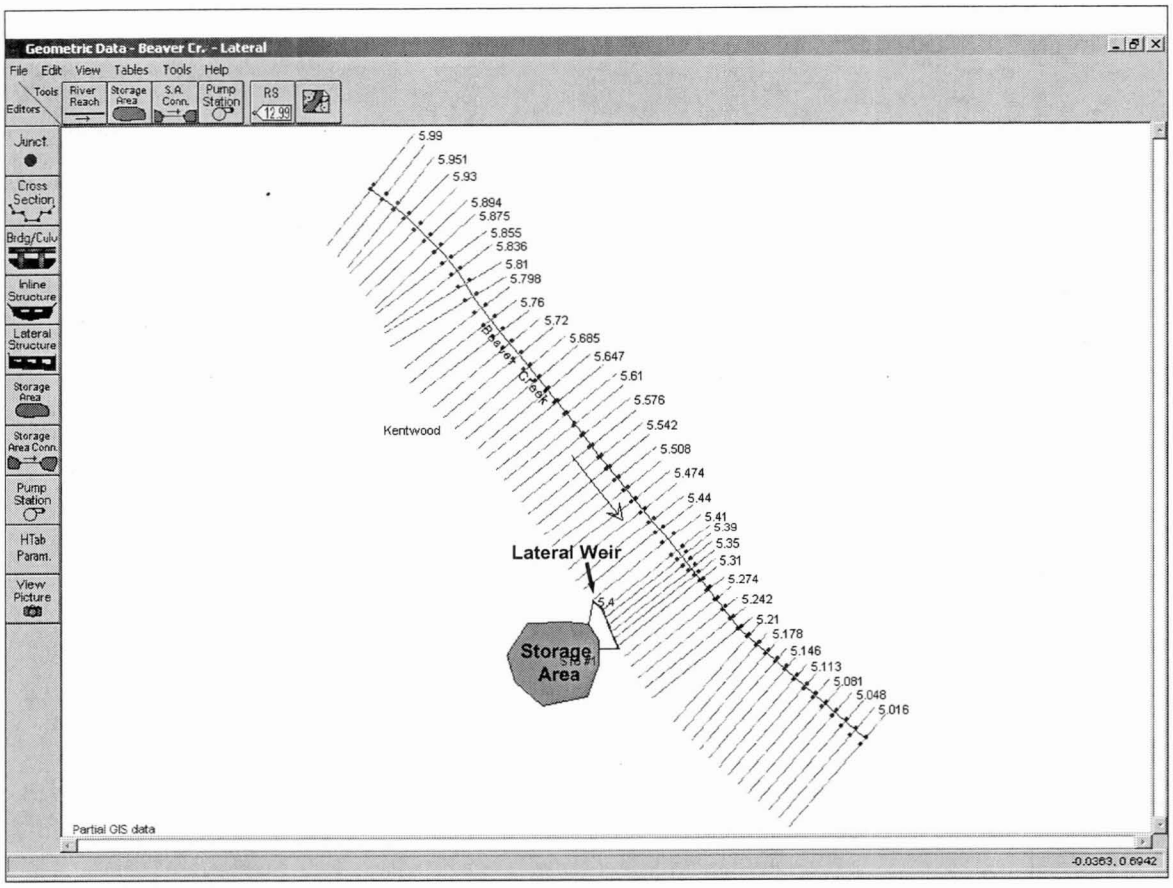
HEC-RAS has the ability to model lateral structures that contain overflow spillways, gated spillways, culverts, and diversion rating curve. The user can set up a lateral structure that is any combination of the available opening types. The gated spillways can have either radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape. Culverts can have flap gates, which will prevent flow from going in both directions. The user can specify which flow direction the culvert is limited to. A Diversion rating curve can also be added to the lateral structure in order to model a hydraulic element that is not available in HEC-RAS.

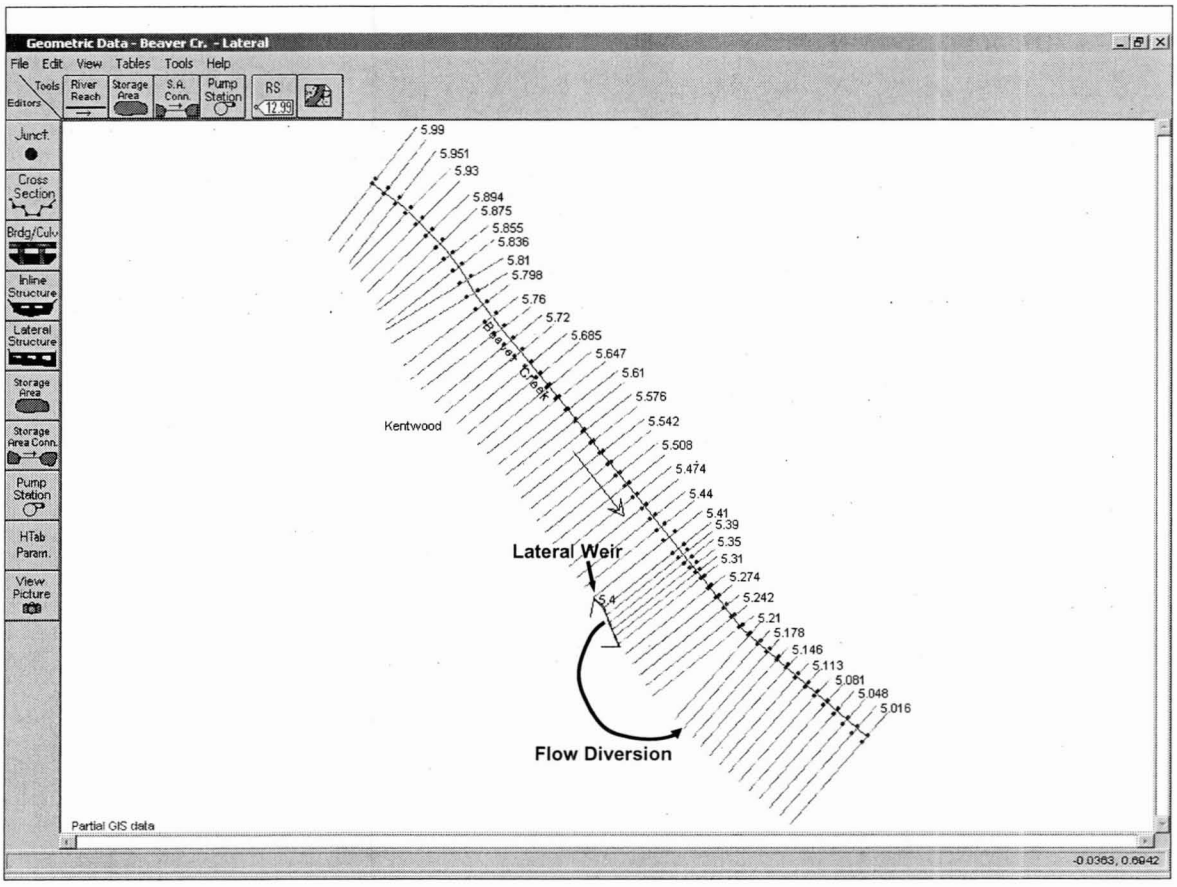
Cross Section Layout

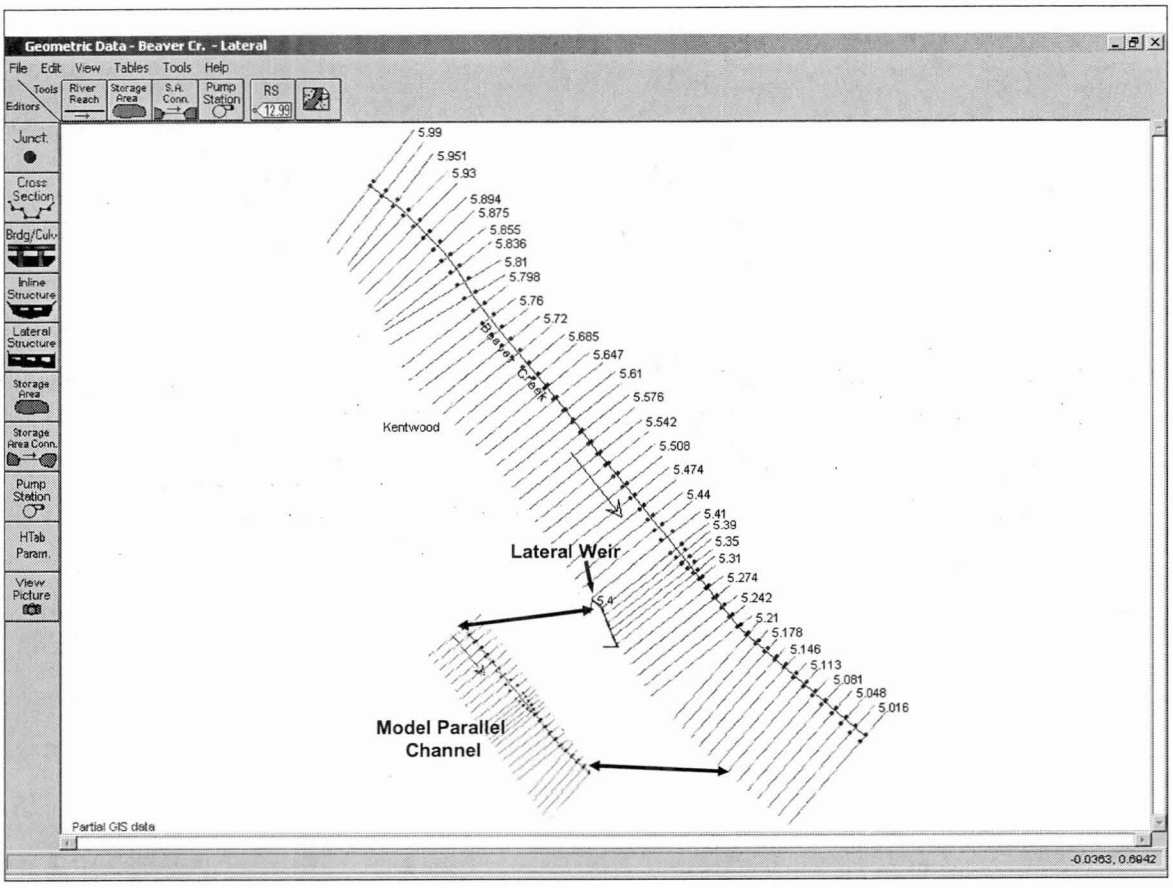


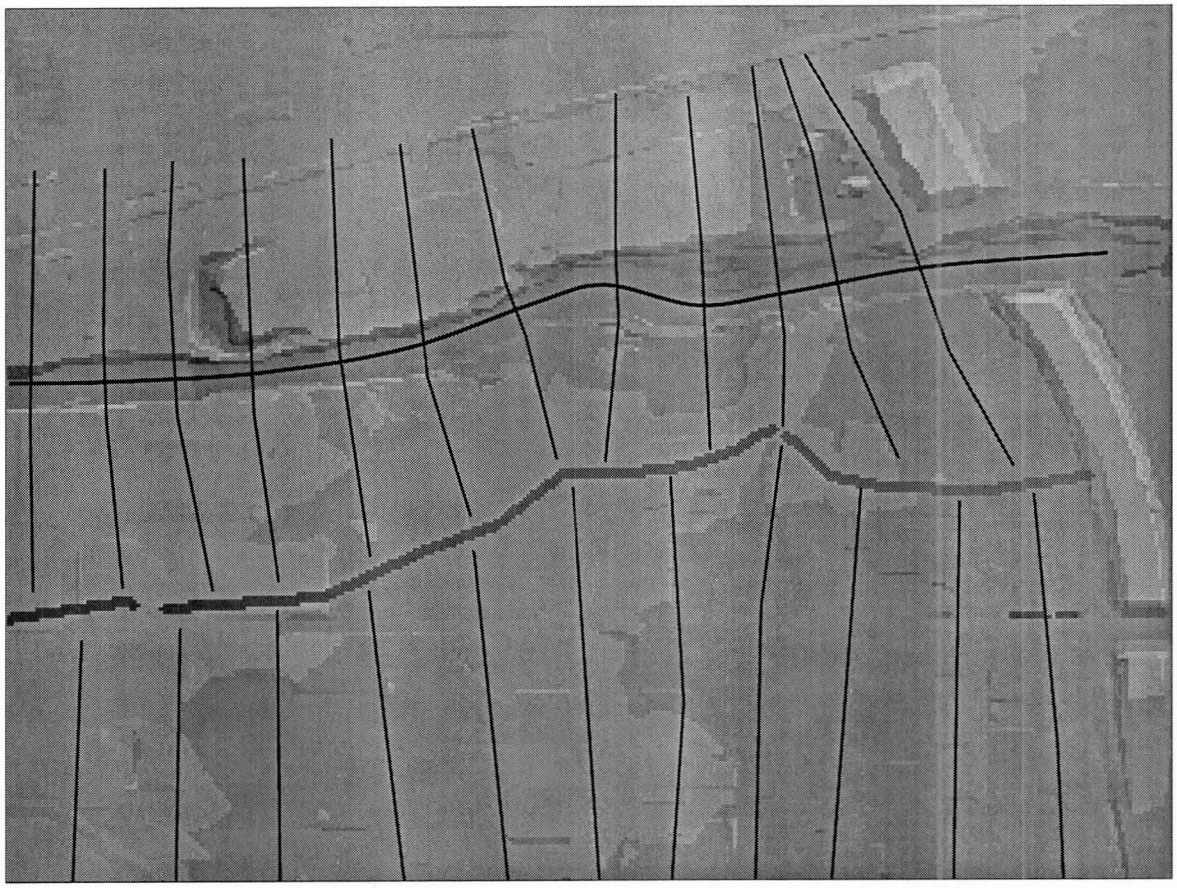
At a minimum there must be a cross section upstream of, and a cross section downstream of the lateral structure. The upstream cross section can either be right at the beginning of the structure, or it can be a short distance upstream. The downstream cross section can be right at the downstream end of the structure or it can be a short distance downstream. The user can have up to 98 additional cross sections in the middle of the structure. The river stationing of the structure must be based on the upstream end of the structure. For the example above, one could have assigned a river station for the lateral structure as 5.25.




Geometric Data - Beaver Cr. - Lateral
File Edit View Tables Tools Help
Tools
Editors
River Reach
Storage Area
S.A. Conn.
Pump Station
RS
12.99
Junct.
Cross Section
Brdg/Culv
Inline Structure
Lateral Structure
Storage Area
Storage Area Conn.
Pump Station
HTab Param.
View Picture
5.99
5.951
5.93
5.894
5.875
5.855
5.836
5.81
5.798
5.76
5.72
5.685
5.647
5.61
5.576
5.542
5.508
5.474
5.44
5.41
5.39
5.35
5.31
5.274
5.242
5.21
5.178
5.146
5.113
5.081
5.048
5.016
Beaver Creek
Kentwood
Lateral Weir
5.4
Storage Area
Partial GIS data
-0.0363, 0.6942

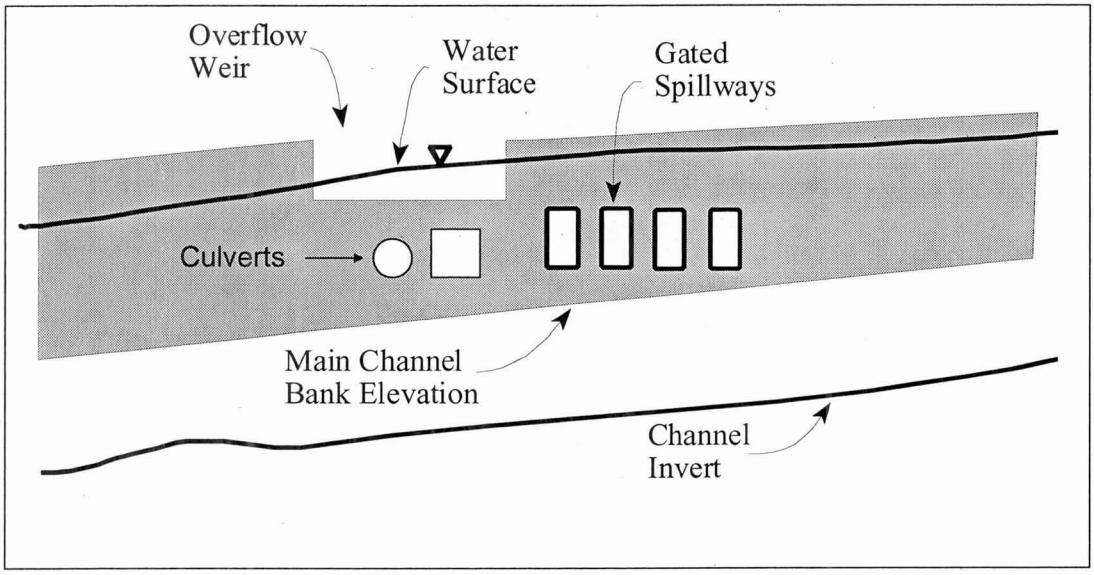

Geometric Data - Beaver Cr. - Lateral
File Edit View Tables Tools Help
Editors
Tools
River Reach
Storage Area
S.A. Conn.
Pump Station
RS
12.99
Junct.
Cross Section
Brdg/Culv
Inline Structure
Lateral Structure
Storage Area
Storage Area Conn.
Pump Station
HTab Param.
View Picture
5.99
5.951
5.93
5.894
5.875
5.855
5.836
5.81
5.798
5.76
5.72
5.685
5.647
5.61
5.576
5.542
5.508
5.474
5.44
5.41
5.39
5.35
5.31
5.274
5.242
5.21
5.178
5.146
5.113
5.081
5.048
5.016
Beaver Creek
Kentwood
Lateral Weir
5.4
Flow Diversion
Partial GIS data
-0.0363, 0.6942


Geometric Data - Beaver Cr. - Lateral
File Edit View Tables Tools Help
Tools
Editors
River Reach
Storage Area
S.A. Conn.
Pump Station
RS
12.99
Junct.
Cross Section
Brdg/Culv
Inline Structure
Lateral Structure
Storage Area
Storage Area Conn.
Pump Station
HTab Param.
View Picture
5.99
5.951
5.93
5.894
5.875
5.855
5.836
5.81
5.798
5.76
5.72
5.685
5.647
5.61
5.576
5.542
5.508
5.474
5.44
5.41
5.39
5.35
5.31
5.274
5.242
5.21
5.178
5.146
5.113
5.081
5.048
5.016
Beaver Creek
Kentwood
Lateral Weir
5.4
Model Parallel Channel
Partial GIS data
-0.0363, 0.6942



Lateral Weirs can be used to control the exchange of flow between channels in RAS and can be specified in Geo-RAS

Lateral Structure

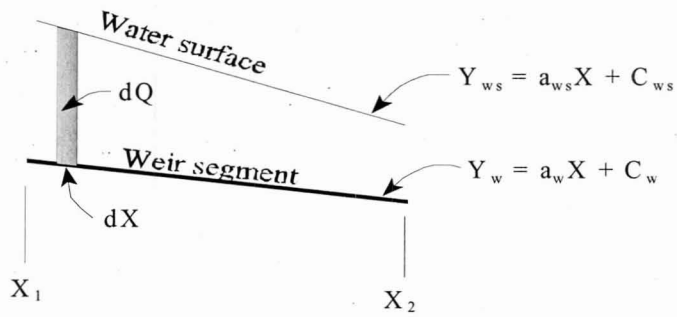


If there are gated openings in the structure, the hydraulic computations for lateral gated spillways are exactly the same as those described previously for inline gated spillways. The only difference is that the headwater energy is computed separately for each gate, based on its centerline location along the stream. The headwater energy for each gate is interpolated linearly between computed points at each cross section.

Culvert hydraulics are computed with the same principles and equations as inline culverts. The only difference is that the program interpolates on the water surface in the river to figure out the water surface at the centerline of each individual culvert.

The water surface across the weir has a slope to it. Additionally, the weir itself could be on a slope. Because of this, an equation for weir flow with a sloping water surface and weir sill had to be derived.

Lateral Weir Calculations



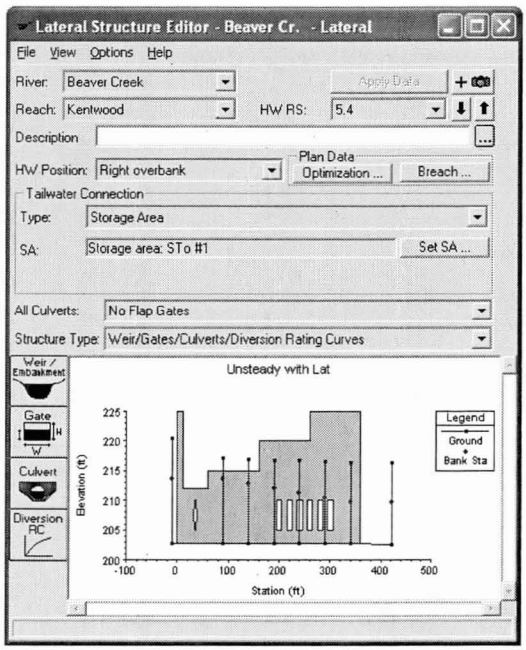
$$Q_{x_1-x_2} = \frac{2C}{5a_1}((a_1x_2+C_1)^{5/2} - (a_1x_1+C_1)^{5/2})$$

The equation for a sloping line representing the water surface and the weir segment are shown. The constants a_{ws} and a_w represent the slope of the water surface and the weir segment, respectively, while the variable C_{ws} and C_w are constants representing the initial elevations.

The standard weir equation assumes that the weir is parallel with the water surface (i.e. that the depth of water is constant from one end of the weir segment to the other). The above general equation is derived for a sloping weir and water surface by integrating the standard weir equation. The above equation is valid as long as a_1 is not zero. When a_1 is zero, this implies that the water surface and the weir segment are parallel. When this is true, the original weir equation is used.

Within HEC-RAS, flow over a lateral weir can be computed from either the energy grade line or the water surface elevation. The standard weir equation is derived with the upstream energy head being based on the distance from the weir sill to the upstream energy gradeline. The water surface is the default for a lateral weir computation. However, the user has the option of instructing the program to use the energy gradeline elevation when computing the head term of the weir equation. Using the water surface would be most appropriate when the weir is located close to the main channel. In this situation the energy due to the velocity head is in the downstream direction, and not over the top of the lateral weir. Therefore, the computation of the energy head over the lateral weir is best depicted by using the water surface of the flow in the channel.

Entering Lateral Structure Data

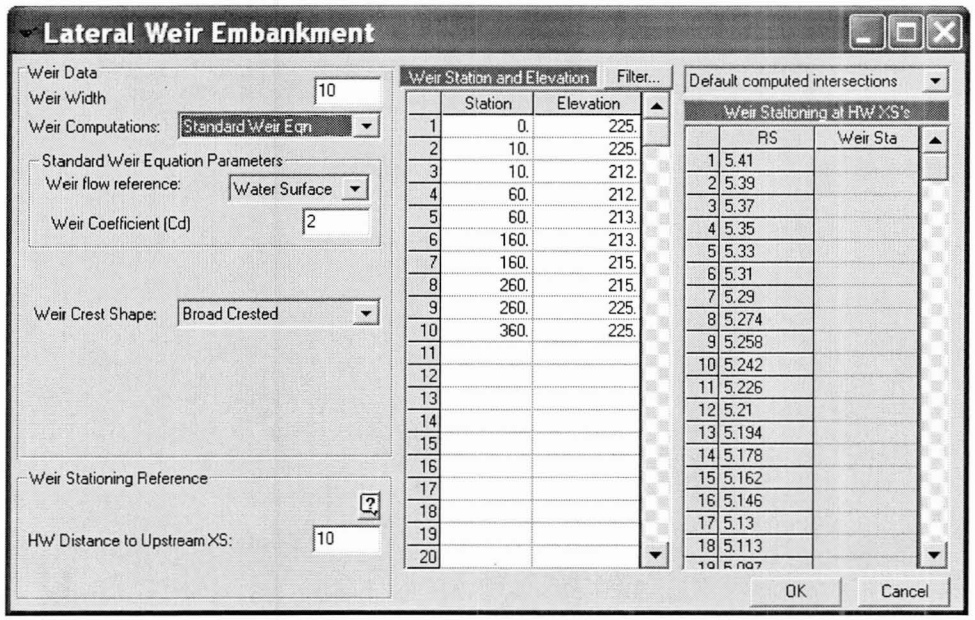


Lateral structure data are entered in a similar manner as bridge and culvert data. To enter data for a lateral structure press the **Lateral Structure** button from the Geometric Data window. Once this button is pressed, the Lateral Structure Data editor will appear.

The user selects "Add Lateral Structure" from the Options menu. A river station just downstream of the cross section upstream of the beginning of the lateral structure should be used to define the location of the structure. A position of the structure should be entered. This tells the program if the structure is located at the left overbank, next to the main channel bank on the left or right side, or in the right overbank. The user must then connect the lateral structure to either a storage area or another river reach.

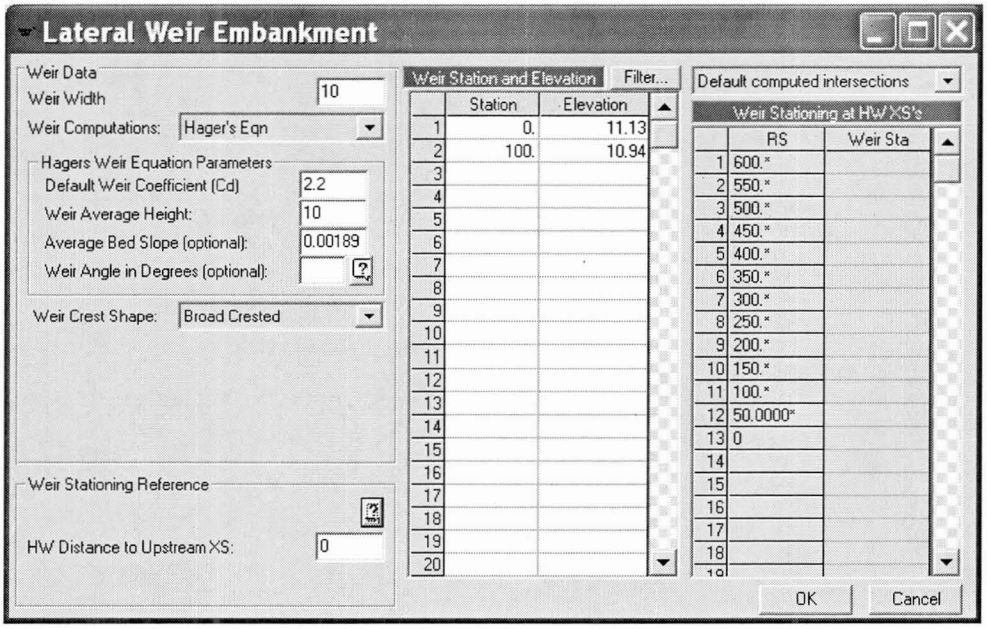
Additional options for the lateral hydraulic structure are to set the routing type and flap gate control for culverts. The routing type can be either "Weir/Gates/Culverts/Diversion Rating Curve" or "Linear Routing". The linear routing option is used to specify a simple storgage routing option using a linear reservoir routing method. The user enters an elevation for an assumed weir and linear routing coefficient. The flap gate option allows the user to control the direction of flow through the culverts. The default is no flap gates, and flow can go in both directions. If the option for "No negative flow flap" is selected, this means that flow can only go through the culverts in the positive flow direction, which is away from the river. If the option "No positive flow flap" is selected, then no flow will go away from the river through the culvert, but it will be allowed to come back out once the river stage is lower than the stage that the lateral structure is connected to. The rating curve option allows the user to specify an elevation versus flow relationship for taking

Lateral Weir Embankment Data



The Lateral Weir/Embankment Data editor is similar to the Deck/Roadway editor for bridges and culverts. The user enters a distance between the upstream end of the structure and the upstream cross section, the lateral width of the structure (used for drawing purposes only), weir flow reference (to use the water surface or energy gradeline to calculate the lateral weir flow), the weir flow coefficient, weir shape, and the station and elevation data describing the embankment and weir profile. Additionally, the user can select between the Standard Weir equation and Hager's Lateral Weir equation (Weir Computations drop down box).

Hager's Lateral Weir Equation



If Hager's Lateral weir equation is selected from the "Weir Computations" field, then the following additional fields will appear:

Default Weir Coefficient (Cd) – This weir coefficient will be used for the first iteration of trying the Hager lateral weir equation. The equation is iterative, and requires hydraulic results in order to make a weir coefficient calculation. The default weir coefficient is only used for the first guess at the hydraulic computations.

Weir Average Height – This field is used for entering the average height of the weir above the ground.

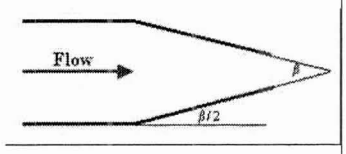
Average Bed Slope (Optional) – This field is used for entering the average slope of the stream bed in the reach of river that contains the lateral weir. If the user does not enter this field, the HEC-RAS program will compute the slope by estimating an average bed elevation for each cross section, then computing the slope of the average bed elevation. Average bed elevation of an irregular cross section is obtained by subtracting hydraulic depth from the water surface elevation.

Weir Angle in Degrees (Optional) – This field can be used to enter and angle for the weir. If the weir is parallel to the stream, the angle is assumed to be zero. If the weir is angled inwards towards the center of the river, an angle (beta) is required. This is used for channels that have a contraction, and weir flow is allowed to go over the contracted section.

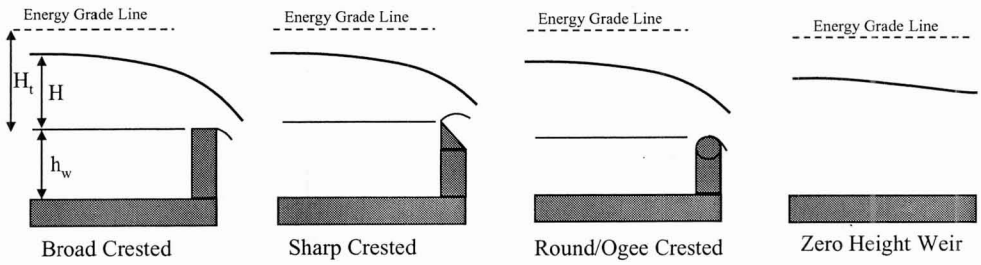
Hager's Lateral Weir Equation - Continued

$Q = CLH^{3/2}$

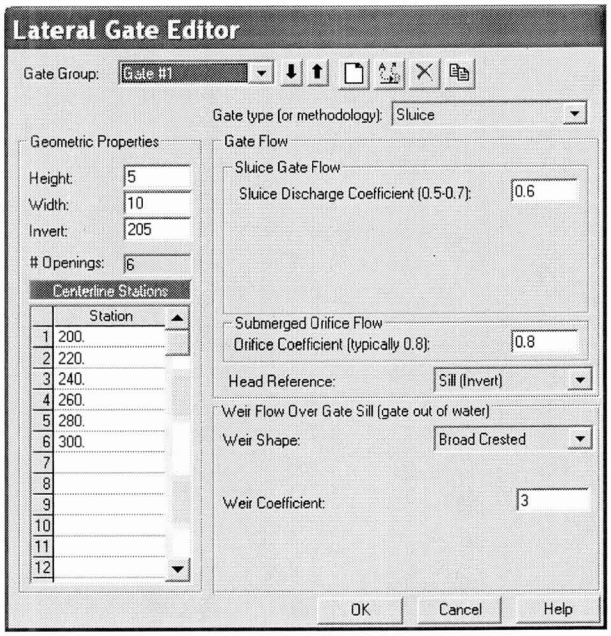
$$C = \frac{3}{5} C_0 \sqrt{g} \left[\frac{1-W}{3-2\nu-W} \right]^{0.5} \left\{ 1-(\beta+S_0) \left[\frac{3(1-\nu)}{\nu-W} \right]^{0.5} \right\}$$



$$W = \frac{h_w}{H_t + h_w} \qquad \nu = \frac{H + h_w}{H_t + h_w} \qquad C_0 = Function(weir\ shape)$$

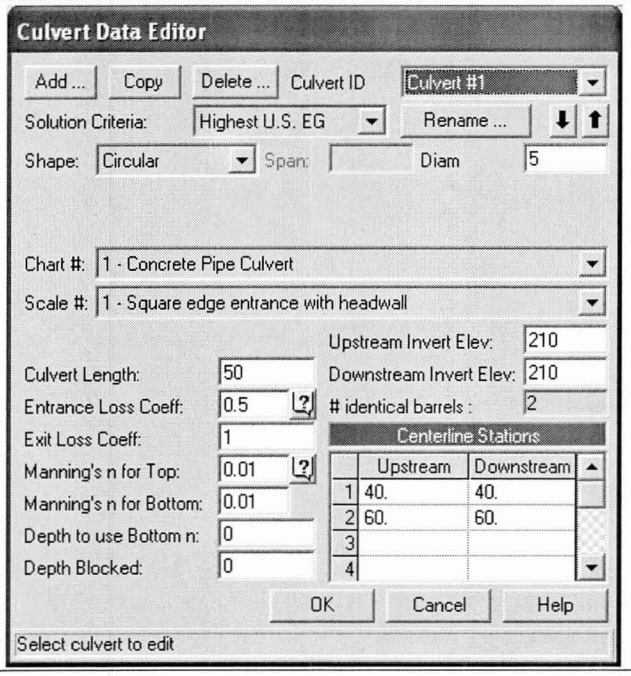


Lateral Gate Editor



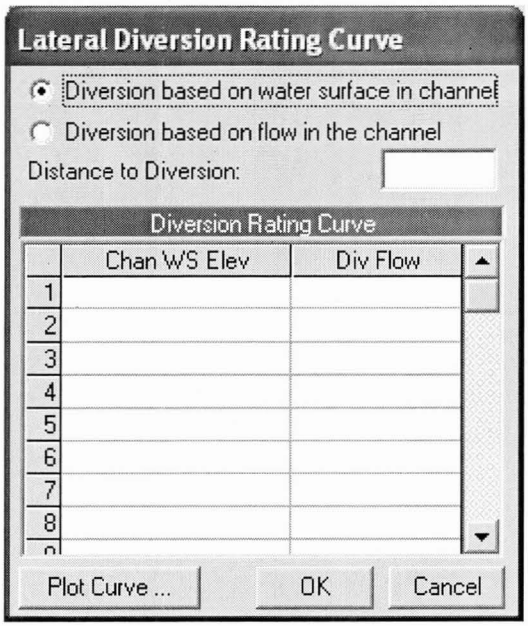
The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. Controlling the gate openings is accomplished from the Unsteady Flow Data editor. Please see chapters 6 and 8 of the user's manual for a complete description of this data.

Lateral Culvert Editor



This editor is almost identical to the normal inline culvert editor. All of the culvert shapes are available.

Lateral Rating Curve

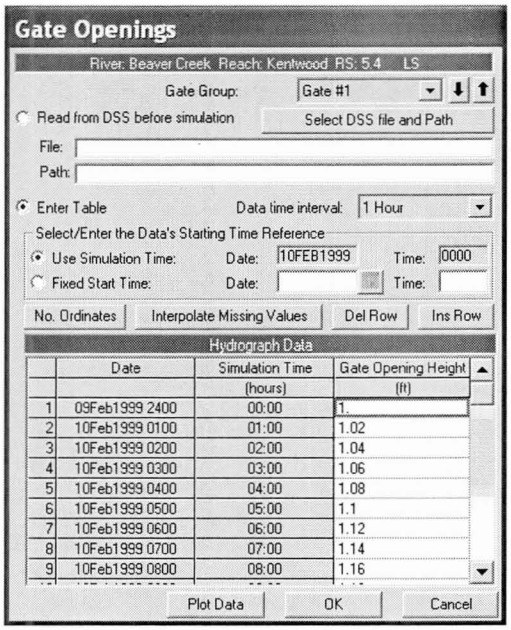


Controlling Inline and Lateral Gates

- For unsteady flow simulations the following gate controls are available from the HEC-RAS unsteady flow data editor:
  - Time Series of Gate Openings
  - Elevation Controlled Gates
  - Navigation Dam Control
  - Rules Editor

Controlling gate openings is an event specific operation. For this reason, the options for controlling the gate setting are contained in the Unsteady Flow Data Editor.

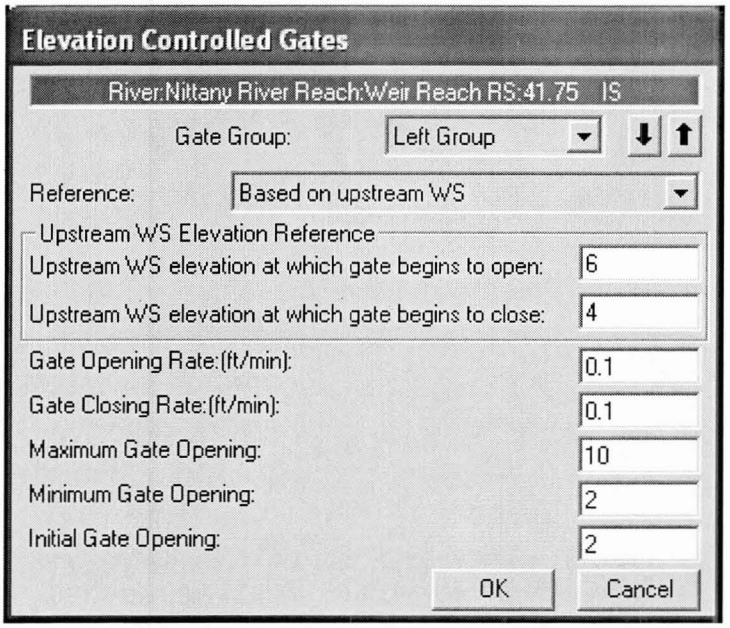
Time Series of Gate Openings



This option allows the user to enter a time series of gate openings for an inline gated spillway, lateral gated spillway, or a spillway connecting two storage areas. The user has the option of reading the data from a DSS file or entering the data into a table from within the editor.

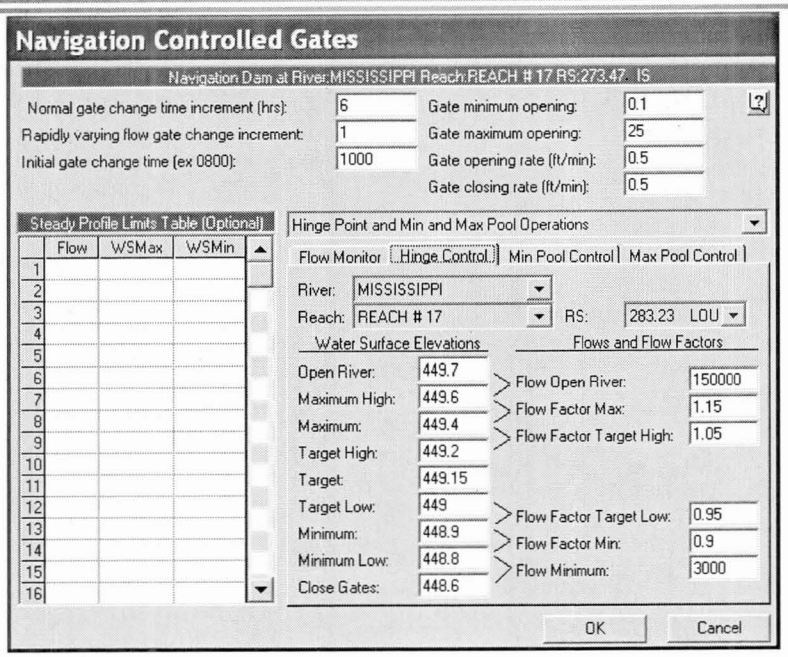
Warning: Opening and closing gates to quickly can cause instabilities in the solution of the unsteady flow equations. If instabilities occur near gated locations, the user should either reduce the computational time step and/or reduce the rate at which gates are opened or closed.

Elevation Controlled Gates



This option allows the user to control the opening and closing of gates based on the elevation of the water surface upstream of the structure. A gate begins to open when a user specified elevation is exceeded. The gate opens at a rate specified by the user. As the water surface goes down, the gate will begin to close at a user specified elevation. The closing of the gate is at a user specified rate (feet/min.). The user must also enter a maximum and minimum gate opening, as well as the initial gate opening.

Navigation Dams



The Navigation Dam option is used to have the program figure out appropriate gate settings to maintain specified water surface elevation within the Dam pool or an upstream hinge point location. This is a new feature for version 3.1.

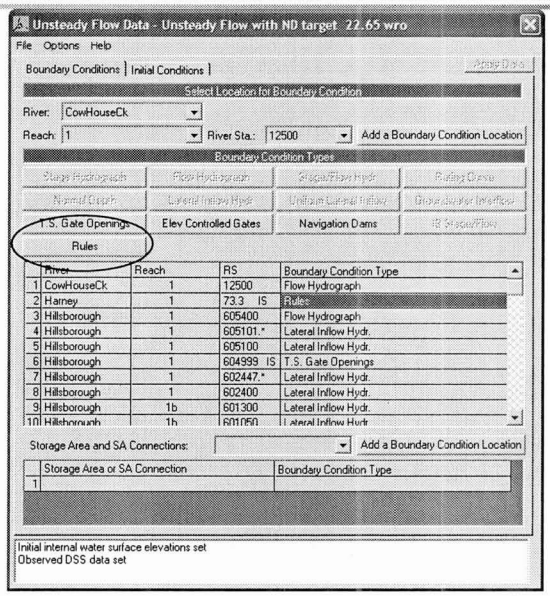
Operation Rules for Gated Structures

- Unsteady Flow Editor "Rules" boundary condition
- Inline/Lateral Structures
- Storage Area Connections

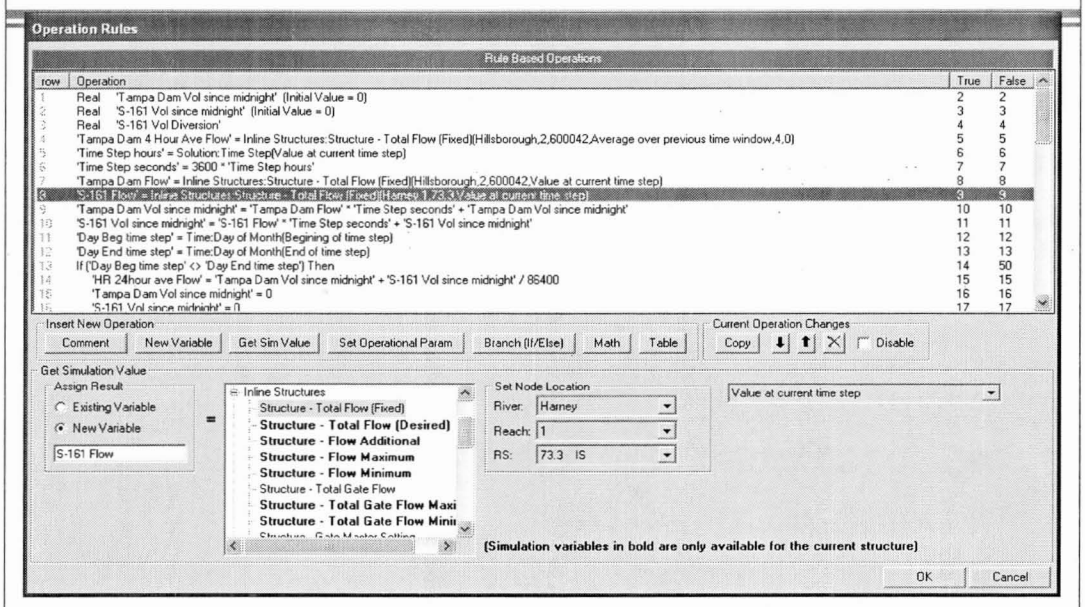
Controls

- Gates
- Weir Coefficients
- Min/Max Flow

- Rules are evaluated at every time step



User Defined Rules Editor for Operating Gated Structures

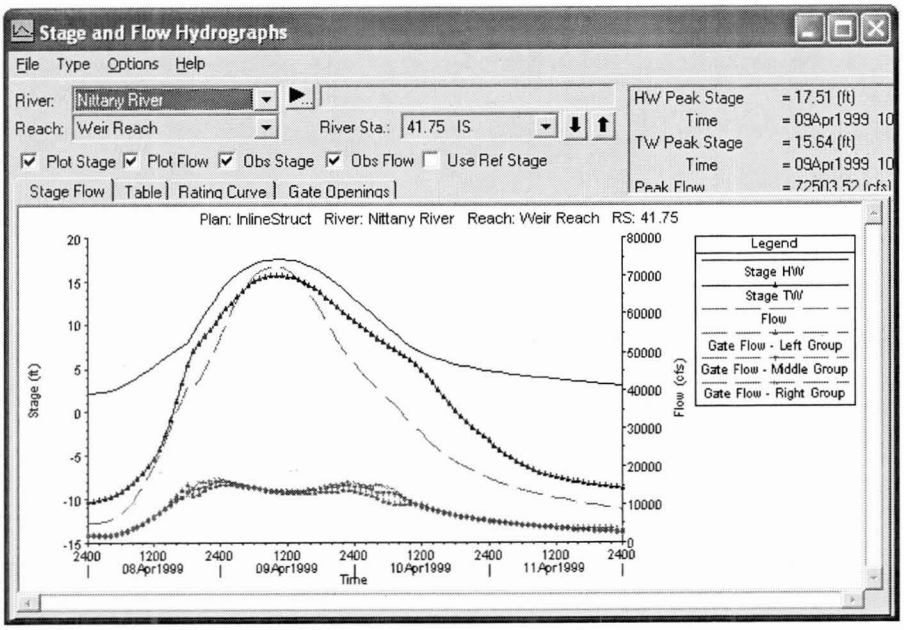


The operating procedures for determining and controlling the releases from reservoirs and other types of hydraulic structures can be quite complex. HEC-RAS allows flexibility in modeling and controlling the operations of gates at hydraulic structures through the use of rules (Figure above). Examples of variables that could be used to control releases from a hydraulic structure are: current flows and water surfaces at the structure, current flows and stages at a downstream or upstream cross section location, time considerations (winter, morning, etc), and/or previously computed values (accumulated outflows, running averages, etc). Rule operations in HEC-RAS are available for inline hydraulic structures, lateral hydraulic structures, and storage area connections that contain gates.

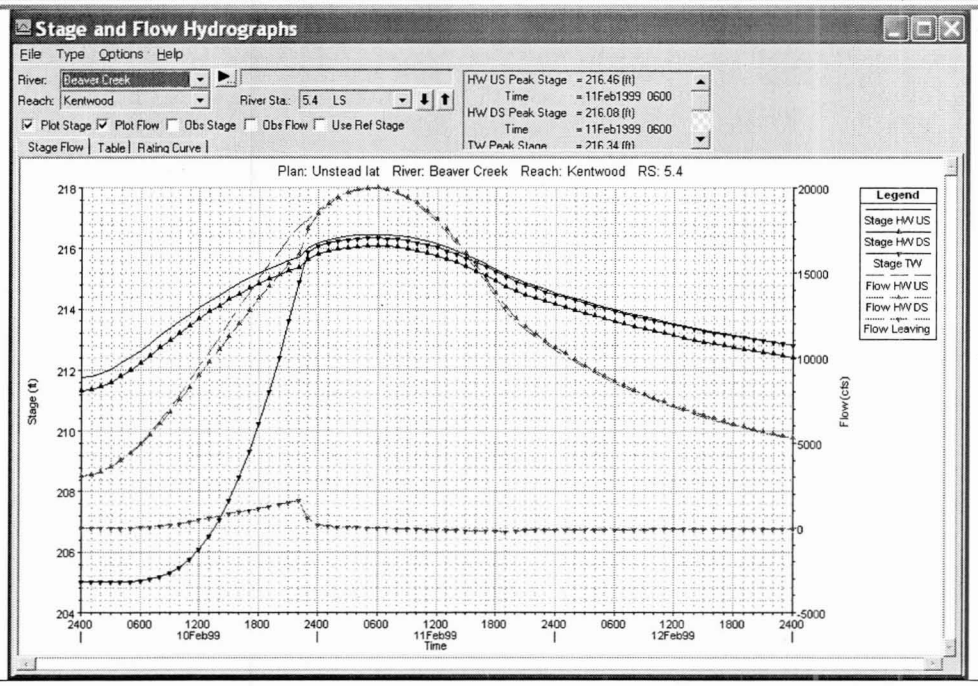
Inline and Lateral Structure Output

- The following types of output are available for inline and lateral spillways/weirs:
  - Stage and Flow Hydrograph Plots
  - Time Series Tables
  - Profile Plots
  - Cross Section Plots (Inline Structures only)
  - Detailed Output Tables
  - Profile Summary Tables

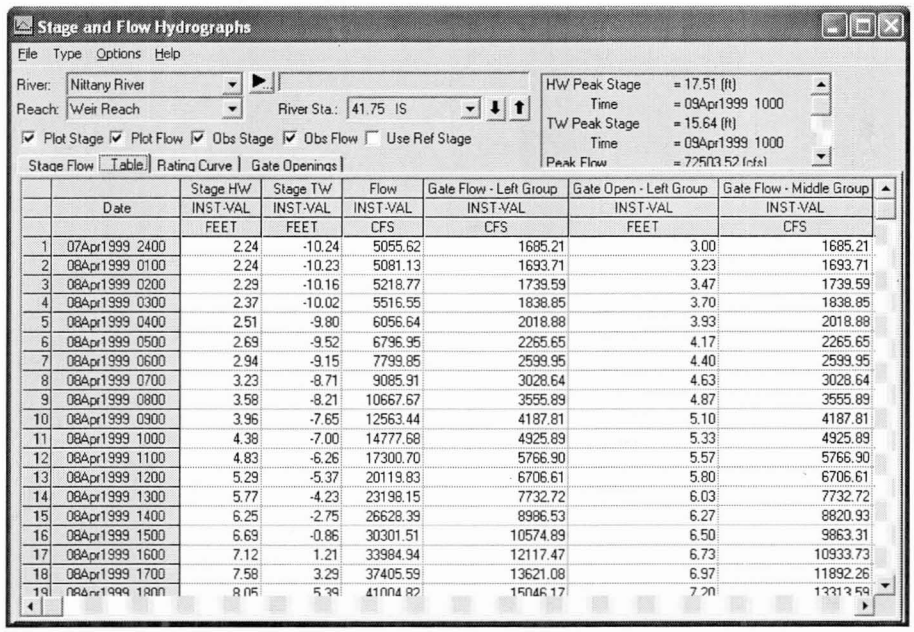
Example Stage and Flow Hydrographs for an Inline Structure



Stage and Flow Hydrograph for Lateral Structure

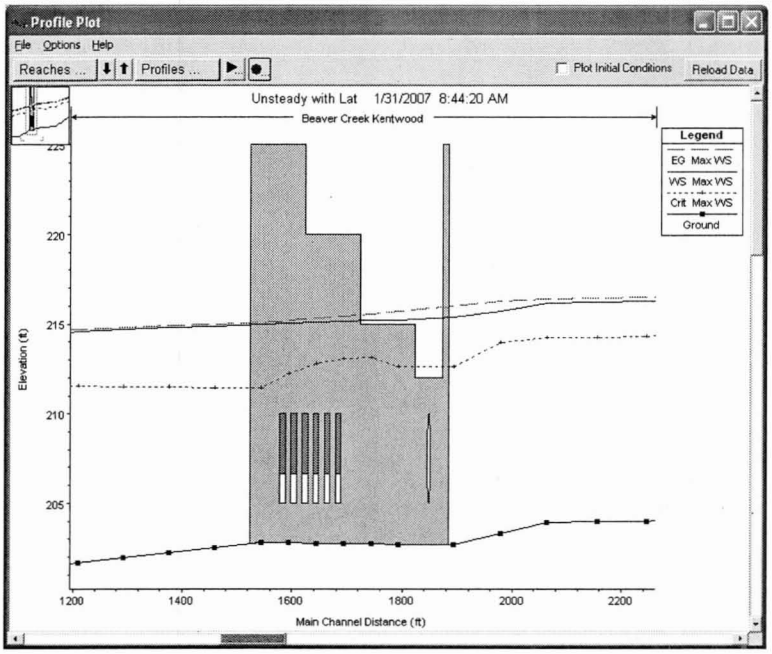


Time Series Table for an Inline Structure

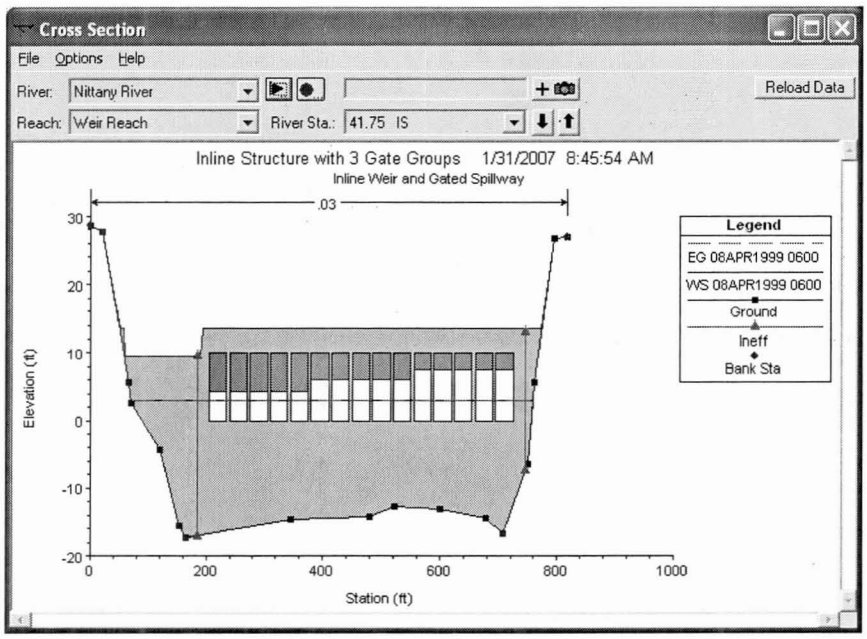


	Date	Stage HW INST-VAL FEET	Stage TW INST-VAL FEET	Flow INST-VAL CFS	Gate Flow - Left Group INST-VAL CFS	Gate Open - Left Group INST-VAL FEET	Gate Flow - Middle Group INST-VAL CFS
1	07Apr1999 2400	2.24	-10.24	5055.62	1685.21	3.00	1685.21
2	08Apr1999 0100	2.24	-10.23	5081.13	1693.71	3.23	1693.71
3	08Apr1999 0200	2.29	-10.16	5218.77	1739.59	3.47	1739.59
4	08Apr1999 0300	2.37	-10.02	5516.55	1838.85	3.70	1838.85
5	08Apr1999 0400	2.51	-9.80	6056.64	2018.88	3.93	2018.88
6	08Apr1999 0500	2.69	-9.52	6796.95	2265.65	4.17	2265.65
7	08Apr1999 0600	2.94	-9.15	7799.85	2599.95	4.40	2599.95
8	08Apr1999 0700	3.23	-8.71	9085.91	3028.64	4.63	3028.64
9	08Apr1999 0800	3.58	-8.21	10667.67	3555.89	4.87	3555.89
10	08Apr1999 0900	3.96	-7.65	12563.44	4187.81	5.10	4187.81
11	08Apr1999 1000	4.38	-7.00	14777.68	4925.89	5.33	4925.89
12	08Apr1999 1100	4.83	-6.26	17300.70	5766.90	5.57	5766.90
13	08Apr1999 1200	5.29	-5.37	20119.83	6706.61	5.80	6706.61
14	08Apr1999 1300	5.77	-4.23	23198.15	7732.72	6.03	7732.72
15	08Apr1999 1400	6.25	-2.75	26628.39	8986.53	6.27	8820.93
16	08Apr1999 1500	6.69	-0.86	30301.51	10574.89	6.50	9863.31
17	08Apr1999 1600	7.12	1.21	33984.94	12117.47	6.73	10933.73
18	08Apr1999 1700	7.58	3.29	37405.59	13621.08	6.97	11892.26
19	08Apr1999 1800	8.05	5.39	41004.82	15046.17	7.20	13313.59

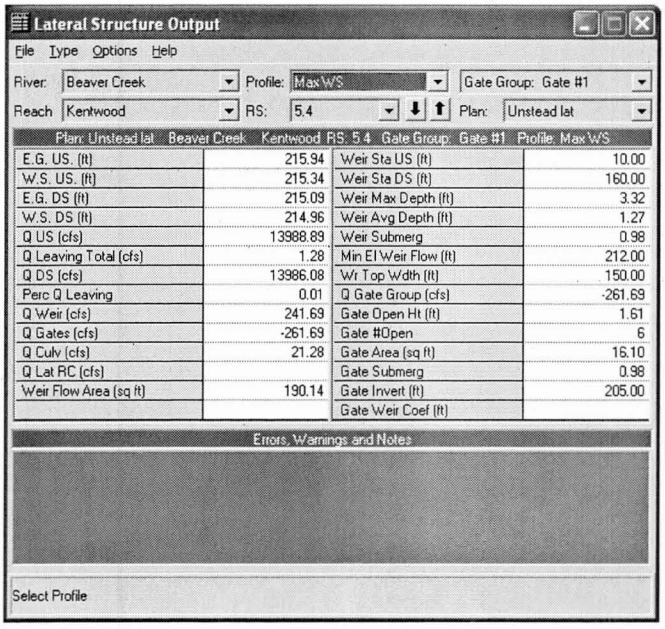
Example Profile Plot With a Lateral Structure



Example Cross Section Plot of Inline Structure



Example Detailed Output Table For a Lateral Structure



Lateral Structure Output

File Type Options Help

River: Beaver Creek | Profile: Max WS | Gate Group: Gate #1

Reach: Kentwood | RS: 5.4 | Plan: Unstead lat

Plan: Unstead lat Beaver Creek Kentwood RS: 5.4 Gate Group: Gate #1 Profile: Max WS

E.G. US. (ft)	215.94	Weir Sta US (ft)	10.00
W.S. US. (ft)	215.34	Weir Sta DS (ft)	160.00
E.G. DS (ft)	215.09	Weir Max Depth (ft)	3.32
W.S. DS (ft)	214.96	Weir Avg Depth (ft)	1.27
Q US (cfs)	13988.89	Weir Submerg	0.98
Q Leaving Total (cfs)	1.28	Min El Weir Flow (ft)	212.00
Q DS (cfs)	13986.08	Wr Top Wdth (ft)	150.00
Perc Q Leaving	0.01	Q Gate Group (cfs)	-261.69
Q Weir (cfs)	241.69	Gate Open Ht (ft)	1.61
Q Gates (cfs)	-261.69	Gate #Open	6
Q Culv (cfs)	21.28	Gate Area (sq ft)	16.10
Q Lat RC (cfs)		Gate Submerg	0.98
Weir Flow Area (sq ft)	190.14	Gate Invert (ft)	205.00
		Gate Weir Coef (ft)	

Errors, Warnings and Notes

Select Profile

Example Profile Output Table for Lateral and Inline Structures

Profile Output Table - Lateral Structures

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: Unsteadlat River: Beaver Creek Reach: Kentwood — Reload Data

Reach	River Sta	Profile	Q US	Q Leaving Total	Q DS	Q Weir	Q Gates	Wr Top Wdth	Weir Max Depth	Weir Avg Depth	Min El Weir Flow	E.G. US.	W.S. US.	E.G. DS	W.S. DS
			(cfs)	(cfs)	(cfs)	(cfs)	(cfs)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)
Kentwood	5.4	Max WS	13988.89	1.28	13986.08	241.69	-261.69	150.00	3.32	1.27	212.00	215.94	215.34	215.69	214.96
Kentwood	5.4	09FEB1999 2400	5043.50	972.89	4087.22	16.20	751.15	50.00	0.32	0.30	212.00	212.66	212.33	211.93	211.87
Kentwood	5.4	10FEB1999 0600	6055.50	854.02	5176.49	62.06	610.98	50.00	0.75	0.73	212.00	213.16	212.76	212.40	212.34
Kentwood	5.4	10FEB1999 1200	8428.90	281.79	8121.03	224.96	-4.14	50.00	1.78	1.75	212.00	214.26	213.79	213.50	213.42
Kentwood	5.4	10FEB1999 1800	11215.39	161.12	11032.87	298.56	-176.38	50.00	2.62	2.59	212.00	215.18	214.63	214.37	214.26
Kentwood	5.4	10FEB1999 2400	13334.88	75.17	13247.81	282.50	-232.03	150.00	3.16	1.10	212.00	215.77	215.17	214.92	214.80
Kentwood	5.4	11FEB1999 0600	13979.87	-1.53	13981.65	239.60	-262.21	150.00	3.32	1.27	212.00	215.94	215.34	215.09	214.96
Kentwood	5.4	11FEB1999 1200	12914.15	-74.07	13000.50	203.44	-293.81	150.00	3.12	1.07	212.00	215.70	215.13	214.88	214.75

Flow in cross section upstream of a lateral structure.

Profile Output Table - Inline Structure

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: InlineStruct River: Nittany River Reach: Weir Reach — Reload Data

Reach	River Sta	Profile	E.G. Elev	W.S. Elev	Q Total	Q Weir	Q Gates
			(ft)	(ft)	(cfs)	(cfs)	(cfs)
Weir Reach	41.75	08APR1999 1600	7.24	7.12	33973.50		33973.50
Weir Reach	41.75	08APR1999 1800	8.21	8.05	40919.32		40919.32
Weir Reach	41.75	08APR1999 2000	10.16	10.05	42693.18	244.17	42449.01
Weir Reach	41.75	08APR1999 2200	12.14	12.02	48081.21	2174.68	45906.54
Weir Reach	41.75	08APR1999 2400	13.95	13.82	53976.07	5695.11	48280.96
Weir Reach	41.75	09APR1999 0200	15.43	15.28	60307.41	13798.10	46509.31
Weir Reach	41.75	09APR1999 0400	16.44	16.28	66122.33	21532.32	44590.01
Weir Reach	41.75	09APR1999 0600	17.12	16.94	70053.87	27124.99	42928.87

Energy gradeline for given WSEL.

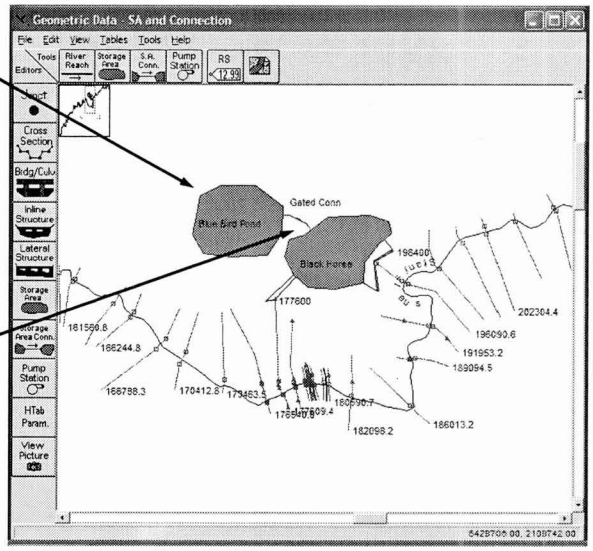
Overview of Storage Areas and Storage Area Connections

Cameron Ackerman, P.E.

1

Introduction

- Storage Areas
  - Purpose
  - Data Entry
  - Output
- SA Connections
  - Purpose
  - Data Entry
  - Output



The output from an unsteady flow run are stage hydrographs, flow hydrographs, and basic stage profiles. These hydrographs and profiles are stored in DSS.

For more detailed output you must run the post processor. The post processor reads the hydrograph and profile information in the DSS file and creates detailed output data for the maximum stage computed and series of snap shots in time on a selected "Detailed Output Interval".

Storage Areas - Purpose

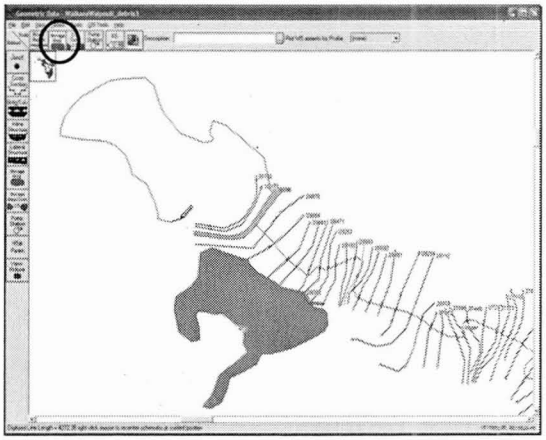
- Good for areas that are either inline or off line storage areas
- Stabilizing influence on computations
- Continuity equation is used to account for volume of storage areas
- Momentum is not computed
- Can be connected to:
  - Lateral structures
  - Top or bottom of a reach
  - Another storage area (with a SA Connection)

3

Storage areas are more stable in computations (and faster) than representing a region with cross sections.

Storage Area - Creating

- Created on the geometry schematic using the Storage Area editor

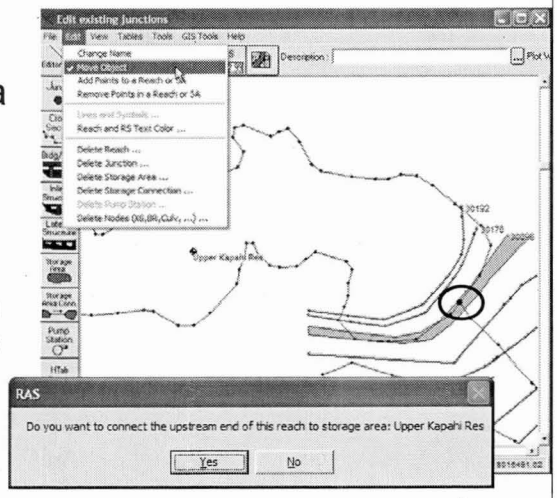


4

Storage Areas are created on the geometry schematic with the "Storage Area" tool. Press the button and the mouse cursor changes to a pen, draw the storage area and double click to end drawing.

Storage Area - Connecting

- When you connect the river to the SA a "node" is formed
- Upstream
  - Inline Structure with 2 cross sections u/s of structure
- Downstream

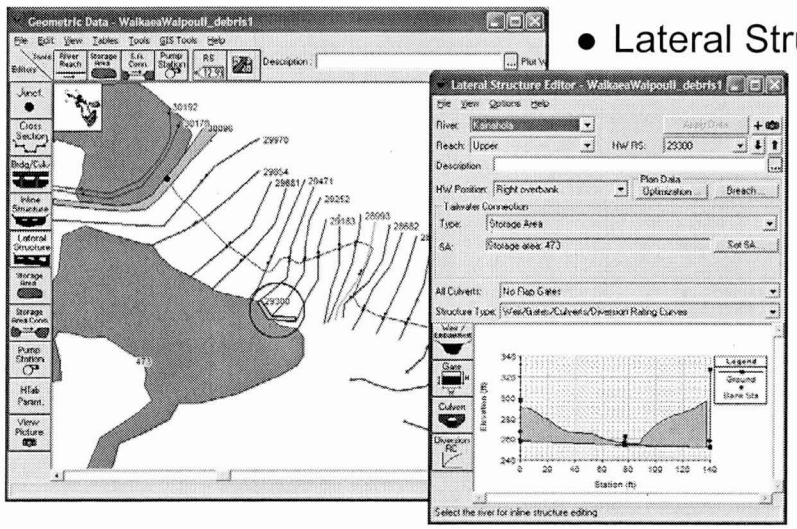


5

In order to connect a storage area to the upstream or downstream end of a river reach, select the Edit | Move Object menu item. Then grab the end of the river reach and move it into the storage area. RAS will ask you if you want to connect the river reach and storage area. If you accept Yes, then a node will be formed at the boundary of the storage area and the river line.

When using a storage area for an upstream boundary, there must be 2 cross sections placed upstream of the an inline structure.

Storage Area - Connecting



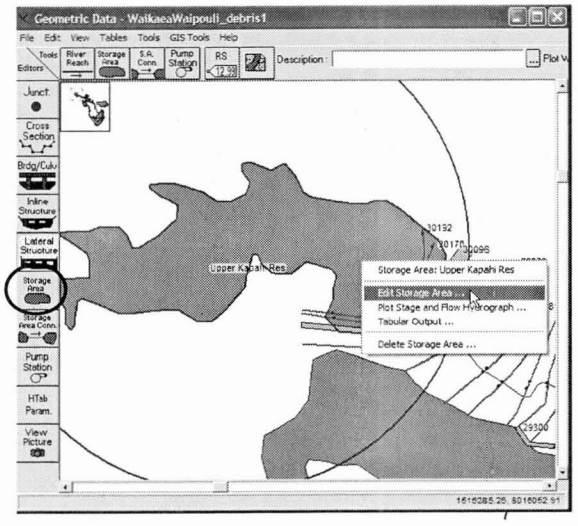
- Lateral Structure

6

To connect a storage are to a river, you do so using a lateral structure. A new lateral structure is added similar to adding a bridge, by selecting a river and reach and specifying a river station. The lateral structure has several options, but for a storage area you are connecting from the river (Headwater) on the right or left side to the storage area (Tailwater)!

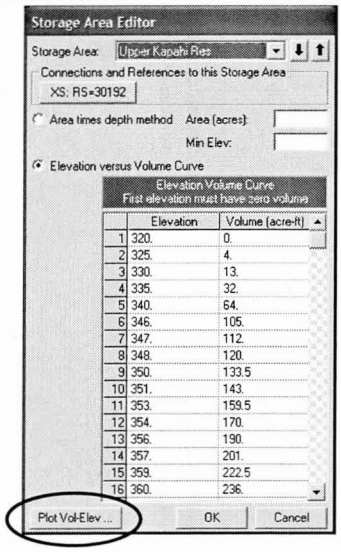
Storage Area - Geometry Data

- Open editor
  - Floating menu from left mouse click
  - Button

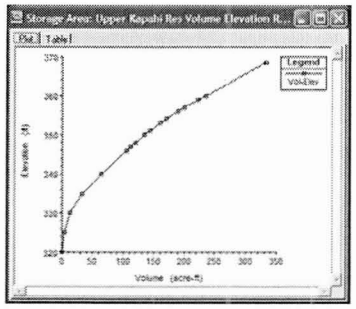


The data editor for storage areas can be accessed by clicking on the "Storage Area" button on the schematic editor or clicking on the storage area with left mouse button, this will display an popup menu with the option to edit .

Storage Area - Geometry Data



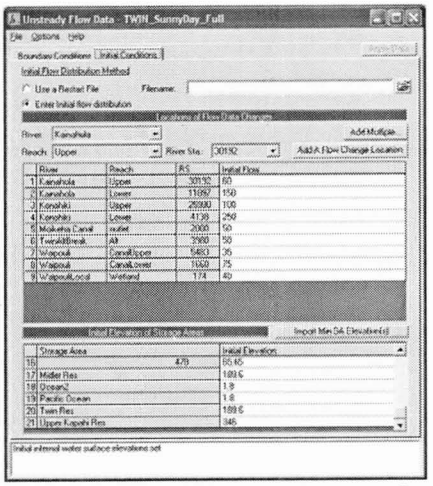
- Volume Methods
  - Area times depth
  - Elevation - Volume Curve
- Plot



The only data needed for storage areas is a method of computing storage verses elevation. Two methods are available, a surface area times a depth or interpolation from a rating curve.

Storage Area – Unsteady Flow Data

- Initial Conditions (Required)

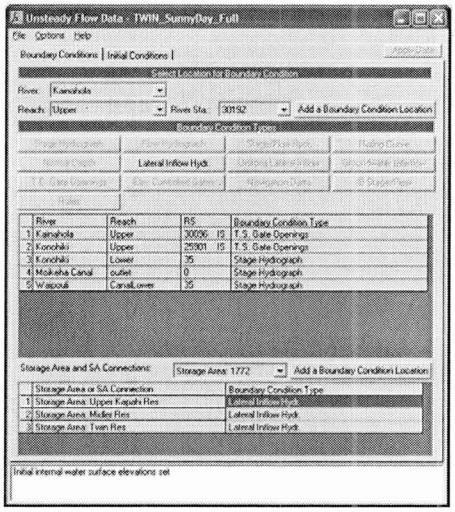


9

The initial elevation of the storage area is required for simulation, this is specified on the unsteady flow data window on the "Initial Conditions" tab.

Storage Area - Unsteady Flow Data

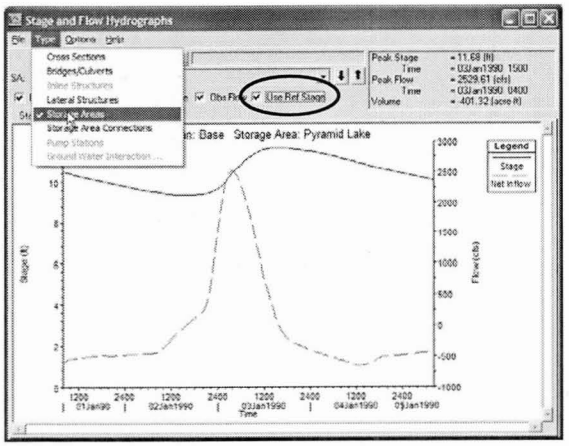
- Boundary Conditions (Optional)
  - Lateral Inflow Hydrograph



A time series of lateral inflows can also be entered for a storage area, this would be entered on the same window, but on the "Boundary Conditions" tab.

Storage Area - Output

- Stage Flow Plot
  - Stage
  - Net Flow



Storage Area - Output

- Profile Output Table

Profile Output Table - Storage Area

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: Base Profile: Max WS (Reload Data)

Storage Area	Profile	W.S. Elev (ft)	SA Min El (ft)	Net Flux (cfs)	SA Area (acres)	SA Volume (acre-ft)
Pyramid Lake	Max WS	11.68	0.00	15.16	1000.00	11684.09
Eagle Lake	Max WS	10.19	0.00	123.20	1500.00	10284.30
Lower Angora	Max WS	11.99	0.00	-20.95	1000.00	11990.83
Upper Angora	Max WS	10.64	0.00	249.67	1000.00	10640.69

12

SA Connection - Purpose

- Convey water from storage area to storage area
- Four types:
  - Inline Weir
  - Inline Weir with Gates
  - Inline Weir with Culverts
  - Linear Routing Method

13

SA Connection Computations

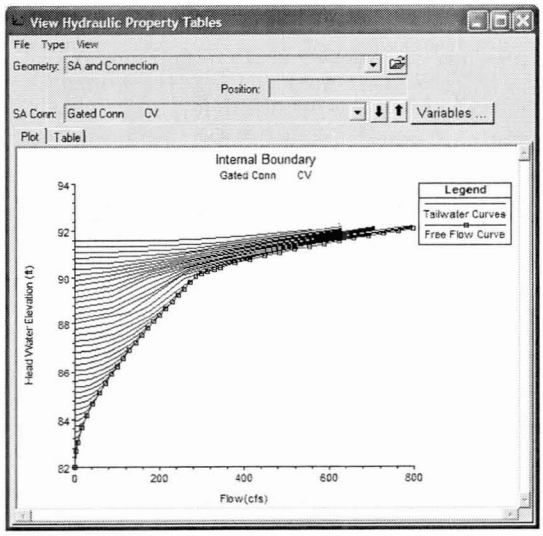
- Inline Weir Type
  - Can use the weir equation in computations
  - Can use a family of curves
- Inline Weir Type with gates
  - Uses the weir and gate equations in computations. Cannot use curves because the gates change the properties over time

14

The data for storage area connections is a combination of things that are available elsewhere in the program. For example the culvert data for a storage area connection is similar to the normal culvert data at a bridge/culvert crossing.

SA Connection Computations

- Culvert Type
  - Makes a family of rating curves similar to bridges and culverts



The family of rating curves can be viewed from the "View Hydraulic Properties" window. Select "SA Connections" under the "Type" menu option.

SA Connection Computations

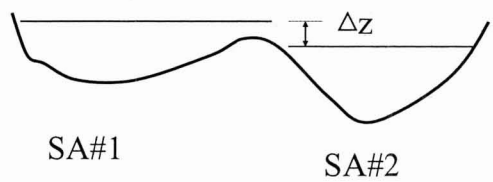
- Linear Routing Type

$Q = k(Available\ Storage)$

$Available\ Storage = Surface\ Area(\Delta z)$

$k\ has\ units\ of\ hours^{-1}\ and\ is\ in\ range\ of\ 0.0 - 1.0$

$with\ a\ typical\ value\ of\ 0.2$

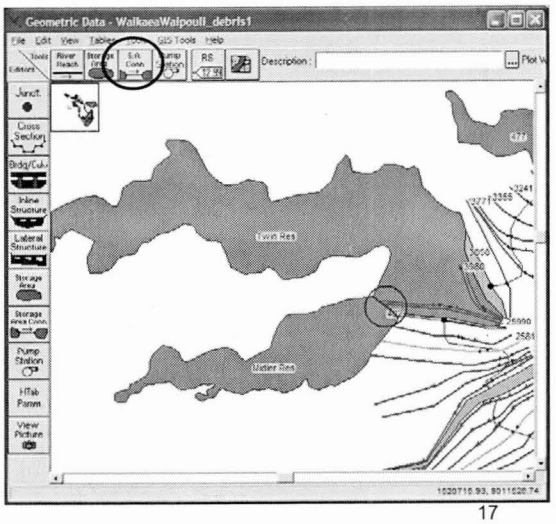


16

Another method of computing flow between storage areas is with linear routing. Linear routing computes the flow between storage areas by a simple fraction of the available storage difference between each storage area for each time step. The fraction, k has units of 1/hours.

SA Connection Creating

- Created on the geometry schematic
- Ends snap to the nearest Storage Area

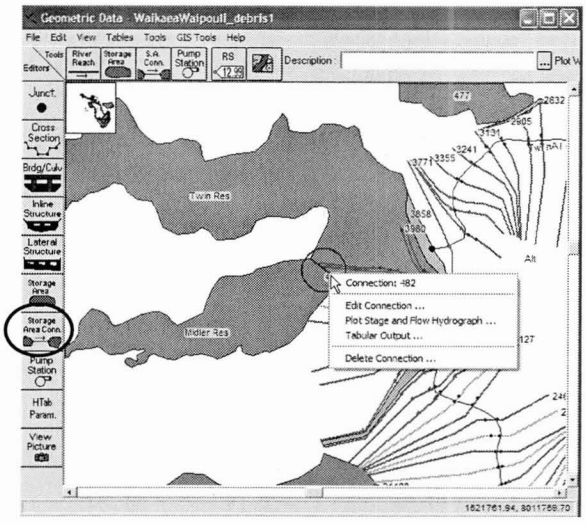


17

Press the SA Conn tool button on the top row of the schematic editor, and the mouse pointer will change to a pen. Draw the connection and double click to finish the polyline. SA connections snap to the nearest possible end point so start and end the connection near the desired storage area.

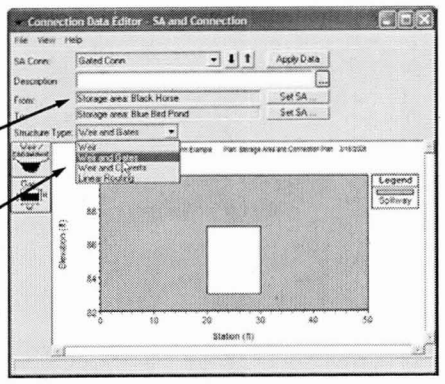
SA Connection - Geometry Data

- Open editor
  - Floating menu from left mouse click
  - Button



SA Connection - Geometry Data

- Connection Selection drop down list
- Description
- Editor Shows connections "From" and "To"
- Pick Type



19

When the SA Connection editor is opened the plot will initially be blank. The description can be entered and the "From" and "To" ends of the connection.

The buttons on the left, load sub-editors for the components of the connection.

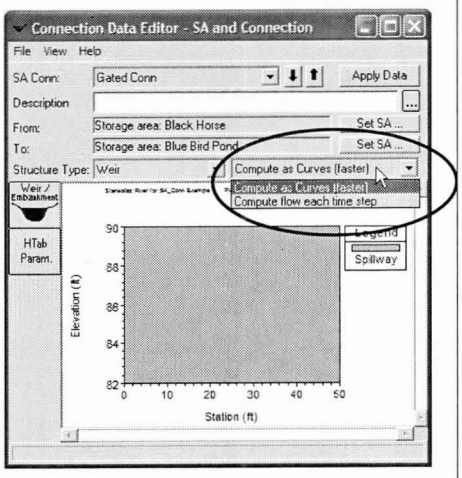
SA Connection - Geometry Data

- Inline Weir
  - Weir/Embankment
- Inline Weir with Gates
  - Weir/Embankment
  - Gates
- Culvert Type
  - Weir/Embankment
  - Culvert
  - HTab Parameters
- Linear Routing – (Coefficients Only)

20

SA Connection - Geometry Data

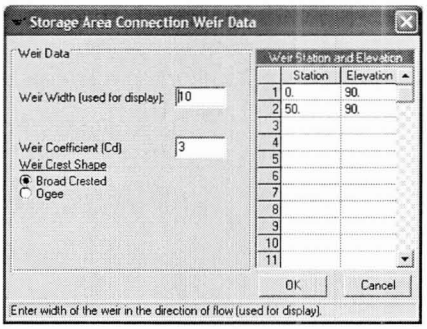
- Inline Weir
  - "On the fly" computations
  - Family of Curves



21

SA Connection - Geometry Data

- Weir/Embankment
  - Weir Width (Optional)
  - Flow Reference
  - Weir Coefficient range 0-4, default = 3 (maybe 2.6 would be better)
  - Weir Crest Shape
  - Weir Profile



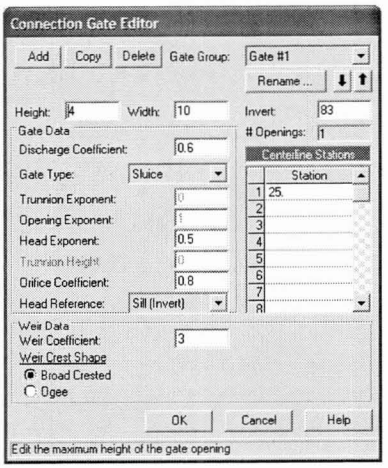
22

Weir Coefficients range from 0 to 4.

Application	Weir Coefficient
For a levee	2.6
Broad Crested Weir	2.6
Natural ground	<2

SA Connection - Geometry Data

- Gate Editor
  - Same editor for Inline and Lateral Structures

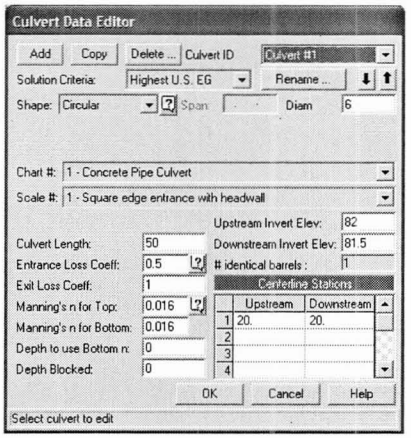


23

The data for a gate editor is the same as for an inline weir. Refer to the inline weir section of the HEC-RAS Users Manual starting on page 6 -57.

SA Connection - Geometry Data

- Culvert Editor
  - Same editor for Bridge/Culverts and Lateral Structures

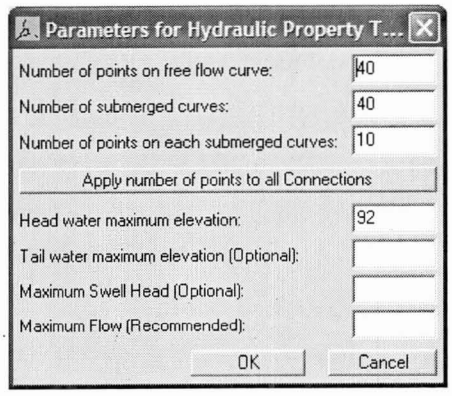


24

The data for a gate editor is the same as for an inline weir. Refer to the inline weir section of the HEC-RAS Users Manual starting on page 6 -42.

SA Connection - Geometry Data

- HTab Parameters
  - Headwater maximum (required)
  - Tailwater maximum (optional)
  - Maximum Swell Head (optional)
  - Maximum Flow (Optional)



25

The HTab data is only needed for the culvert type connections. The data required here is the same as covered for bridges and culverts, covered earlier in this course.

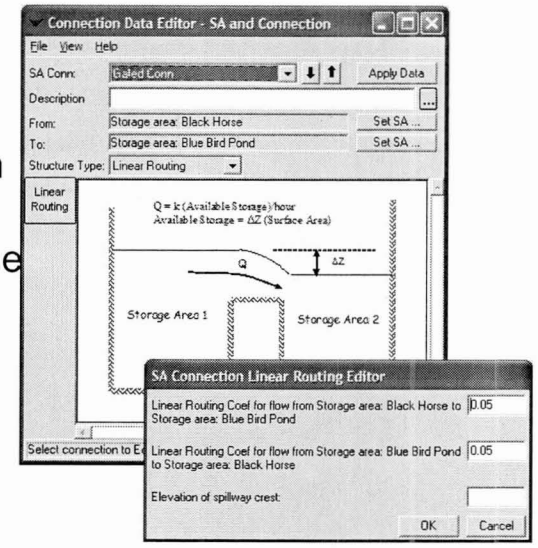
The maximum swell head parameter is the largest difference between head water and tailwater.

All four parameters maximum headwater, tailwater, swell head and maximum flow are used to limit the range of the family of rating curves. It is important to keep the curves in the possible flow and stage ranges to minimize the errors in interpolation

The only parameter required for connections is the headwater maximum, the program computes default values for the others.

SA Connection - Geometry Data

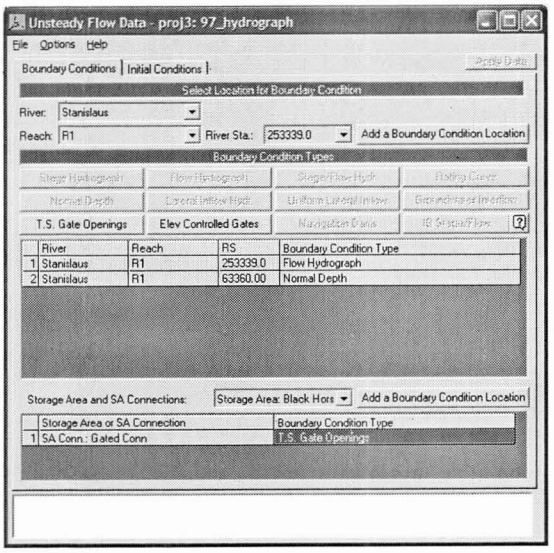
- Linear routing
- Coefficient used to compute change in Q per hour. Modified by the time step



The linear routing coefficient is direction dependent, so one for each flow direction must be entered. The "elevation of crest" is a reference that determines when water will start flowing from one storage area to the next.

SA Connection - Unsteady Flow Data

- Gated Connections
  - Time series of Gate Openings
  - Elevation Controlled Gates

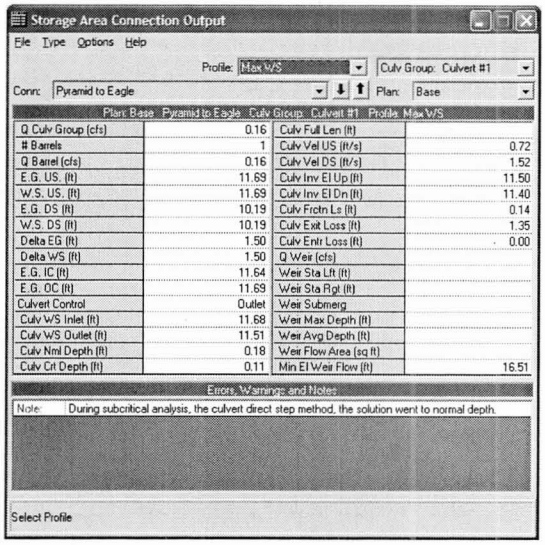


If a SA Connection has gates, then boundary condition data will need to be entered on the unsteady flow data editor. The lower portion of the boundary condition editor is where the connection is added to the table. Two types are possible, a time series of gate openings and elevation controlled gates.

The data for these boundary conditions are the same as for the gates on inline and lateral weirs.

SA Connection - Tabular Output

- Culvert Type



Storage Area Connection Output

File Type Options Help

Profile: Max WS | Culv Group: Culvert #1

Conn: Pyramid to Eagle | Plan: Base

Plan: Base Pyramid to Eagle Culv Group: Culvert #1 Profile: Max WS

Q Culv Group (cfs)	0.16	Culv Full Len (ft)	
# Barrels	1	Culv Vel US (ft/s)	0.72
Q Barrel (cfs)	0.16	Culv Vel DS (ft/s)	1.52
E.G. US. (ft)	11.69	Culv Inv El Up (ft)	11.50
W.S. US. (ft)	11.69	Culv Inv El Dn (ft)	11.40
E.G. DS (ft)	10.19	Culv Frctn Ls (ft)	0.14
W.S. DS (ft)	10.19	Culv Exit Loss (ft)	1.35
Delta EG (ft)	1.50	Culv Entr Loss (ft)	0.00
Delta WS (ft)	1.50	Q Weir (cfs)	
E.G. IC (ft)	11.64	Weir Sta Lft (ft)	
E.G. OC (ft)	11.69	Weir Sta Rgt (ft)	
Culvert Control	Outlet	Weir Submerg	
Culv WS Inlet (ft)	11.68	Weir Max Depth (ft)	
Culv WS Outlet (ft)	11.51	Weir Avg Depth (ft)	
Culv Nml Depth (ft)	0.18	Weir Flow Area (sq ft)	
Culv Crt Depth (ft)	0.11	Min El Weir Flow (ft)	16.51

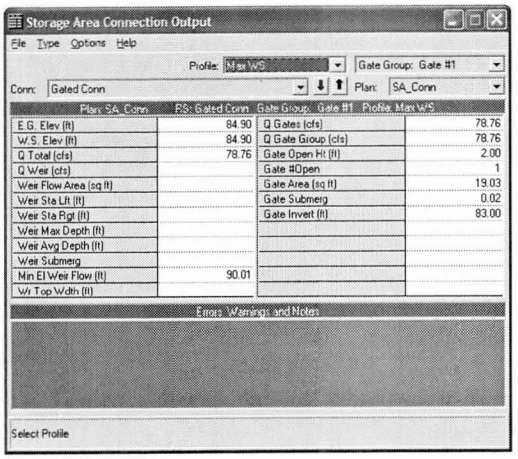
Errors, Warnings and Notes

Note: During subcritical analysis, the culvert direct step method, the solution went to normal depth.

Select Profile

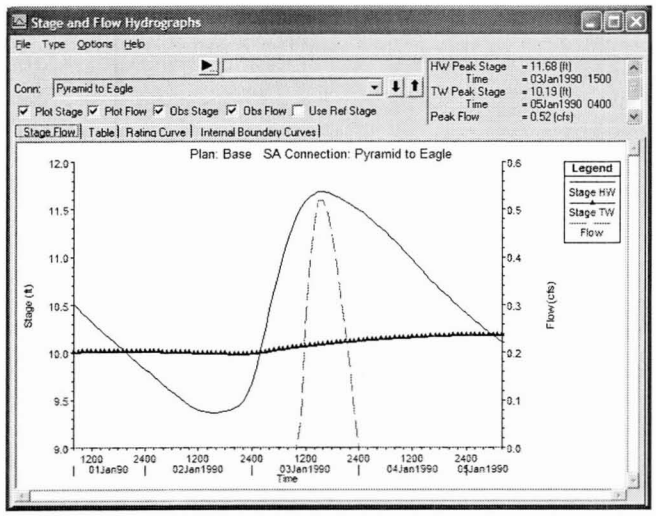
SA Connection - Tabular Output

- Inline Weir with Gates



29

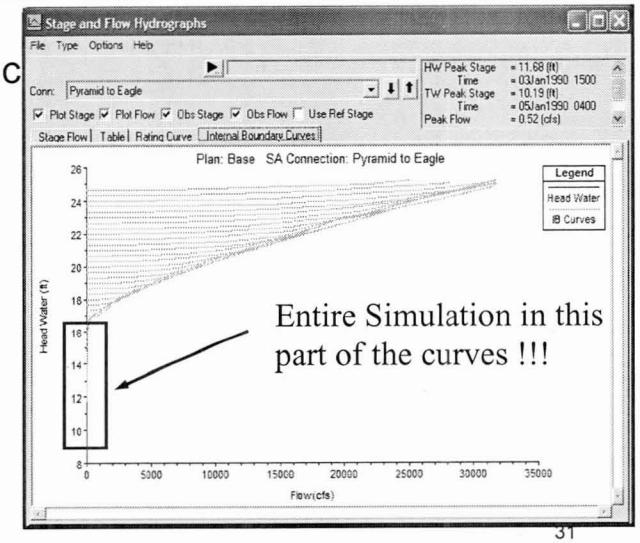
SA Connection - Stage Flow Plots



30

SA Connection - Stage Flow Plots

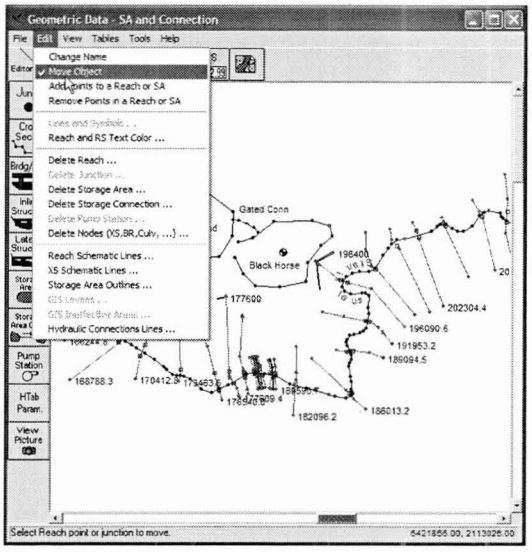
- Limit hydraulic properties



31

Schematic Editing

- Select "Move Object" under "Edit"
- Move individual points
- Move whole SA with centroid
- Move Text
- SA Conn snap to nearest point



Once the outline is created, it can be edited by selecting the "Move Object" menu item under "Edit". The schematic will change its appearance and allow features to be moved.

Storage Areas & SA Connections

- Questions?

33

Lateral Structures, Storage Areas, and SA Connections Workshop

Introduction

This workshop will help students learn how to use HEC-RAS to create storage areas, lateral structures with gates, and storage area connections. Students will learn how to connect storage areas to river reaches and other storage areas using Storage Area Connections. *While this data is from a USGS study, the model and results of this workshop do not represent current or future conditions of the river.*

Background

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. The bridge crossing is located along State Highway 1049, near the middle of the river reach. The bridge, cross section geometry, and high water flow data were used to evaluate the flood that occurred on May 22, 1974 with a peak of 14000 cfs. Figure 1 displays a geo-referenced map of the area with the surveyed cross sections.

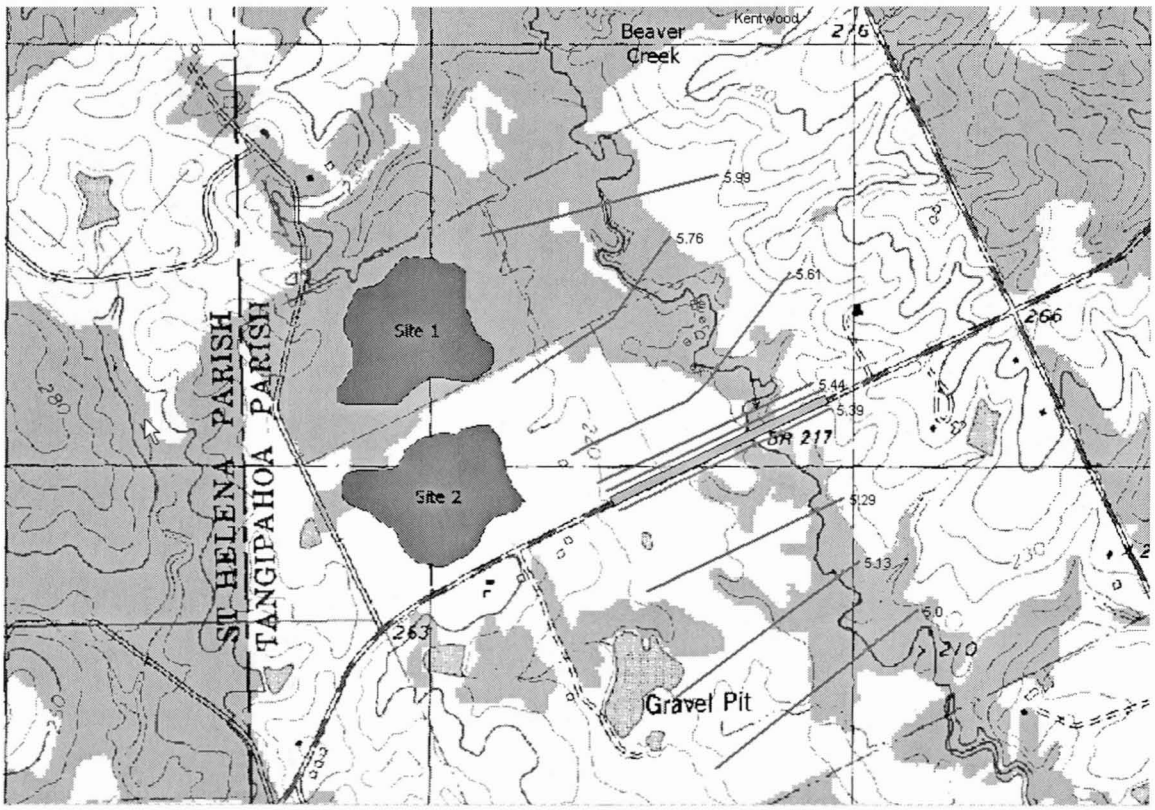


Figure 1 Beaver Creek Located Near Kentwood, Louisiana

Problem Description

A project file (**BeaverSA.prj**) with the title "**Beaver Cr. SA - and SA Connection**" has been loaded on your workshop computers. The file contains all the data for this workshop. The goal of this workshop is to evaluate a proposed plan to stop the overtopping of the bridge located at River Station 5.4 during a design event that peaks at 14000 cfs. It is proposed that two areas located to the west of Beaver Creek, labeled Site 1 and Site 2, be used for off-stream storage during peak flood events to prevent the bridge from overtopping. Site 1 is 80 acres and Site 2 is 85 acres. Each storage area will be connected via a lateral structure to Beaver Creek. The storage areas will be connected to each other with a storage area connection. The downstream lateral structure will be equipped with gates to control the release of stored water. The unsteady flow file already contains the design hydrograph.

Tasks

The following is a summary of the required tasks for each group:

1. Develop Storage Areas, Lateral Structures, and SA Connections

Utilizing the storage area, lateral structure, and storage area connection options within HEC-RAS evaluate the proposed plan to develop the available area to the west of Beaver Creek for off-stream storage. Remember that the goal of the design is to prevent the highway bridge from being overtopped. Some design information is unknown and should be evaluated by the modeler.

a. Construct storage areas for Site 1 and 2. Site 1 has an average minimum elevation of 209 feet and Site 2 has an average minimum elevation of 207 feet.

b. In the **Unsteady Flow Data Editor** set the initial water surface elevations of the storage areas.

c. Connect the storage areas to each other with a storage area connection. The connection should consist of a weir and culvert. The proposed weir length and elevation are 100 feet and 217 feet, respectively. The weir width is 20 feet. The culvert is a concrete pipe with a square edge entrance from the headwall, 7 feet in diameter and 30 feet long. The culvert is located along the centerline of the weir with an upstream invert elevation of 209 feet.

   In the **SA Connection Data Editor** select the **HTAB Parameter** button, set the maximum headwater elevation to 220 feet.

d. Connect the two storage areas to the reach with lateral structures. (Note: Do not try to use a storage area connection to connect the lateral structures from the reach to the storage areas.)

   The proposed upstream weir is located at River Station 5.9 and is 400 feet in length. The proposed downstream weir is located at River Station 5.570 and is 200 feet in length. The correct elevation of the weirs in unknown and must be evaluated by the modeler. The height of each weir will most likely fall within the range of 213 to 218 feet.

e. Save the geometry file as a new file. Make a new plan with the new geometry file and proceed with the simulation.

f. Evaluate the bridge for weir flow by viewing the output tables, or by examining the **Profile Plot** to see if the energy grade line rises above the bridge deck. Make adjustments to the weir elevations and storage areas until the bridge is not overtopped.

2. Place Gates on the Downstream Lateral Structure

After the flood wave has passed, re-route the stored water back into Beaver Creek. To accomplish this task, place gates along the downstream weir. (The upstream storage area should empty into the downstream storage area through the storage area connection).

a. Construct sluice gates along the lateral structure. The sluice gates have a discharge coefficient of 0.6. Save the geometry file as a different file for Task 2.

b. In the **Unsteady Flow Data Editor** develop a time series of operation for the gates that will release the water in the downstream storage area after the flood wave. Save the unsteady flow file as a different file for Task 2.

c. Save a new plan for Task 2 with the new geometry and unsteady flow files and then proceed with the simulation.

Questions

1. Did your design meet the requirement to prevent weir flow over the highway bridge?

2. When did the maximum flowrate occur in the storage areas? What was the maximum stage of both storage areas? What was the maximum flowrate between the two storage areas? What was the maximum flowrate over the lateral structure located at River Station 5.9?

3. How did the stage and flow hydrographs change from Task 1 to Task 2? Did any numerical oscillations occur in the stage and flow hydrographs? If so, how can these oscillations be smoothed?

4. What are some other alternatives, besides off-stream storage areas, to keep the bridge from being overtopped?

Common Model Stability Problems When Performing an Unsteady Flow Analysis

Gary W. Brunner, P.E.

Hydrologic Engineering Center

1

Objectives

- For students to have a better understanding of model stability problems.
- To become familiar with the available parameters and techniques within HEC-RAS that will allow you to develop a stable and accurate model.
- To learn how to detect, find, and fix model stability problems.

Hydrologic Engineering Center 2

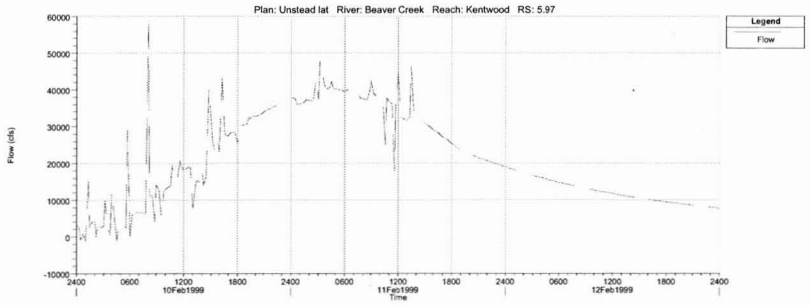
Overview

- Model Stability
- Factors Affecting Model Stability
  - Cross section spacing
  - Computational time step selection
  - Theta weighting factor
  - Calculation tolerances and iterations
  - Lateral Structures/weirs
  - Manning's n values
  - Initial/Low flow conditions
  - Steep Streams/Mixed Flow regime
  - Drops in the bed profile
  - Bridge/Culverts
  - Cross section geometry and table properties
  - Breach characteristics
- Detecting and fixing Stability Problems

Hydrologic Engineering Center 3

Model Stability

- An unstable numerical model is one for which certain types of numerical errors grow to the extent at which the solution begins to oscillate, or the errors become so large that the computations can not continue.

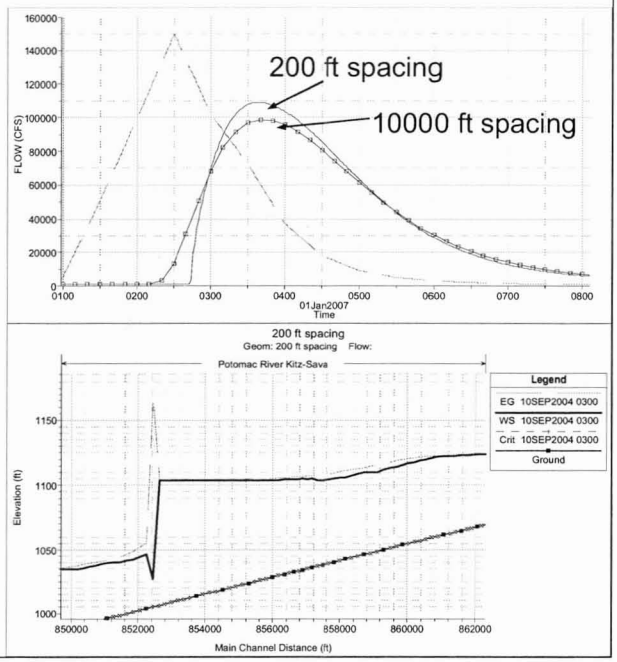


Hydrologic Engineering Center 4

Developing a stable model is a common problem when working with an unsteady flow model of any size or complexity. Modeling a dam break flood wave is one of the most difficult unsteady flow problems to model.

Cross Section Spacing

- Cross sections placed to far apart can cause numerical damping of the flood wave (to low of a peak flow downstream), and/or model instability.
- Cross sections placed to close together can cause wave steepening and model instability on the rising side of the flood wave.



Hydrologic Engineering Center

Not enough cross sections: When cross sections are spaced far apart, and the changes in hydraulic properties are great, the solution can become unstable. In general, cross sections spaced too far apart will cause additional numerical diffusion, due to the derivatives with respect to distance being averaged over to long of a distance. Also, if the distance between cross sections is so great, such that the Courant number would be much greater than 1.0, then the model may also become unstable.

Cross Sections too Close. If the cross sections are too close together, then the derivatives with respect to distance may be overestimated, especially on the rising side of the flood wave. This can cause the leading edge of the flood wave to over steepen, to the point at which the model may become unstable.

XS Spacing
Maximum and Minimum

Use Dr. Fread's and Samuals equations as a guide for maximum spacing.

Dr. Fread's Equation:

$$\Delta x \leq \frac{cT_r}{20}$$

Samuals Equation:

$$\Delta x \leq \frac{0.15D}{S_0}$$

Where:

Δx	= Cross section spacing (feet)
T_r	= Time of rise of the main flood wave (seconds)
c	= Wave speed of the flood wave (ft/s)
D	= Average bank full depth of the channel (ft)
S_0	= Average bed slope (ft/ft).

Minimum spacing for a dam break model should be in the range of 50 to 100 ft.

Hydrologic Engineering Center 6

One of the first steps in stabilizing a dam breach model is to apply the correct cross section spacing. Fread's equation and Samuel's equations are good starting points. Samuels equation is a little easier to use since you only have to estimate the depth and slope. Frequently, bank full depth is used. For Fread's equation, although the time of rise of the hydrograph (Tr) is easy enough to determine, the wave speed (c) is a little more difficult to come by. Once a cross section spacing is decided upon, apply it to the entire reach using the HEC-RAS cross section interpolation routines. Make sure that the reach-wide method is applicable. At areas of extreme contraction and expansion, at grade breaks, or in abnormally steep reaches, further localized interpolation may be necessary.

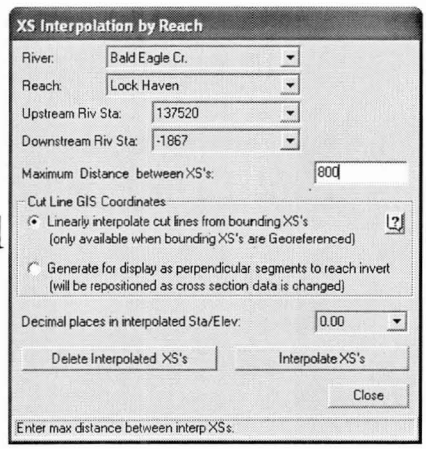
Fread, D.L. (1988) (Revision 1991). "The NWS DAMBRK Model. Theoretical Background and User's Documentation." National Weather Service, Office of Hydrology, Silver Spring, Md.

Fread, D.L., Lewis, J.M. (1993). "Selection of Dx and Dt Computational Steps for Four-Point Implicit Nonlinear Dynamic Routing Models" ASCE National Hydraulic Engineering Conference Proceedings, San Francisco, CA.

Samuels, P.G. (1989). "Backwater lengths in rivers", Proceedings -- Institution of Civil Engineers, Part 2, Research and Theory, 87, 571-582.

Cross Section Interpolation

- Apply the XS interpolation to ensure a maximum spacing is not exceeded.
- At problem areas you may need to use tighter spacing:
  - Steep reaches
  - Transition zones
  - Grade breaks

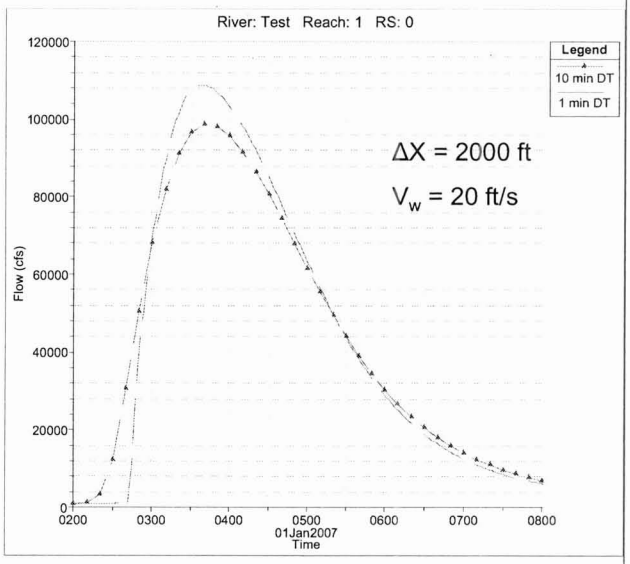


Hydrologic Engineering Center 7

In general, it is always better to use real cross sections rather than interpolated. However, if acquiring more cross section data is not possible, then the cross section interpolation routines in HEC-RAS should be used to ensure that the cross section do not go over a maximum distance estimated from Samual's, or Fread's equation.

Computational Time Step

- To large a time step will cause numerical diffusion (attenuation of the peak) and also model instability.
- To small of a time step can also lead to model instability as well as very long computation times.
- For this example a 5 sec time step caused the model to go unstable.



Hydrologic Engineering Center 8

Too large of a time step: When the solution scheme solves the unsteady flow equations, derivatives are calculated with respect to distance and time. If the changes in hydraulic properties at a give cross section are changing rapidly with respect to time, the program may go unstable. The solution to this problem in general is to decrease the time step.

Too Small of a Time Step. If a time step is selected that is much smaller than what the Courant condition would dictate for a given flood wave, this can also cause model stability problems. In general to small of a time step will cause the leading edge of the flood wave to steepen, possible to the point of oscillating and going unstable.

Computational Time Step - continued

Stability and accuracy can be achieved by selecting a time step that satisfies the Courant Condition :

$$C_r = V_w \frac{\Delta t}{\Delta x} \leq 1.0 \qquad \Delta t \leq \frac{\Delta x}{V_w}$$

For most rivers, the flood wave velocity is calculated more accurately by:

$$V_w = \frac{dQ}{dA}$$

An approximate flood wave velocity can be calculated as:

$$V_w = \frac{3}{2}\overline{V}$$

Hydrologic Engineering Center 9

Where: V_w = The flood wave speed, which is normally greater than the average velocity.

V = Average velocity of the flow

Δx = Distance between cross sections

Δt = computational time step

Q = flow rate

A = Flow area

User's should pay close attention to the Courant condition for selecting the computational interval.

Practical Time Step Selection

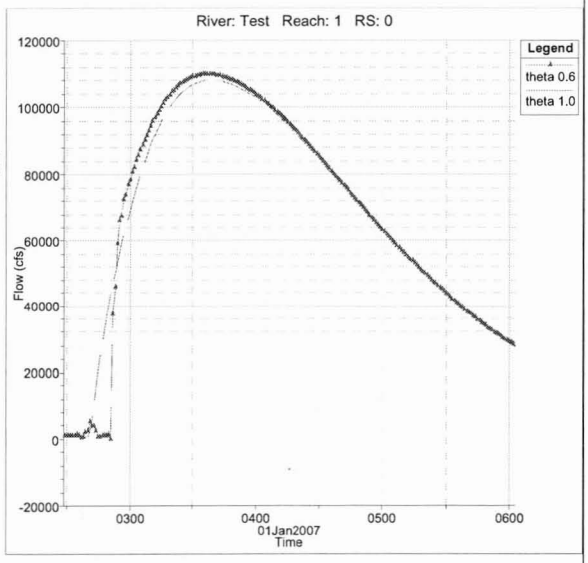
- For medium to large rivers the Courant condition may yield time steps that are to restrictive (i.e. a larger time step could be used and still maintain accuracy and stability).
- A practical time step is = $\Delta t \leq \frac{Tr}{20}$
- Remember that for Dambreak models, typical time steps are in the range of 1- 60 seconds due to the very fast flood wave velocities..

Hydrologic Engineering Center 10

Where: T_r = Time of rise of the flood wave.

Theta Weighting Factor

- Theta is a weighting applied to the finite difference approximations when solving the unsteady flow equations.
- Theoretically Theta can vary from 0.5 to 1.0. However a practical limit is from 0.6 to 1.0
- Theta of 1.0 provides the most stability. Theta of 0.6 provides the most accuracy.
- The default in HEC-RAS is 1.0. Once you have your model developed, try to reduce theta towards 0.6, as long as the model stays stable.



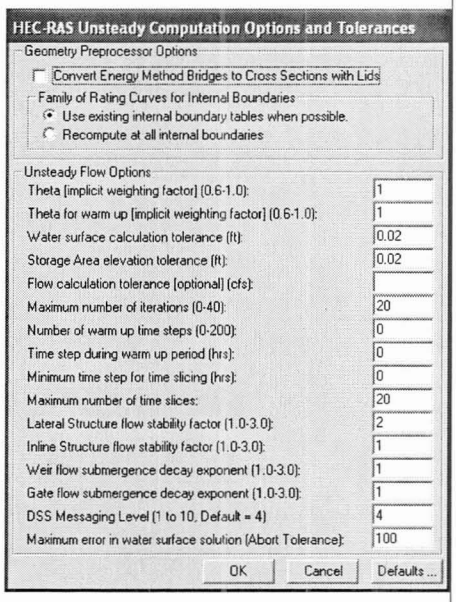
Hydrologic Engineering Center 11

Larger values of theta increase numerical diffusion, but, by how much? Experience has shown that for short period waves that rapidly rise, theta of 1.0 can produce significant errors. However, these errors can be reduced by using smaller time steps.

When choosing theta, one must balance accuracy and computational robustness. Larger values of theta produce a solution that is more robust, less prone to blowing up. Smaller values of theta, while more accurate, tend to cause oscillations in the solution, which are amplified if there are large numbers of internal boundary conditions.

Calculation Options and Tolerances

- Theta
- Water surface/SA/flow tolerance
- Maximum Number of Iterations
- Warm up: duration and time step
- Time slicing
- Lateral/Inline Structure Stability
- Weir/Gate flow submergence



Hydrologic Engineering Center 12

Water Surface, Storage Area, and Flow Tolerances: Three solution tolerances can be set or changed by the user: Water surface calculation (0.02 default); Storage area elevation (0.05 default); and Flow calculation (Default is that it is not used). The default values should be good for most river systems. Only change them if you are sure!!!

Making the tolerances larger can reduce the stability of the solution. Making them smaller can cause the program to go to the maximum number of iterations every time.

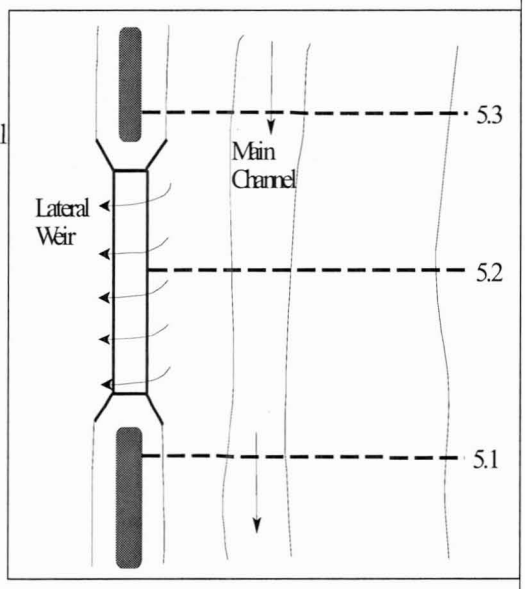
Maximum Number of Iterations: At each time step derivatives are estimated and the equations are solved. All of the computation nodes are then checked for numerical error. If the error is greater than the allowable tolerances, the program will iterate. The default number of iterations in HEC-RAS is set to 20. Iteration will generally improve the solution. This is especially true when your model has lateral weirs and storage areas.

Warm up time step and duration: The user can instruct the program to run a number of iterations at the beginning of the simulation in which all inflows are held constant. This is called the warm up period. The default is not to perform a warm up period, but the user can specify a number of time steps to use for the warm up period. The user can also specify a specific time step to use (default is to use the user selected computation interval). The warm up period does not advance the simulation in time, it is generally used to allow the unsteady flow equations to establish a stable flow and stage before proceeding with the computations.

Time Slicing: The user can control the maximum number of time slices and the minimum time step used during time slicing. There are two ways to invoke time slicing: rate of change of an inflow hydrograph or when a maximum number of iterations is reached.

Inline/Lateral Structure Stability Issues

- Long and flat lateral weirs can often be a source of model instability.
  - Small change in stage in the river results in big change in flow going out the lateral structure
  - Flow is assumed to be constant over the time step
- Solutions:
  - Reduce the computation interval
  - Put a small slope on the lateral weir
  - Use lateral structure stability factors
- Opening and Closing Gates Quickly
  - Reduce the computational time step
  - Open and close gates slower



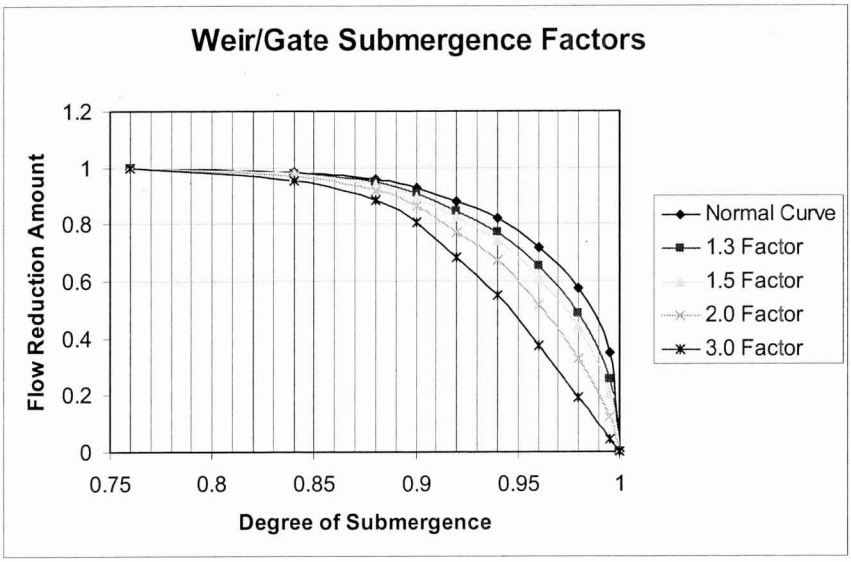
Hydrologic Engineering Center 13

Inline and Lateral Structures can often be a source of instability in the solution. Especially lateral structures, which take flow away or bring it into the main river. During each time step, the flow over a weir/spillway is assumed to be constant. This can cause oscillations by sending too much flow during a time step. One solution is to reduce the time step. Another solution is to use Inline and Lateral Structure stability factors, which can smooth these oscillations by damping the computed flows. However, using these stability factors can reduce the accuracy of the computed values. The Inline and Lateral Structure stability factors can range from 1.0 to 3.0. The default value of 1.0 is essentially no damping of the computed flows. As you increase the factor you get greater dampening of the flows (which will provide for greater stability), but less accuracy.

Long and flat Lateral Weirs/Spillways: during the computations there will be a point at which for one time step no flow is going over the lateral weir, and then the very next time step there is. If the water surface is rising rapidly, and the weir is wide and flat, the first time the water surface goes above the weir could result in a very large flow being computed (i.e. it does not take a large depth above the weir to produce are large flow if it is very wide and flat). This can result in a great decrease in stage from the main river, which in turn causes the solution to oscillate and possible go unstable. This is also a common problem when having large flat weirs between storage areas. The solution to this problem is to use smaller computational time steps, and/or weir/spillway stability factors.

Opening gated spillways to quickly: When you have a gated structure in the system, and you open it quickly, if the flow coming out of that structure is a significant percentage of the flow in the receiving body of water, then the resulting stage, area and velocity will increase very quickly. This abrupt change in the hydraulic properties can lead to instabilities in the solution. To solve this problem you should use smaller computational time steps, or open the gate a littler slower, or both if necessary.

Weir/Gate Submergence Exponents

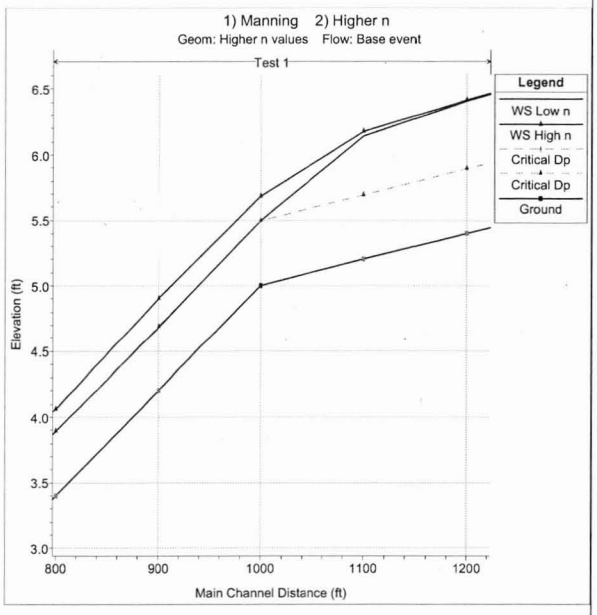


Hydrologic Engineering Center 14

To reduce the oscillations, the user can increase the Weir/Gate Submergence decay exponent. This factor can vary from 1.0 to 3.0. A factor of 1.0 leaves the submergence criteria in its original form. Using a factor greater than one causes the program to use larger submergence factors earlier, and makes the submergence curve less steep at high degrees of submergence. A plot of the submergence curves for various factors is shown in the Figure above.

Manning's n Values

- Manning's n values that are too low can cause model instability
  - Lower depths
  - Higher velocities
  - Supercritical flow
  - Flow transitions
- Manning's n values that are too high will locally increase stage and attenuate the hydrograph more as it moves downstream



Hydrologic Engineering Center 15

Manning's n values can also be a source of model instability. Manning's n values that are too low, will cause shallower depths of water, higher velocities, and possibly even supercritical flows. This is especially critical in steep streams, where the velocities will already be high. User's should check there estimated Manning's n values closely in order to ensure reasonable values. It is very common to underestimate Manning's n values in steep streams. Use Dr. Robert Jarrets equation for steep streams to check your main channel Manning's n values.

Over estimating Manning's n values will cause higher stages and more hydrograph attenuation than may be realistic.

Low Flow Conditions

- Low flows are often a source of model instability
  - Pools and riffles (flow passes through critical depth)
  - Very shallow depths (When flood wave starts the change in depth/velocity is very large, therefore derivatives are large)
- Solutions
  - Increase Base Flow of Hydrograph (change hydrographs directly or use the Qmin option on the hydrograph editor)
  - Rule of thumb: Start with 1% of peak, don't exceed 10% of peak
  - Use a "Pilot Channel" to smooth our bed irregularities and provide some artificial depth.

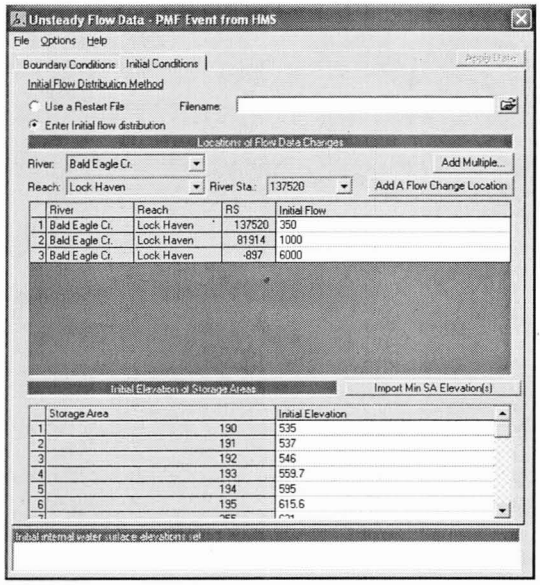
Hydrologic Engineering Center 16

If any portion of an inflow hydrograph is so low that it causes the stream to go through a pool and riffle sequence, it may be necessary to increase the base flow. The minimum flow value must be small enough that it is negligible when compared to the peak of the flood wave. A good rule of thumb is to start with a minimum flow equal to about 1 % of the peak flood (inflow hydrograph, or dam breach flood wave) and increase as necessary to 10%. If more than 10% is needed, then the problem is probably from something else.

Very shallow depths of water: When starting a simulation it is very common to start the system at low flows. If you have some cross sections that are fairly wide, the depth will be very small. As flow begins to come into the river, the water surface will change quickly. The leading edge of the flood wave will have a very steep slope. Sometimes this steep slope will cause the solution to reduce the depth even further downstream of the rise in the water surface, possible even producing a negative depth. This is do to the fact that the steep slope gets projected to the next cross section downstream when trying to solve for its water surface. The best solution to this problem is to use what is called a pilot channel. A pilot channel is a small slot at the bottom of the cross section, which gives the cross section a greater depth, without adding much flow area. This allows the program to compute shallow depths on the leading edge of the flood wave without going unstable. Another solution to this problem is to use a larger base flow at the beginning of the simulation.

Initial Conditions

- Initial condition flows need to be consistent with boundary condition flows at time zero.
  - If dendritic system – can leave initial condition flows blank.
- Initial Reservoir elevations need to be consistent with initial flows and gate settings used.
- Too low of, or inconsistent, initial conditions flow can often cause the model to blow up right at the start of the simulation



Hydrologic Engineering Center 17

Make sure that the initial conditions flow is consistent with the first time step flow, or minimum flow value, which ever is greater. User's must also pay close attention to initial gate settings for the reservoir, and the initial stage of the pool in the reservoir. The initial condition flow values must be consistent with all inflow hydrographs, as well as the initial flows coming out of the reservoir.

Flows entered on the initial conditions tab are used for calculating stages in the river system based on steady flow backwater calculations. If these flows and stages are inconsistent with the initial flows in the hydrographs, and coming out of the reservoir, then the model may have computational stability problems at the very beginning of the unsteady flow computations.

Steep Streams and Mixed Flow Regime

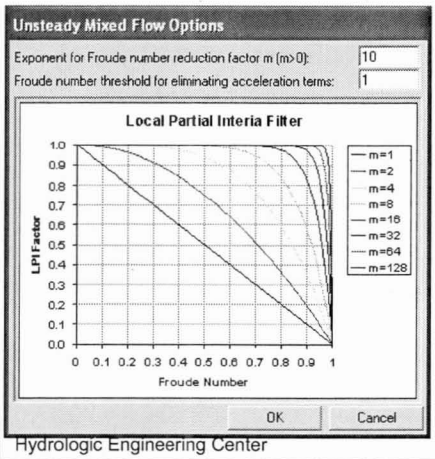
- Higher velocities and rapid changes in depth and velocity are more difficult to model and keep a stable solution.
- As Froude number approaches 1.0 (critical depth), the inertial terms of the St. Venant equations and their derivatives tend to cause model instabilities.
- Model goes to critical depth – RAS is limited to subcritcal flow for unsteady flow simulations, unless you turn on the mixed flow option.
- Solutions:
  - Higher n values (n values often under estimated in steep streams)
  - Increase base flow of hydrographs and initial condition flows
  - Try turning on the Mixed Flow regime option

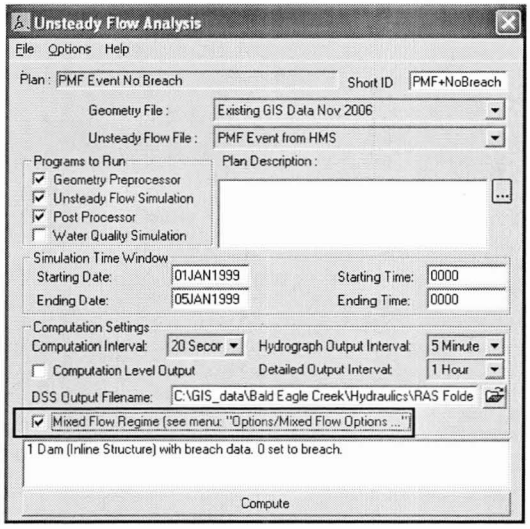
Hydrologic Engineering Center 18

Model goes to critical depth: The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritcal flow. However, the software does have an option to run in a mixed flow regime mode. However, this option should not be used unless you truly believe you have a mixed flow regime river system. If you are running the software in the default mode (subcritical only, no mixed flow), and if the program goes down to critical depth at a cross section, the changes in area, depth, and velocity are very high. This sharp increase in the water surface slope will often cause the program to overestimate the depth at the next cross section upstream, and possible underestimate the depth at the next cross section downstream (or even the one that went to critical depth the previous time step). One solution to this problem is to increase the Manning's n value in the area where the program is first going to critical depth. This will force the solution to a subcritical answer and allow it to continue with the run. If you feel that the true water surface should go to critical depth, or even to a supercritical flow regime, then the mixed flow regime option should be turned on. Another solution is to increase the base flow in the hydrographs, as well as the base flows used for computing the initial conditions. Increased base flow will often dampen out any water surfaces going towards or through critical depth due to low flows.

Mixed Flow Regime Option for Unsteady Flow

- HEC-RAS uses the Local Partial Inertia technique to model mixed flow regimes.
- User's can adjust the LPI parameters





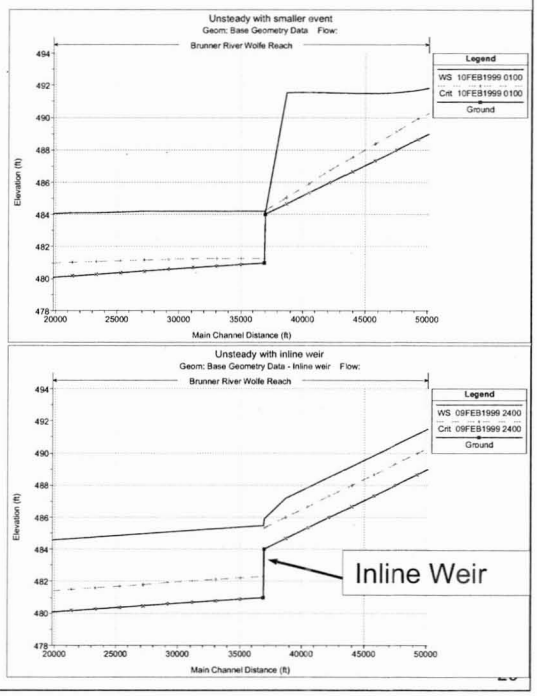
Hydrologic Engineering Center 19

In order to solve the stability problem for a mixed flow regime system, Dr. Danny Fread (Fread, 1986) developed a methodology called the "Local Partial Inertia Technique." The LPI method has been adapted to HEC-RAS as an option for solving mixed flow regime problems when using the unsteady flow analysis portion of HEC-RAS. This methodology applies a reduction factor to the two inertia terms in the momentum equation as the Froude number goes towards 1.0.

The default values for the equation are FT = 1.0 (Froude Number Threshold) and m = 10 (exponent). When the Froude number is greater than the threshold value, the factor is set to zero. The user can change both the Froude number threshold and the exponent. As you increase the value of both the threshold and the exponent, you decrease stability but increase accuracy. As you decrease the value of the threshold and/or the exponent, you increase stability but decrease accuracy. To change either the threshold or the exponent, select **Mixed Flow Options** from the **Options** menu of the Unsteady Flow Analysis window.

Drops In The Bed Profile

- Significant drops in the elevation of the channel bed can cause flow to pass through critical depth and results in an unstable model solution.
- Solutions:
  - Use an Inline Weir to model drop
  - Increase Base Flow
  - more cross sections and turn on mixed flow regime option
  - Rating curve for cross section at top of the drop



Hydrologic Engineering Center

Significant drops in the bed profile can also be a source of model stability problems, especially at low flows. If the drop is very small, then usually an increase in baseflow will drowned out the drop, thus preventing the model from passing through critical depth. If the drop is significant, then it should be modeled with an inline structure using a weir. This will allow the model to use a weir equation for calculating the upstream water surface for a given flow, rather than using the unsteady flow equations. This produces a much more stable model, as the program does not have to model the flow passing through critical depth with the unsteady flow equations. HEC-RAS automatically handles submergence on the weir, so this is not a problem.

Bridges and Culverts

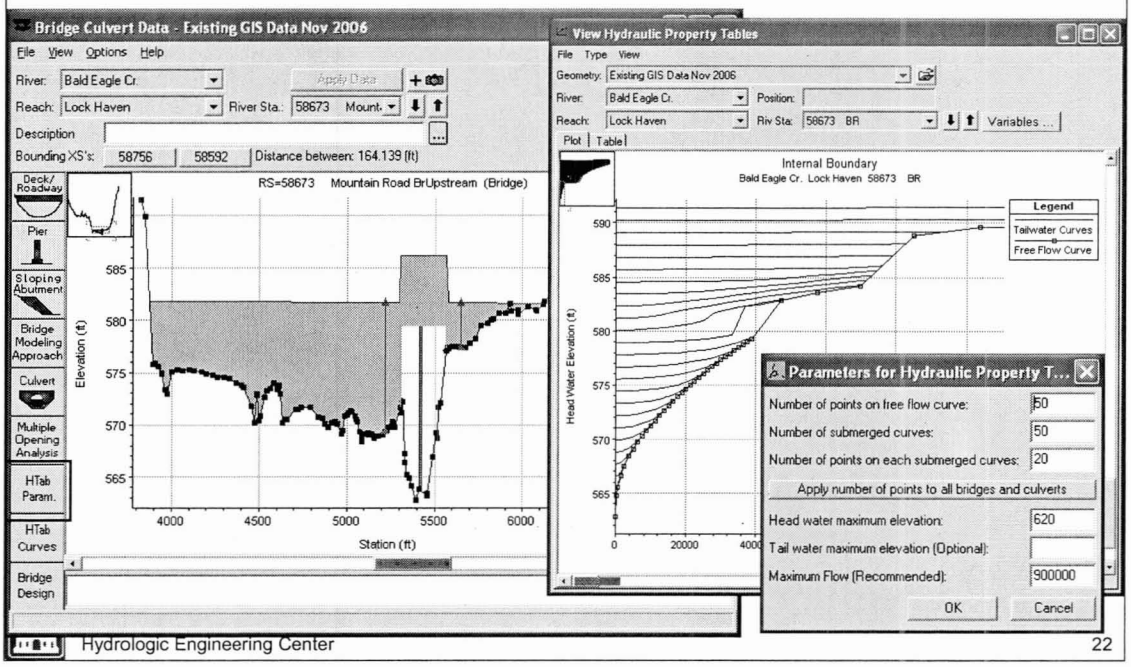
- It is very common to have rapid changes in depth and velocity at Bridges and Culverts, however this can be a source of model instability
- As flow transitions from low flow to pressure flow at the structure the water surface upstream will jump up very quickly.
- HEC-RAS pre-processes bridges/culverts into a family of curves (tailwater/headwater vs flow). Common Problems with the curves are:
  - Curve extents not high enough (change default extents)
  - Abrupt transitions in curves (adjust bridge parameters or use smaller ΔT)

Hydrologic Engineering Center 21

Bridge/Culvert crossings can be a common source of model stability problems when performing a Dam Break analysis. Many bridges will be overtopped during such an event. Many of those bridges may in fact be washed out during such an event. Common problems at bridges/culverts are the extreme rapid rise in stages when flow hits the low chord of the bridge deck or the top of the culvert. Modelers need to check the computed curves closely and make sure they are reasonable. One solution to this problem is to use smaller time steps, such that the rate of rise in the water surface is smaller for a given time step. Modelers may also need to change hydraulic coefficients to get curves that have more reasonable transitions.

An additional problem is when the curves do not go high enough, and the program extrapolates from the last two points in the curve. This extrapolation can cause problems when it is not consistent with the cross section geometry upstream and downstream of the structure.

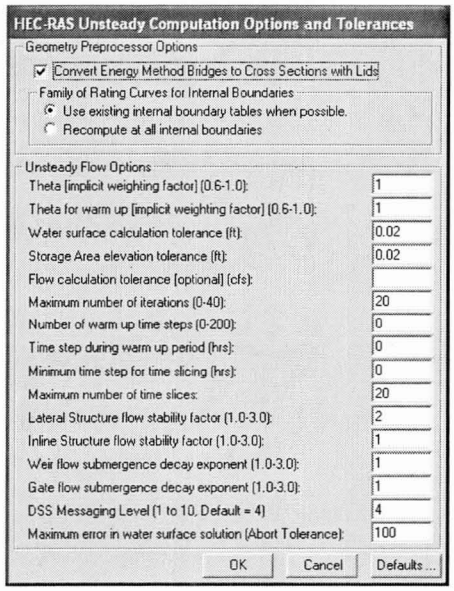
Bridge/Culvert Family of Curves



Bridge/Culvert Family of rating curves: The program creates a family of rating curves to define all the possible headwater, tailwater, and flow combinations that can occur at a particular structure. The user can control how many submerged curves get calculated (default = 50), how many points in each curve (default = 20), and the properties used to define the limits of the curves (maximum headwater, maximum tailwater, maximum flow, and maximum head difference). By default, the software will take the curves up to an elevation equal to the highest point in the cross section just upstream of the structure. This may lead to curves that are too spread out and go up to a flow rate that is way beyond anything realistic for that structure. These type of problems can be reduced by putting in specific table limits for maximum headwater, tailwater, flow, and head difference.

Convert Energy Bridges to Cross Sections with Lids

- HEC-RAS has on option to model a bridge as normal cross sections with a lid instead of the family of curves.
  - Good option when bridge/embankment are a small obstruction to flow
  - Bad idea when depth/velocity will change rapidly through the bridge (Will often cause model to blow up)

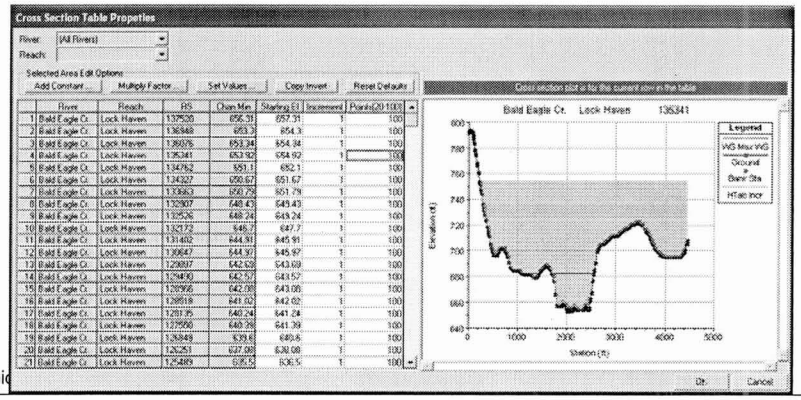


Hydrologic Engineering Center 23

The option to convert bridges that have been modeled with the energy equation for low and high flow is a good option if the structure and the road embankment are a small obstruction to the flow. If the flow will have to contract greatly inside of the structure, then modeling the bridge in this manner may lead to an unstable modeling solution through the structure. If you have bridges and culverts that will cause the flow to contract greatly through the structure, and even if you have chosen to use the energy equation for low and high flow, computing a family of curves for the structure will produce a more stable model. When the family of curves is used, the program does not solve the momentum and continuity equations inside of the structure, only outside of the structure. The curves themselves are use to obtain a resulting headwater for a given flow and tailwater, without solving for the hydraulics inside of the structure.

Cross Section Geometry and Table Properties

- Bad cross section properties, commonly caused by: levee options, ineffective flow areas, Manning's n values, etc…
- Cross section properties that do not go high enough, or are way too high (curves are spread to far apart).
- Not enough definition in the properties tables.



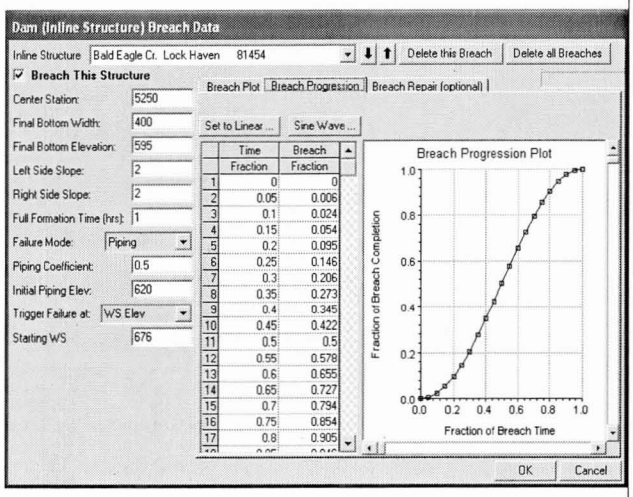
Bad cross section properties: All of the cross sections get converted to tables of hydraulic properties (elevation versus area, conveyance, and storage). If the curves that represent these hydraulic properties have abrupt changes with small changes in elevation, this can also lead to instability problems. This situation is commonly caused by: levees being overtopped with large areas behind them (since the model is one dimensional, it assumes that the water surface is the same all the way across the entire cross section); and ineffective flow areas with large amounts of storage areas that are turned on at one elevation, and then turn off at a slightly higher elevation (this makes the entire area now used as active conveyance area). There are many possible solutions to these problems, but the basic solution is to not allow the hydraulic properties of a cross section to change so abruptly. If you have a levee with are large amount of area behind it, model the area behind the levee separately from the cross section. This can be done with either a storage area or another routing reach, whichever is most hydraulically correct for the flow going over the levee or if the levee breaches. With large ineffective flow areas, the possible solutions are to model them as being permanently on, or to put very high Manning's n values in the ineffective zones.

Cross section property tables that do not go high enough: The program creates tables of elevation versus area, conveyance, and storage area for each of the cross sections. These tables are used during the unsteady flow solution to make the calculations much faster. By default, the program will create tables that extend up to the highest point in the cross section, however the user can override this and specify their own table properties (increment and number of points). If during the solution the water surface goes above the highest elevation in the table, the program simply extrapolates the hydraulic properties from the last two points in the table. This can lead to bad water surface elevations or even instabilities in the solution.

Not enough definition in cross section property tables: The counter problem to the previous paragraph is when the cross section properties in a given table are spread too far apart, and do not adequately define the changes in the hydraulic properties. Because the program uses straight-line interpolation between the points, this can lead to inaccurate solutions or even instabilities. To reduce this problem, we have increased the allowable number of points in the tables to 100.

Breach Characteristics

- By default HEC-RAS uses a linear growth rate for forming the breach.
- Rapid change in flow at the beginning of the breach can cause instability
- Try non-linear breach to smooth out change in flow (Sine Wave or User entered values)



Hydrologic Engineering Center 25

If the user puts in a very large breach, over a very short period of time, and they use the linear breach growth rate, the model will have a very abrupt change in flow starting right at the beginning of the breaching process. This rapid change in flow at the leading edge of the flood wave may cause an instability at the beginning of the breaching process, just downstream of the dam. Some possible solutions to this are:

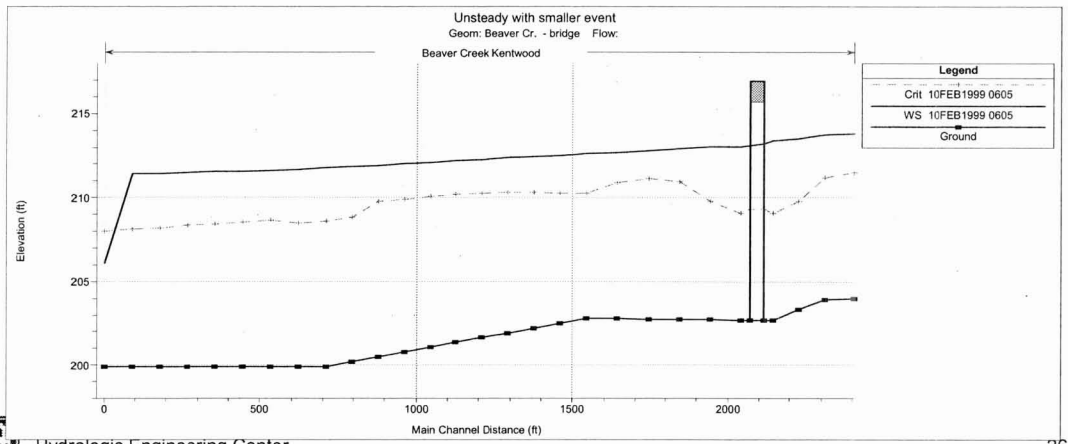
•Smaller time step

•Use the non-linear breach progression (sine wave or user entered)

•Increasing the overall breach time.

Bad Downstream Boundary Conditions

- Rating curve with bad points or not high enough
- Slope for normal depth to steep



Hydrologic Engineering Center 26

Bad downstream boundary condition: If the user entered downstream boundary condition causes abrupt jumps in the water surface, or water surface elevations that are too low (approaching or going below critical depth), this can cause oscillations in the solution that may lead to it going unstable and stopping. Examples of this are rating curves with not enough points or just simply to low of stages for a given flow; and normal depth boundaries where the user has entered to steep of a slope for the energy gradeline.

Detecting Stability Problems

- How do you know you have a model stability problem?
  - Program completely blows up during run.
  - Program says matrix solution went completely unstable during the calculations.
  - Computed error in water surface calc is very large
  - Program goes to maximum number of iterations for several time steps in a row, with large errors.
  - Program has oscillations in the computed stage and flow hydrographs.

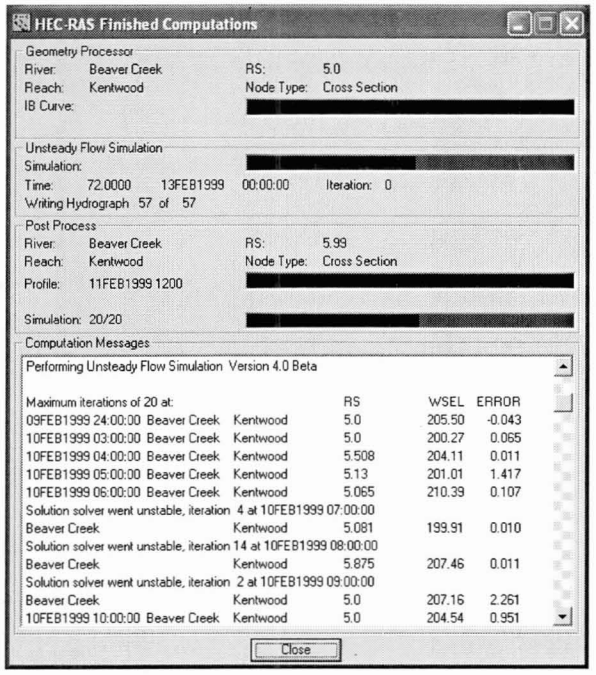
Hydrologic Engineering Center 27

Detecting Stability Problems - Continued

- What do you do when this happens?
  - Note the simulation time and location from the computation window when the program either blew up or first started to go to the maximum number of iterations with large water surface errors.
  - Use the HEC-RAS Profile and Cross Section Plots as well as the Tabular Output to find the problem location and issue.
  - If you can not find the problem using the normal HEC-RAS output - Turn on the "Detailed Output for Debugging" option and re-run the program.
  - View the text file that contains the detailed log output of the computations. Locate the simulation output at the simulation time when the solution first started to go bad.
  - Find the river station locations that did not meet the solution tolerances. Then check the data in this general area.

Computation Window

- First place to look for problems
- When the maximum number of iterations is reach, and solution error is greater than the predefined tolerance, the time step, river, reach, river station, water surface elevation and the amount of error is reported.
- When the error increases too much, the solution will stop and say "Matrix Solution Failed".
- Often the first RS to show up on the window can give clues to the source of instabilities

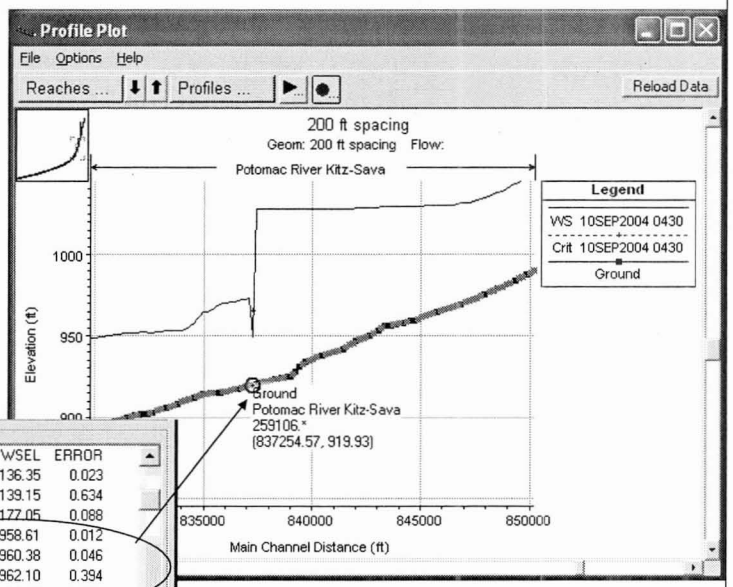


Hydrologic Engineering Center 29

The first place to look for instabilities and errors is the Computations Window during and just after the simulation is run. The red progress bar indicates the model went unstable and could not complete the simulation. The Computation Messages window provides a running dialog of what is happening in the simulation at a given time step in a given location. This allows the user to watch errors propagate during the simulation. Once the simulation has crashed, don't close the Computations Window. Instead, scroll up through the messages and try to determine where the propagation of errors began, and at what time.

Computation Window

- Small errors are generally not problematic
- Focus on larger compounding errors.
- Try to find time and location when errors first begin to occur.



Computation Messages

Maximum iterations of 20 at:		RS	WSEL	ERROR
10SEP2004 03:28:45	Potomac River Kitz-Sava	267788.9	1136.35	0.023
10SEP2004 03:29:00	Potomac River Kitz-Sava	267788.9	1139.15	0.634
10SEP2004 03:29:15	Potomac River Kitz-Sava	267788.9	1177.05	0.088
10SEP2004 04:48:30	Potomac River Kitz-Sava	259106.*	958.61	0.012
10SEP2004 04:48:45	Potomac River Kitz-Sava	259106.*	960.38	0.046
10SEP2004 04:49:00	Potomac River Kitz-Sava	259106.*	962.10	0.394
10SEP2004 04:49:15	Potomac River Kitz-Sava	259106.*	983.33	0.290
10SEP2004 05:10:30	Potomac River Kitz-Sava	242421.*	772.69	0.014

Close

30

Sometimes the first error to occur is at the beginning of the simulation and is just a result of the model settling out after the transition from initial conditions to the first time step. Particularly if the error only occurs once for that given river station. It is better to focus on reoccurring errors or compounding errors first. The example on this slide shows a relatively small error at river station 259106* that grows to 0.4 ft in the next few time steps.

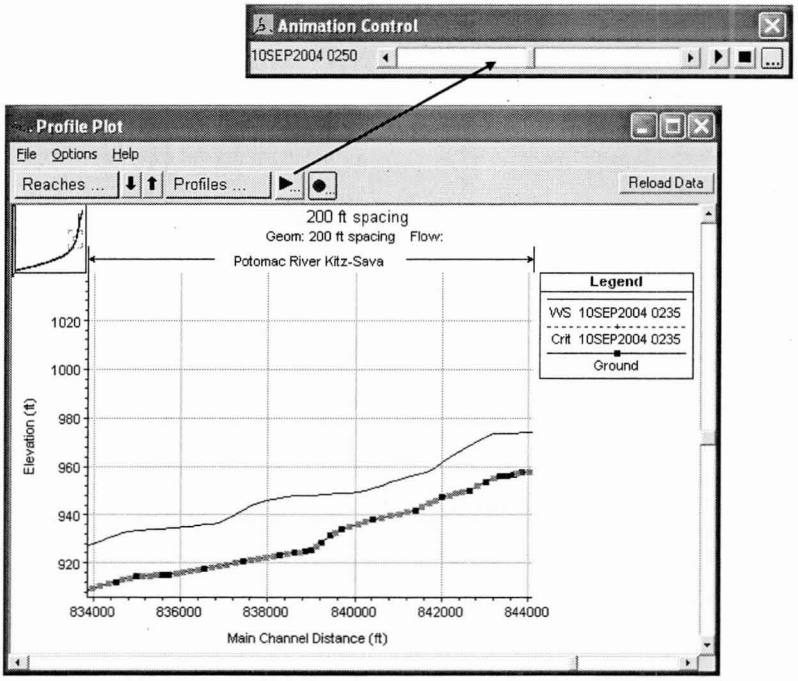
Profile Plot

- Great visual tool for finding problem areas.
- Use the "Animation" option to look for obvious instabilities. Zoom in to get a closer look.
- May need to refine the Detailed Output Interval to see where and when the instability occurs.
- When the first hints of an instability is revealed, click on that "node" and investigate further.

Hydrologic Engineering Center 31

The profile plot is typically the first graphical tool to use to try to pinpoint instabilities. Obvious errors are shown distinctly in this plot and you can see what is going on in the entire reach at the same time. Stepping through each profile using the animation tool allows you to see changes over time, including the progression of the flood wave as well as propagation of errors. The profile output is taken from the detailed output file. Therefore, it is sometimes necessary to refine the detailed output interval to adequately see the beginning of instabilities. The profile plot allows the user to click on a given node to determine its river stationing. Find the node where the instability first occurs and investigate further.

Profile Plot Animation

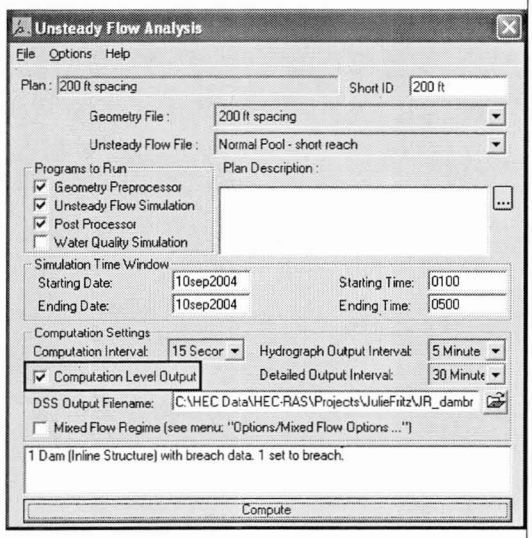


Hydrologic Engineering Center 32

The above slide was an animation of the profile plot, showing the progression of a model instability problem. The profile plot can assist you in locating where an instability is occurring and when. You may need to zoom in to get a closer look. You may also need to set the **Detailed Output Interval** to a smaller value and re-run the simulation in order to see what is happening at a finer time step increment.

Computation Level Output

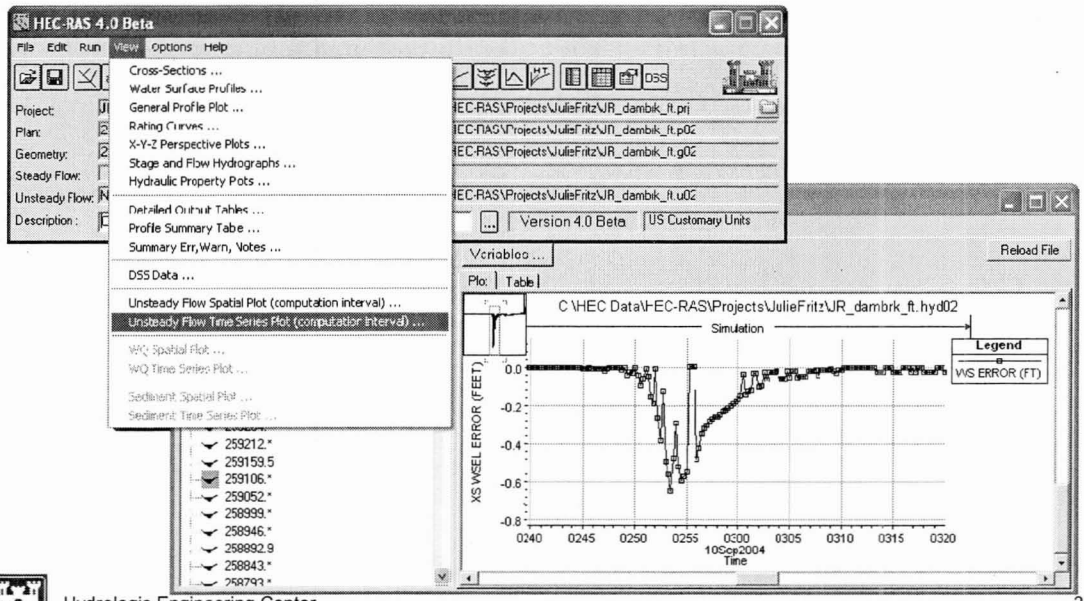
- Writes flow and stage at all locations to a separate file.
- Tools available from the View menu:
  - Spatial Plots
    - profile
    - schematic
  - Time Series plots
    - water surface, depth, flow
    - WS and flow errors
- Warning: Can create large output files when used with large data sets for long times



Hydrologic Engineering Center 33

When performing an unsteady flow analysis the user can optionally turn on the ability to view output at the computation interval level. This is accomplished by checking the box labeled **Computation Level Output** on the Unsteady Flow Analysis window (In the Computations Settings area on the window). When this option is selected an additional binary file containing output at the computation interval is written out. After the simulation the user can view computation level output by selecting either **Unsteady Flow Spatial Plot** or **Unsteady Flow Time Series Plot** from the **View** menu of the main HEC-RAS window.

Computation Level Output Visualization Tools



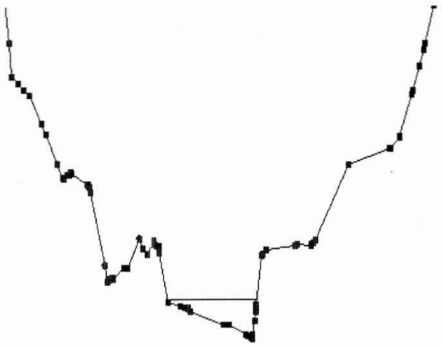
Hydrologic Engineering Center 34

Visualization of computation level output can be accomplished with either **Spatial Plots** or **Time Series Plots**. From the Spatial Plots the user can view either a profile plot, a spatial plot of the schematic, or tabular output. The user can select from a limited list of variables that are available at the computation level output. These are water surface elevation (XS WSEl); Flow (XS Flow); computed maximum error in the water surface elevation (XS WSEL ERROR); computed maximum error in the flow (XS FLOW ERROR); and maximum depth of water in the channel (DEPTH). Each of the plots can be animated in time by using the video player buttons at the top right of the window. This type of output can often be very useful in debugging problems within an unsteady flow run. Especially plotting the water surface error and animating it in time.

The other type of plot available at the computation interval output level is the **Unsteady Flow Time Series Plot.** When this option is selected the user will get a plot as shown in the Figure above. Some of the same options and variables are available for the Time Series Plots as were available for the Spatial Plots.

Cross Section Plot

- Can help spot isolated problems such as:
  - Incorrect Bank Station locations
  - Bad Manning's n Values
  - Bad Station-elevation points
- Can help spot transition problems
  - Contraction/Expansion Areas
  - Ineffective Flow Areas
  - Levees

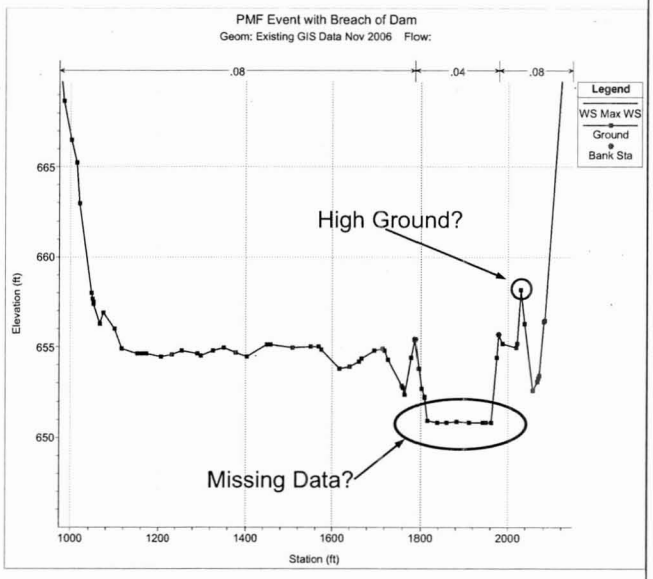


Hydrologic Engineering Center 35

Once a location for an instability is determined on the profile plot, the cross section plot can be used to investigate the cause of the instability. The cross section plot will show isolated problems such as incorrectly placed bank stations, poor n-values, and bad station-elevation data. In addition, scrolling through its neighboring cross sections can give you an idea of transition problems like contractions and expansions that occur to abruptly, poorly defined ineffective flow areas, or incorrectly handled levees or natural high ground spots.

Cross Section Plot

- Wide, Horizontal Beds
  - Estimated XS?
  - LIDAR, no bathymetry?
  - Prone to instabilities – High Area:Depth ratio
- High Ground
  - Levee Option
  - Ineffective Flows?
- Solutions?



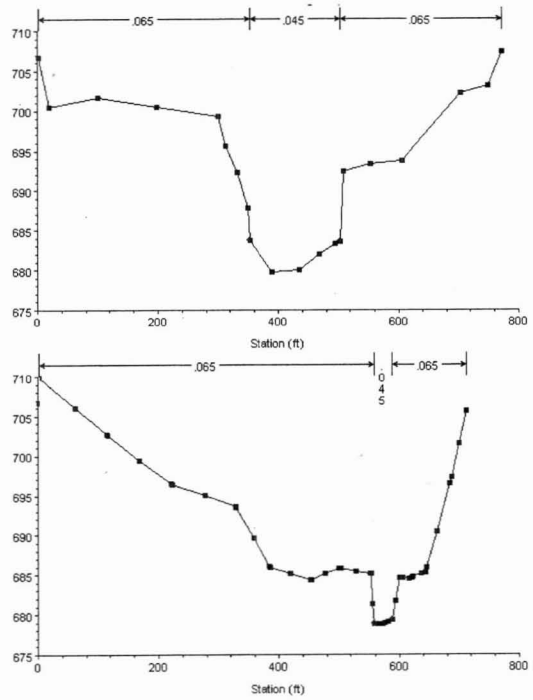
Hydrologic Engineering Center 36

Another typical source of instabilities occurs when the main channel has a wide flat bed. This is usually found when cross sections are approximated or when terrain data is used to develop cross sections exclusive of real bathymetric data. Many times reaches are developed in GIS using LIDAR data or other aerial means. These survey methods don't penetrate water surfaces so the main channel is left with a flat horizontal bed equal to the water surface elevation. For shallow streams in dam breach analyses, this is normally okay, since the dam break flood wave is usually much greater than the depth of water. However, wide flat stream beds lend to instabilities because at lower flows, the area to depth ratio is very high. Again this presents the same problem of a small increase in depth amounting to a large relative increase.

Additionally, in the cross section plot, high ground that is not appropriately accounted for can be detected and fixed to remove sources of instabilities. High ground can be modeled as levees or with ineffective flows to remove the abrupt changes in storage and conveyance when the high ground is overtopped.

Cross Section Plot

- Transitions
  - If sudden contraction or expansion occurs over a short distance, how can this be handled?
  - Ineffective Flow Areas
  - More Cross Sections
    - Interpolation



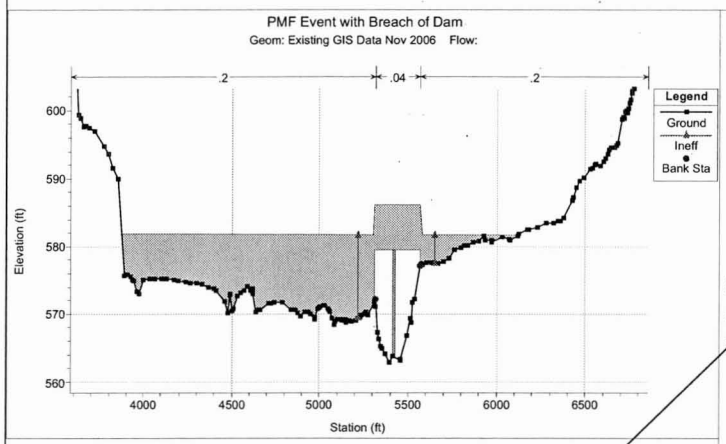
Hydrologic Engineering Center

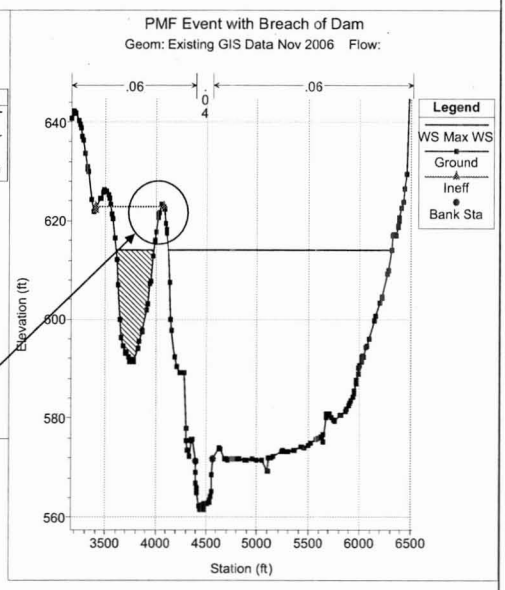
37

The example in this slide shows an abrupt transition from a wide main channel to a narrow main channel. If these cross sections are close enough, the flow man not be able to contract so suddenly and the approximate numerical methods may not be able to handle this situation. In this case, ineffective flow areas can be placed in the wide cross section to help smooth the transition from wide to narrow. If these cross sections are far enough apart, then perhaps additional interpolated cross sections are warranted.

Cross Section Plot

- Ineffective Flow Areas





- Levee or Ineffective Flow? Or Neither?

Hydrologic Engineering Center

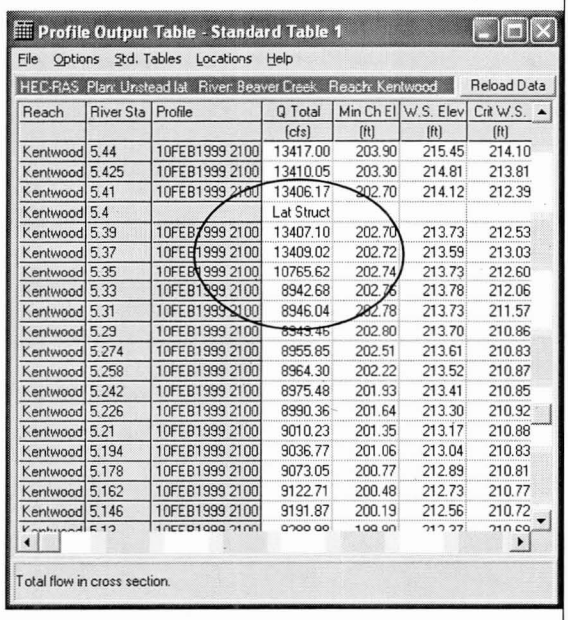
38

Ineffective flow areas are required up and downstream of bridges and culverts to properly define the contraction and expansion zones. Unsteady flow models, and particularly dam breach models, need these zones to be adequately defined. When the bridge is overtopped, the ineffective flow areas will turn off. This sudden and large increase in conveyance can cause model instability. One solution is to use very high Manning's n values (.2 to 1.0) in the ineffective flow zones, so when they turn off the increase in conveyance is not so great. This is also more physically appropriate as the cross sections just upstream and downstream can not flow completely freely because of the bridge embankment.

When an isolated high ground area is causing an instability problem, the user must decide if this high ground is better modeled with the levee option or with ineffective flow.

Profile Summary Tables

- Sometimes visual clues are not available. Tabular output help.
  - lateral inflow/outflow
    - Tributaries
    - Interaction with storage areas
  - Lateral structure flow
  - Inline structure flow
  - Flow inconsistency
    - Main channel to overbanks
  - Other internal boundaries
    - Groundwater



Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

HEC-RAS Plan: Unstead Iat River: Beaver Creek Reach: Kentwood Reload Data

Reach	River Sta	Profile	Q Total	Min Ch El	W.S. Elev	Crit W.S.
			(cfs)	(ft)	(ft)	(ft)
Kentwood	5.44	10FEB1999 2100	13417.00	203.90	215.45	214.10
Kentwood	5.425	10FEB1999 2100	13410.05	203.30	214.81	213.81
Kentwood	5.41	10FEB1999 2100	13406.17	202.70	214.12	212.39
Kentwood	5.4		Lat Struct			
Kentwood	5.39	10FEB1999 2100	13407.10	202.70	213.73	212.53
Kentwood	5.37	10FEB1999 2100	13409.02	202.72	213.59	213.03
Kentwood	5.35	10FEB1999 2100	10765.62	202.74	213.73	212.60
Kentwood	5.33	10FEB1999 2100	8942.68	202.76	213.78	212.06
Kentwood	5.31	10FEB1999 2100	8946.04	202.78	213.73	211.57
Kentwood	5.29	10FEB1999 2100	8943.46	202.80	213.70	210.86
Kentwood	5.274	10FEB1999 2100	8955.85	202.51	213.61	210.83
Kentwood	5.258	10FEB1999 2100	8964.30	202.22	213.52	210.87
Kentwood	5.242	10FEB1999 2100	8975.48	201.93	213.41	210.85
Kentwood	5.226	10FEB1999 2100	8990.36	201.64	213.30	210.92
Kentwood	5.21	10FEB1999 2100	9010.23	201.35	213.17	210.88
Kentwood	5.194	10FEB1999 2100	9036.77	201.06	213.04	210.83
Kentwood	5.178	10FEB1999 2100	9073.05	200.77	212.89	210.81
Kentwood	5.162	10FEB1999 2100	9122.71	200.48	212.73	210.77
Kentwood	5.146	10FEB1999 2100	9191.87	200.19	212.56	210.72

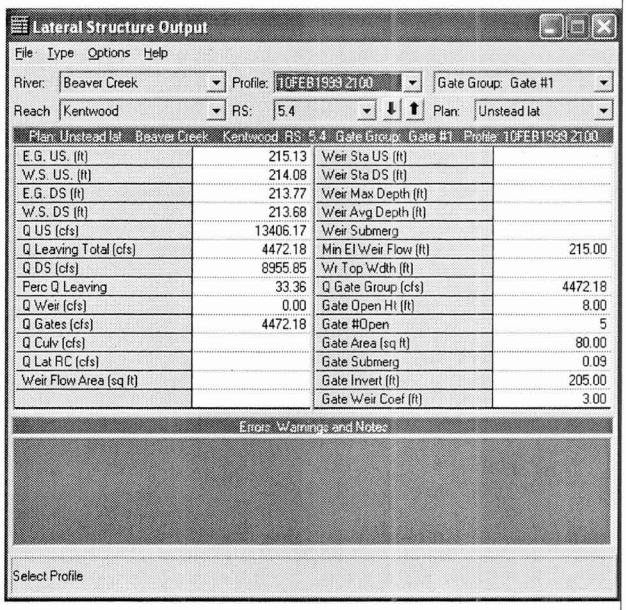
Total flow in cross section.

Hydrologic Engineering Center 39

Often times the graphical options alone are not adequate to determine the source of instability. Another option is to go to the profile output table and analyze values of hydraulic parameters from one cross section to the next or from one profile to the next. Problems that don't always show up graphically are lateral inflows and outflows, groundwater interaction and the effects of lateral structures. It is imperative that the important hydraulic parameters (flow, depth, area, storage) change as gradually as possible. Flow consistency between the overbanks and the main channel is also important.

Detailed Output Tables

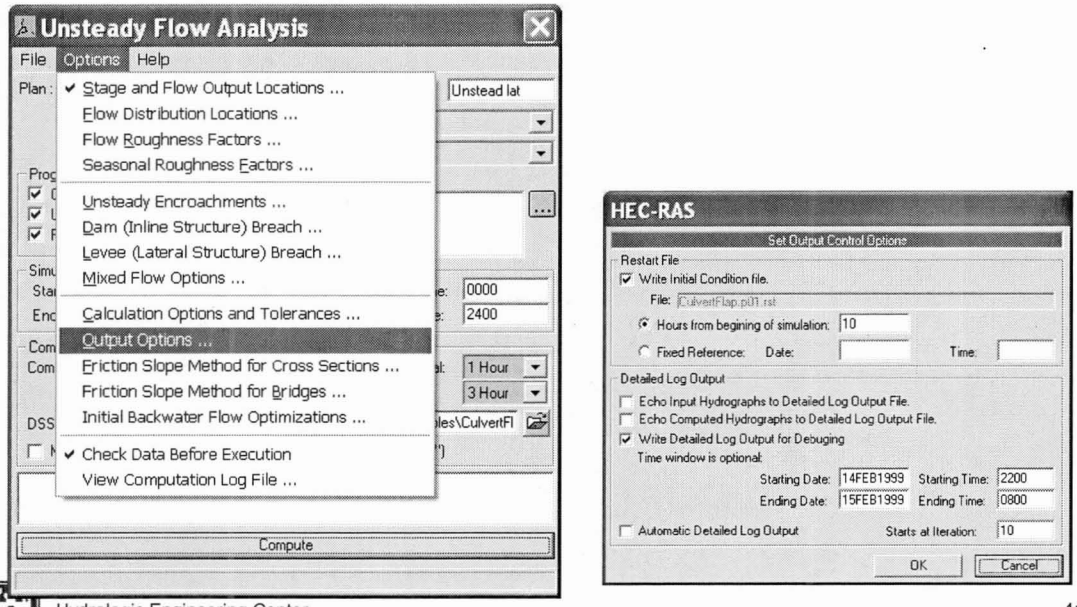
- Very good for looking at details of :
  - Inline Structures
  - Lateral Structures
  - Bridges/Culverts
  - Storage Areas
  - Pump Stations
  - Cross Sections



Hydrologic Engineering Center 40

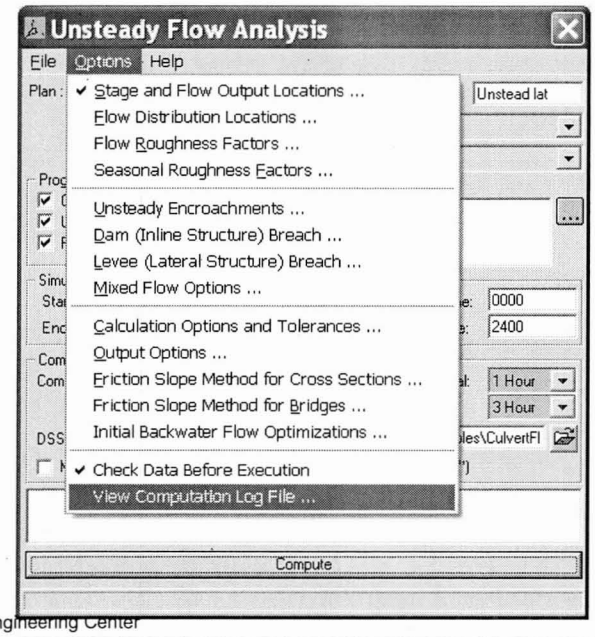
Detailed output tables are available for many types of nodes in HEC-RAS, including: cross sections, bridges, culverts, inline structures, lateral structures, storage areas, and pump stations. These detailed tables can be very helpful in seeing what is going on at that structure for a particular time step.

Turning on Detailed Log Output for Debugging



As shown in the figure above, the section at the bottom half of this editor is used for controlling the **detailed log output**. Three check boxes are listed. The first box can be used to turn on an echo of the hydrograph input to the model. This can be used to ensure that the model is receiving the correct flow data. The second check box can be used to turn on an echo of the computed hydrographs that will be written to the HEC-DSS. This is a good option for checking what was computed. However, if the user has selected to have hydrographs computed at many locations, this could end up taking a lot of file and disk space. The third check box is used to control the detailed output of results from the unsteady flow simulation. Selecting this options will cause the software to write detailed information on a time step by time step basis. This option is useful when the unsteady flow simulation is going unstable or completely blowing up (stopping). Checking this box turns on the detailed output for every time step. The user has the option to limit this output to a specific time window during the unsteady flow simulation. Limiting the log output is accomplished by entering a starting date and time and an ending date and time.

Viewing Detailed Log Output



Hydrologic Engineering Center 42

Viewing Detailed Log Output: After the user has turned on the detailed log output option, re-run the unsteady flow simulation. The user can then view the detailed log output by selecting **View Computational Log File** from the **Options** menu of the Unsteady flow simulation window. When this option is selected the detailed log output file will be loaded into the default text file viewer for your machine (normally the NotePad.exe program, unless you have changed this option within HEC-RAS).

What is found in the detailed Output

- DSS Data – shows all the data that was read from DSS.
- Unsteady Flow Computations Output – Detailed unsteady flow calculations:
  - Job control parameters
  - Initial conditions calculations
  - Detailed output for each time step
- TABLE Output – final hydrographs that are written to DSS

Hydrologic Engineering Center 43

The detailed log output file will contain the following output:

DSS Output: Shows all of the hydrograph data that will be used as input to the model, including data read from HEC-DSS.

Unsteady Flow Computations Output: Detailed unsteady flow calculations including:

Job control parameters

Initial conditions calculations

Detailed output for each time step

Table Output: Final computed hydrographs that are written to HEC-DSS.

Initial Conditions Output

Diamond.bco - Notepad

File Edit Format Help

Initial Conditions from Backwater

Diamond North

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
6.0	100.0	11.21		0.0000036	320.88	43.21	0.312	0.00000	T
5.8	100.0	11.20		0.0000027	355.63	44.00	0.281	0.00000	T
5.6	100.0	11.20		0.0000020	391.05	44.80	0.256	0.00000	T
5.4	100.0	11.20		0.0000015	443.55	100.40	0.225	0.00000	T
5.2	100.0	11.20		0.0000011	553.25	174.29	0.181	0.00000	T
5.0	100.0	11.20		0.0000008	720.29	230.22	0.139	0.00000	T
4.8	100.0	11.20		0.0000008	720.08	230.22	0.139	0.00000	T
4.6	100.0	11.20		0.0000008	719.88	230.22	0.139	0.00000	T
4.4	100.0	11.19		0.0000008	719.68	230.22	0.139	0.00000	T
4.2	100.0	11.19		0.0000008	719.47	230.21	0.139	0.00000	T
4.0	100.0	11.19		0.0000008	719.27	230.21	0.139		

Diamond Northwest

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
4.0	70.0	11.19		0.0000004	719.32	230.21	0.097	0.00000	T
3.8	70.0	11.19		0.0000004	719.24	230.21	0.097	0.00000	T
3.6	70.0	11.19		0.0000004	719.16	230.21	0.097	0.00000	T
3.4	70.0	11.19	-0.51	0.0000004	498.62	45.00	0.140	0.00000	T
3.395 Culvert									
3.39	70.0	11.00	-0.51	0.0000005	489.99	45.00	0.143	0.00000	T
3.35	70.0	11.00		0.0000005	489.99	45.00	0.143		

Diamond Northeast

Riv. Sta.	Flow	WSEL	Crit Depth	EG Slope	Area	Topwidth	Velocity	Error	Converged
3.9999	30.0	11.00		0.0000001	675.21	230.00	0.044	0.00000	T
3.77768	30.0	11.00		0.0000001	675.19	230.00	0.044	0.00000	T
3.55547	30.0	11.00		0.0000001	675.17	230.00	0.044	0.00000	T
3.33326	30.0	11.00		0.0000001	675.14	230.00	0.044	0.00000	T
3.11105	30.0	11.00		0.0000001	675.12	230.00	0.044	0.00000	T
2.88884	30.0	11.00		0.0000001	675.10	230.00	0.044	0.00000	T
2.66663	30.0	11.00		0.0000001	675.08	230.00	0.044	0.00000	T
2.44442	30.0	11.00		0.0000001	675.06	230.00	0.044	0.00000	T
2.22221	30.0	11.00		0.0000001	675.03	230.00	0.044	0.00000	T

44

The program lists the computed initial conditions from a backwater calculation for each of the river/reaches. They are generally listed in an upstream to downstream order. However, they are computed from downstream to upstream under the assumption of subcritical flow.

Example Detailed Time Step Output for cross sections

Beaver_spec_store.bco - Notepad

File Edit Format Help

```
                    Solving for T =    -3.250

Iter  River          Station    Elev       DZ      Storage          Zsa        DZsa    River          Station        Q
  0 Beaver Creek     5.0        210.53   0.51156   Bayou            206.13   0.01598 Beaver Creek     5.0         5358
  1 Beaver Creek     5.0        210.22  -0.43984   Bayou            206.13   0.00000 Beaver Creek     5.0         5538
  2 Beaver Creek     5.0        209.94  -0.39653   Bayou            206.13   0.00000 Beaver Creek     5.0         5700
  3 Beaver Creek     5.0        209.68  -0.37383   Bayou            206.13   0.00000 Beaver Creek     5.0         5805
  4 Beaver Creek     5.0        209.43  -0.35521   Bayou            206.13   0.00000 Beaver Creek     5.0         5883
  5 Beaver Creek     5.0        209.22  -0.30138   Bayou            206.13   0.00002 Beaver Creek     5.0         6041
  6 Beaver Creek     5.065*     211.02   0.62017   Bayou            206.13   0.00005 Beaver Creek     5.0         5909
  7 Beaver Creek     5.065*     214.64   5.17663   Bayou            206.13   0.00076 Beaver Creek     5.0         2843
WARNING!  EXTRAPOLATED ABOVE THE TOP OF THE PROPERTY TABLE AT XSEC(S):
      Beaver Creek    5.065*        214.642090
  8 Beaver Creek     5.0        207.13  -1.03604   Bayou            206.13  -0.00055 Beaver Creek     5.065*      3234
  9 Beaver Creek     5.0        206.04  -1.55023   Bayou            206.13   0.00057 Beaver Creek     5.0         2142
 10 Beaver Creek     5.0        204.72  -1.89683   Bayou            206.13   0.00053 Beaver Creek     5.0          930
 11 Beaver Creek     5.0        203.50  -1.73679   Bayou            206.13  -0.00003 Beaver Creek     5.065*      1564
 12 Beaver Creek     5.0        202.14  -1.94503   Bayou            206.13   0.00028 Beaver Creek     5.0          457
 13 Beaver Creek     5.0        200.64  -2.13693   Bayou            206.13   0.00038 Beaver Creek     5.0         -175
 14 Beaver Creek     5.0        199.25  -1.98779   Bayou            206.13  -0.00013 Beaver Creek     5.065*       802
 15 Beaver Creek     5.0        197.84  -2.01339   Bayou            206.13   0.00001 Beaver Creek     5.0           -1
 16 Beaver Creek     5.0        196.46  -1.97657   Bayou            206.13  -0.00006 Beaver Creek     5.0           88
 17 Beaver Creek     5.0        195.08  -1.97219   Bayou            206.13  -0.00002 Beaver Creek     5.0          126
 18 Beaver Creek     5.0        193.70  -1.96514   Bayou            206.13  -0.00002 Beaver Creek     5.0          155
 19 Beaver Creek     5.0        192.33  -1.95701   Bayou            206.13  -0.00002 Beaver Creek     5.0          184
 20 Beaver Creek     5.0        190.97  -1.94689   Bayou            206.13  -0.00002 Beaver Creek     5.0          218

!WARNING, USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR.  MINIMUM ERROR OCCURED DURING ITERATION  5.
```

Hydrologic Engineering Center 45

One way to find and locate potential stability problems with the solution is to do a search in the file for the word "**WARNING**". The user then needs to look at the detailed output closely to try and detect both where and why the solution is going bad. The variables that are printed out during the iterations are the following:

Iter	= Iteration Number.
River	= River Name of the location with the largest error in stage.
Station	= River station with the larges error in the calculated stage.
ELEV	= Computed water surface elevation at that river station.
DZ	= The "Numerical Error" in the computed stage at that location.
Storage	= Name of the storage area that has the larges error for this iteration.
Zsa	= Computed elevation of the storage area.
Dzsa	= The "Numerical Error" in the computed storage area elevation.
River	= River Name of the location with the largest error in flow.
Station	= River station with the largest error in the calculation of flow.
Q	= Computed flow
DQ	= The "Numerical Error" in the computed flow at the listed river station

Note: If the program goes to the maximum number of iterations, it will choose the iteration that had the minimum amount of error, set that as the solution for the current time step, and then go to the next time step.

Example Detailed Time Step Output for cross sections - Continued

beaver.bco - Notepad

File Edit Format Help

```
     COMPUTED STAGES AND DISCHARGES AT T =      0.1167 HOURS -  2/10/1999 AT 0007 HOURS

                Beaver Creek      Kentwood

Riv. Station      Z        Q       V        Riv. Station      Z        Q       V        Riv. Station      Z        Q       V
    5.99     213.03    599.    1.09            5.97     212.94    588.    1.22            5.951    212.83    579.    1.37
    5.93     212.71    571.    1.56           5.913     212.56    564.    1.79            5.894    212.38    558.    2.04
   5.875     212.19    552.    2.34           5.855     211.98    547.    2.65            5.836    211.74    543.    2.91
    5.81     211.52    540.    2.96           5.798     211.36    536.    2.45            5.779    211.24    532.    1.76
    5.76     211.17    528.    1.18           5.741     211.07    523.    1.64             5.72    210.91    521.    2.15
   5.703     210.75    519.    2.31           5.685     210.61    517.    2.30            5.666    210.48    516.    2.19
   5.647     210.37    515.    1.92           5.628     210.27    514.    1.62             5.61    210.21    513.    1.31
   5.593     210.13    512.    1.52           5.576     210.03    511.    1.80            5.559    209.93    511.    2.06
   5.542     209.85    510.    2.23           5.525     209.78    510.    2.24            5.508    209.72    510.    2.14
   5.491     209.67    510.    1.94           5.474     209.64    510.    1.76            5.457    209.61    510.    1.60
    5.44     209.58    510.    1.47           5.425     209.58    509.    1.11             5.41    209.57    509.    0.88
    5.39     209.54    509.    0.88            5.37     209.52    509.    1.11             5.35    209.48    509.    1.46
    5.33     209.40    510.    1.67            5.31     209.28    510.    1.67             5.29    209.15    510.    1.47
   5.274     208.95    510.    1.66           5.258     208.68    511.    1.95            5.242    208.29    511.    2.55
   5.226     207.85    511.    3.05            5.21     207.46    512.    3.29            5.194    207.15    512.    3.13
   5.178     206.95    513.    2.68           5.162     206.83    513.    2.16            5.146    206.75    514.    1.75
    5.13     206.71    514.    1.45           5.113     206.65    515.    1.50            5.097    206.59    515.    1.56
   5.081     206.53    516.    1.60           5.065     206.46    517.    1.63            5.048    206.39    517.    1.65
   5.032     206.31    518.    1.66           5.016     206.23    519.    1.66             5.0     206.13    519.    1.64


                         Solving for T =      0.133

Iter  River          Station    Elev       DZ        River          Station        Q        DQ
  0 Beaver Creek     5.99       213.07   0.03530   Beaver Creek     5.99         613       14
  1 Beaver Creek     5.0        206.13  -0.00050   Beaver Creek     5.93         584        1
```

Hydrologic Engineering Center 46

During the unsteady flow computations, the program will output detailed information for cross sections, bridges/culverts, inline weir/spillways, lateral weir/spillways, storage areas, and storage area connections. This information should be reviewed closely when the software is having stability problems. An example of the detailed output for cross sections is shown in the Figure above.

Trouble Shooting Workshop

Introduction

This workshop will help students learn how to trouble shoot instability problems associated with developing unsteady flow models with HEC-RAS. This workshop contains four separate instability problems, which are to be solved by the student. *While the data is from actual studies, the models and results of this workshop do not represent current or future conditions of the rivers.*

Problem Description 1

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Stability1**) with the title "**Stability Problem Number 1**" has been loaded on your workshop computers. The file contains all the data for this workshop. The geometry for this example includes the bridge crossing and interpolated cross sections.

Task 1

Load the project file and run the model. Examine the output for instabilities. Based on the output, geometry, boundary conditions, and initial conditions form a hypothesis on what is causing the instability. If time permits, revise the model based on your hypothesis until a stable computation is achieved.

Note on Viewing Profile Plot

Stepping through a time series animation with the **Profile Plot** is a good starting point to view where and when an instability occurred. When the **Profile Plot Window** is first opened the program will default to the maximum water surface. Unfortunately, when the program goes unstable the maximum water surface will usually have erroneous data. The bad data skews the dimensions of the plot, as seen in Figure 1.

The view needs to be corrected to analyze the instability. Under the **Options** menu of the **Profile Plot**, select **Profiles.** Next, clear the maximum water surface profile from the selection window and select the first profile. An example is shown in Figure 2. This should restore the view, then proceed through the time steps for analysis.

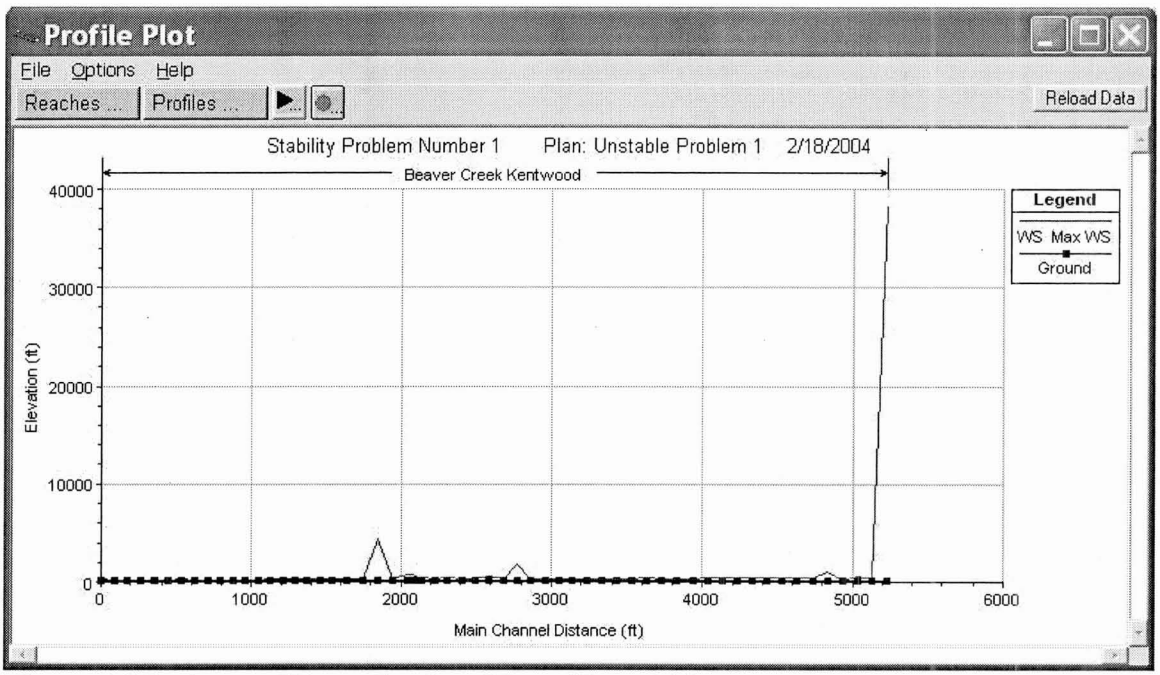


Figure 1 Maximum Water Surface Plot of Instability

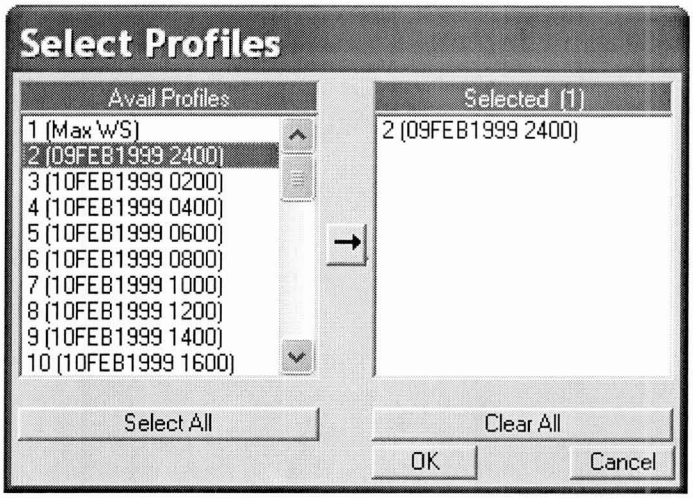


Figure 2 Initial Profile Selected

Questions

At what time step and at what location did the program begin to go unstable?

What caused the instability?

If you were able to fix the instability, how was this achieved?

Problem Description 2

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Stability2**) with the title "**Stability Problem Number 2**" has been loaded on your workshop computers. The file contains all the data for this workshop. The geometry for this example includes a bridge, interpolated cross sections, lateral weir with gates, and a storage area.

Note: The computation interval for this simulation is set to 5 minutes. Determine what is causing the instability without adjusting this interval.

Task 2

Load the project file and run the model. Examine the output for instabilities. Based on the output, geometry, boundary conditions, and initial conditions form a hypothesis on what is causing the instability. If time permits, revise the model based on your hypothesis until a stable computation is achieved.

Questions

At what time step and at what location did the program begin to go unstable?

What caused the instability?

If you were able to fix the instability, how was this achieved?

Problem Description 3

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Stability3**) with the title "**Stability Problem Number 3**" has been loaded on your workshop computers. The file contains all the data for this workshop. The geometry for this example includes the bridge crossing and interpolated cross sections.

Note: The computation interval for this simulation is set to 5 minutes. Determine what is causing the instability without adjusting this interval.

Task 3

Load the project file and run the model. Examine the output for instabilities. Based on the output, geometry, boundary conditions, and initial conditions form a hypothesis on what is causing the instability. If time permits, revise the model based on your hypothesis until a stable computation is achieved.

Questions

At what time step and at what location did the program begin to go unstable?

What caused the instability?

If you were able to fix the instability, how was this achieved?

Problem Description 4

The stream for this example is a section of Brunner River located near Jensen Town. The reach is a trapezoidal, concrete lined channel. A project file (**Stability4**) with the title "**Stability Problem Number 4**" has been loaded on your workshop computers. The file contains all the data for this workshop

Note: The computation interval for this simulation is set to 1 minute. Determine what is causing the instability without adjusting this interval.

Task 4

Load the project file and run the model. Examine the output for instabilities. Based on the output, geometry, boundary conditions, and initial conditions form a hypothesis on what is causing the instability. If time permits, revise the model based on your hypothesis until a stable computation is achieved.

Questions

At what time step and at what location did the program begin to go unstable?

What caused the instability?

If you were able to fix the instability, how was this achieved?

ADVANCED FEATURES For UNSTEADY FLOW With HEC-RAS

Cameron Ackerman, P.E.

Objectives:

- Mixed/Supercritical flow for Unsteady Flow
- Pumps for Unsteady Flow
- Dam Break and Levee Breaching for Unsteady Flow

Hydrologic Engineering Center 2

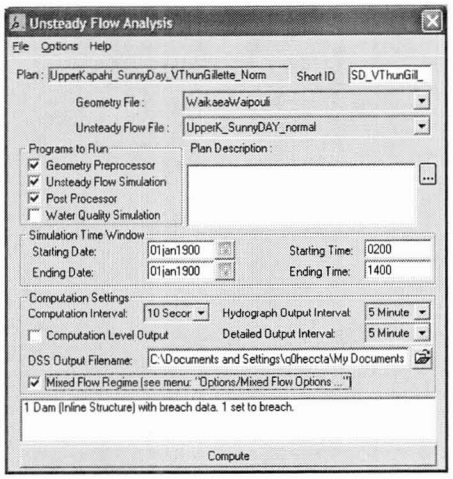
OBJECTIVES:

To know how to turn the mixed flow option on and be able to adjust the accuracy/stability parameters.

To know how to enter pump data for unsteady flow.

To learn how to enter and model Dam Break and Levee Breaching for unsteady flow.

Mixed/Supercritical Flow



- Checking Mixed Flow Regime Box
- Allows Subcritical/ Supercritical Mixed Flow
- Allows Critical Depth/ Hydraulic Jump Moving Interfaces

Hydrologic Engineering Center 3

When the mixed flow option is checked, the program will modify the unsteady flow equations. This can increases model stability when there is subcritical flow, supercritical flow, hydraulic jumps, and/or draw downs (sub to supercritical transitions). This option should only be selected if you actually have a mixed flow regime situation.

Local Partial Inertia (LPI) Technique

$$\sigma\left[\frac{\partial Q}{\partial t}+\frac{\partial\left(\frac{BQ^2}{A}\right)}{\partial x}\right]+gA\left(\frac{\partial h}{\partial x}+S_f+S_e\right)=0$$

$$\sigma=\begin{cases} F_t-F_r^m & (F_r\le F_t;\ m\ge 1)\\ 0 & (F_r>F_t)\end{cases}$$

Hydrologic Engineering Center 4

The methodology used for mixed flow regime analysis is called the **Local Partial Inertia (LPI)** solution technique (Fread, 1996). When this option is turned on, the program monitors the Froude number at all cross section locations for each time step. As the Froude number gets close to F_t (which has a default value of 1.0), the program will automatically reduce the magnitude of the inertial terms in the momentum equation (the terms in the large bracket on the top equation). Reducing the inertial terms increases the model's stability. When the Froude number is equal to or greater than F_t (1.0), the inertial terms are completely zeroed out and the model is reduced to a diffusion wave routing procedure. For Froude numbers close to F_t (1.0), the program will use partial inertial effects, and when the Froude number is low, the complete inertial effects are used. The reduction factor for the inertial terms is the power function shown in the lower equation where:

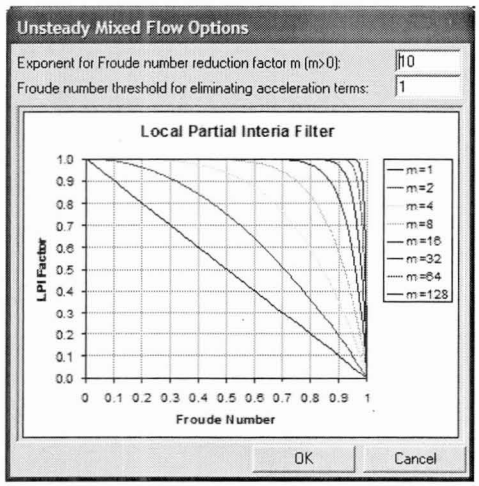
σ = LPI Factor to multiply the inertial terms by.

F_T = Froude number threshold at which the factor is set to zero. This value can range from 1.0 to 2.0 (default is 1.0)

F_r = Froude number.

m = Exponent of equation, which changes shape of the curve. This exponent can range between 1 and 128 (default value is 10).

LPI Filter



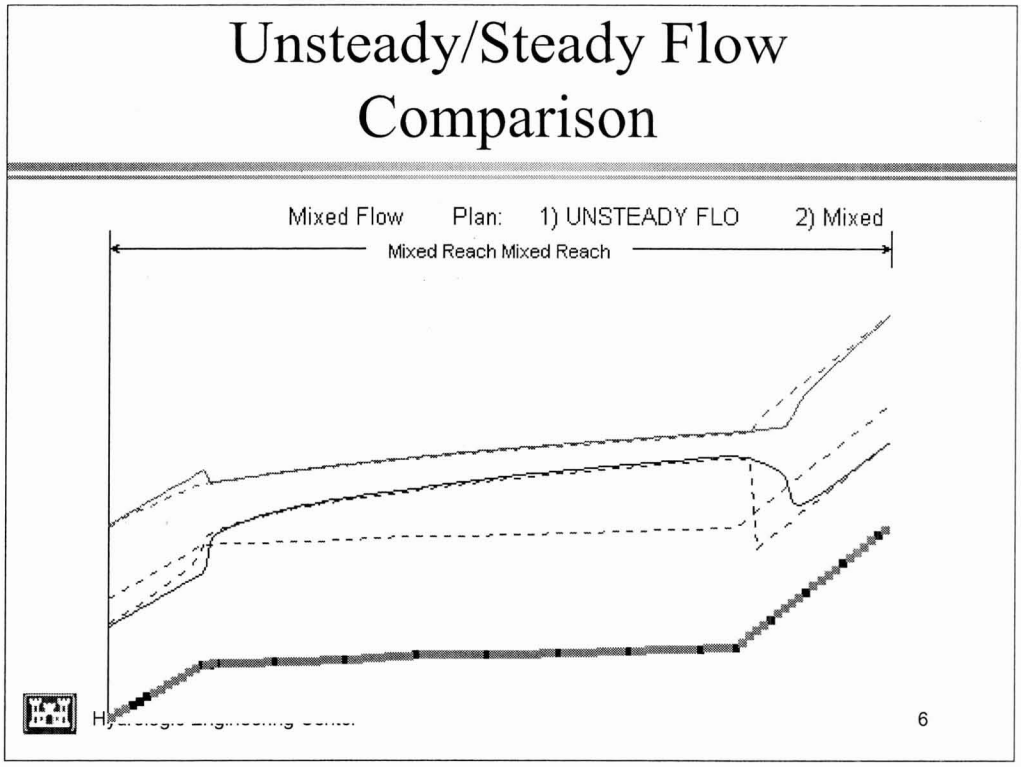
- Larger Exponent & Threshold:
  - Greater Accuracy
  - Less Stability
- Smaller Exponent & Threshold:
  - Greater Stability
  - Less Accuracy

Hydrologic Engineering Center 5

The above figure shows the LPI reduction versus Froude number for various values of m (for the graphical illustration, the Froude threshold is at 1.0). For the default value of m = 10, there is only a 10% reduction at a Froude number of 0.8 and a 30% reduction at a Froude number of 0.9. From there the reduction quickly approaches 100% at a Froude number of 1.0.

The diffusion wave model (100% reduction of the inertial terms) generally produces good results for steep streams. An example from Henderson's *Open Channel Flow* (1966), for a stream with a bed slope of 26feet per mile, found that the inertial terms were typically less than 1% of the value of the gravitational terms. Fread (1996) has reported generally small errors for the flow hydrographs with the use of the LPI technique, typically less than 2%. Errors reached the 4-6% level for very rapid dam-failure in channels of flat slope (for earth dam failures the unknowns of breach size and growth would produce much greater uncertainty, see later discussion).

The next graphic compares an HEC-RAS steady flow run (dotted lines) with an unsteady (but constant) flow run (solid lines). For the steep reach at the upstream end, the water surfaces and energies match closely. The unsteady flow has a drawn out hydraulic jump.



This may actually be a better representation since the steady flow assumption of the jump happening entirely between two adjacent cross sections is unrealistic when the cross sections are closely spaced . However, in the vicinity of the hydraulic jump the inertial terms can be quite large (rapid change in velocity with respect to distance), so the water surfaces in this region should be regarded as approximate at best. The subcritical region in the middle has a very good match. The unsteady flow dips back down to supercritical flow more quickly than the steady flow run--it even shows a slight rise in energy. The steady flow answer is more realistic. For the rapidly changing velocities associated with the draw down, the inertial terms are again significant (but ignored). Furthermore, the unsteady flow momentum equation may be slightly under computing the values associated with the quick contraction in flow area.

For more discussion see:

Fread, "An LPI Numerical Implicit Solution for Unsteady Mixed-Flow Simulation", ASCE North American Water & Environmental Congress, 1996.

U.S. Army Corps of Engineers, 1993. River Hydraulics, EM 1110-2-1416

U.S. A.C.E., 1993. Flood-Runoff Analysis, EM 1110-2-1417

Mixed Flow Considerations

- SUBCRITICAL
- For a model that is primarily Subcritical, Mixed Flow option can improve stability.

- SUPERCRITICAL
- For a model that is primarily Super-critical, unsteady flow less important. Steady flow analysis may be preferable.

Hydrologic Engineering Center 7

Even though the diffusion routing technique may have acceptably small errors for steep streams, it is not necessarily the best choice. Steep streams tend to have relatively little storage and less attenuation of the flood hydrograph than mild streams. In this situation, there may be little justification for the additional data and stability problems inherent in an unsteady flow model. A steady flow analysis may give comparable (or even better) answers with far less work.

Where the mixed flow option is most useful is in subcritical models that have short sections of supercritical flow. Without the mixed flow option, in order to maintain stability, the user would have to arbitrarily change the data in order to keep the flow subcritical.

For the mixed flow option, the user should choose the largest values of F_t (Froude Threshold) and m (the exponent) that allow the model to maintain stability.

Model stability may require more than just the mixed flow option.

Mixed Flow Stability

- Mixed Flow Not a Panacea
- Cross Section Spacing
- Time Step
- Stability
- Consider Rating Curve to "Drown Out" Supercritical Flow

Hydrologic Engineering Center 8

The mixed flow option does not guarantee that supercritical stretches will be stable.

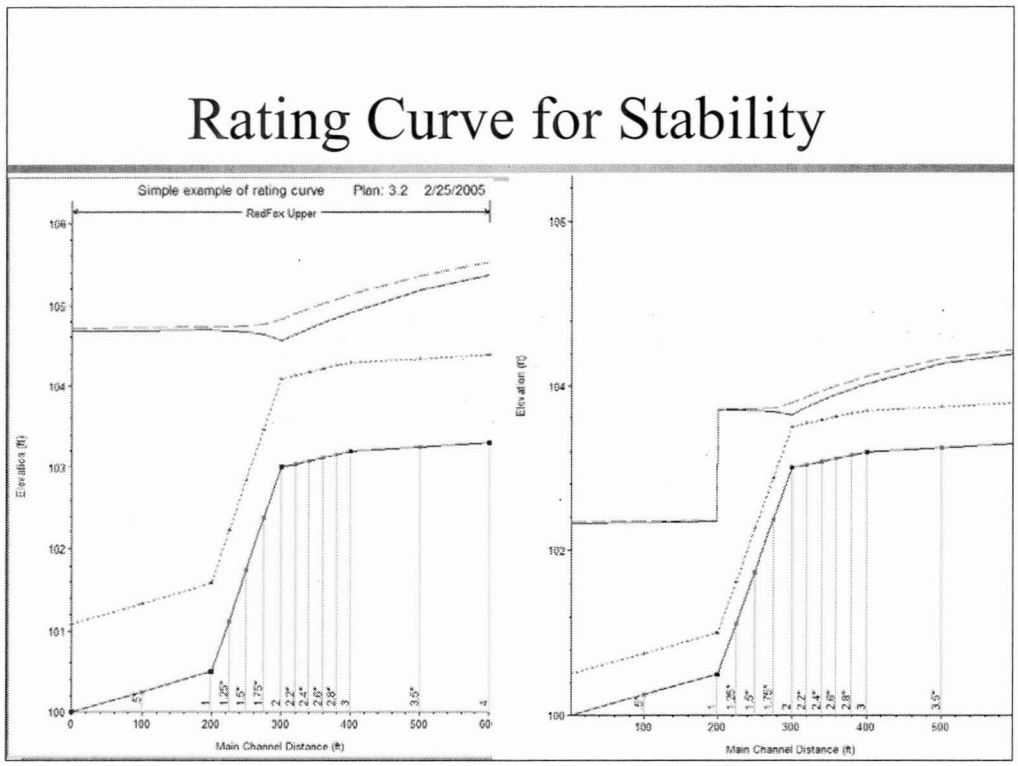
In areas that have rapidly changing depths and velocities, it is important to have close cross sections spacing. This includes the region upstream of critical depth (before the flow goes supercritical), the supercritical region itself, and the downstream hydraulic jump, if any.

In addition, a short time step may also be required.

A rating curve at a normal cross section is a feature added for version 3.1.2.

For a short and steep reach section, a rating curve can be used to "drown out" the supercritical flow. This may be useful in solving stability problems.

To do this, choose a cross section at the downstream end of the supercritical region. Make a copy of this cross section and put this new cross section immediately upstream of the existing cross section so that they are only a short distance apart (i.e., one foot). Add a rating curve to the upstream cross section (from the cross section editor, go to "options" and choose "add a rating curve"), by entering a series of flows and the corresponding water surfaces.

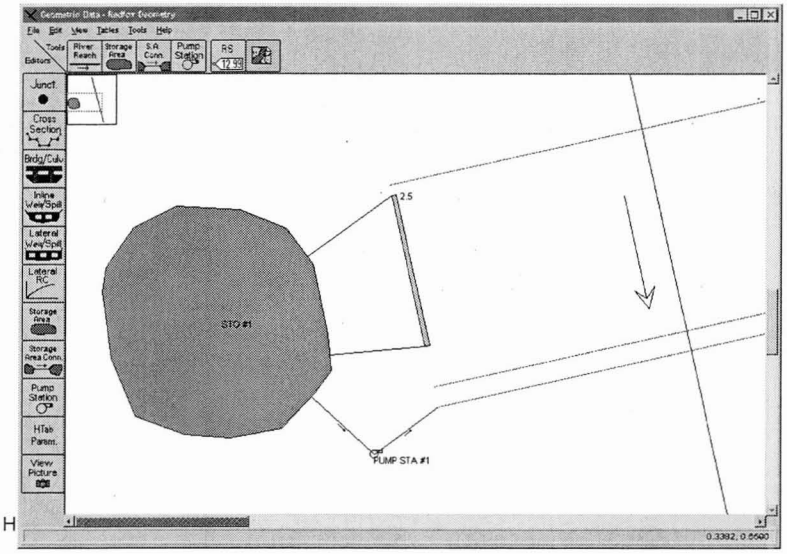


Care (and some experimentation) should be used in developing the rating curve. A higher rating curve will generally be more stable. However, a higher water surface could affect the results further upstream. For the profile on the right, the water surface almost touches critical depth. This should have minimal, if any, effect on the results further upstream. Steady flow experimentation may be quicker.

As an additional new rating curve option, the user can select the "Headwater Check" box on the rating curve. If this is done, the program will check to make sure that water surface in the upstream cross section ("headwater") is at least as high as the water surface in the downstream cross section ("tailwater"). If the rating curve returns a water surface lower than the tailwater, the program will ignore it and instead it will use the downstream water surface (because this water surface is "horizontally projected," as opposed to being computed, it is important that the two cross sections be identical and close together). This is useful if the water surface does not pass through critical depth at higher flows, such as the profile on the left. To do this, add additional points on the rating curve for high flows, but put in arbitrarily low water surfaces (Note: the program will extrapolate the rating curve, so make sure large enough flows are entered in the table!)

Concerns: the water surfaces in the formerly supercritical region are obviously "fake," but make sure that any region that is affected upstream is not used as an "official" water surface; the location of the hydraulic jump is no longer apparent—in reality the supercritical flow may continue downstream of the rating curve; short cross section spacing, a short time step, and even mixed flow may still be required.

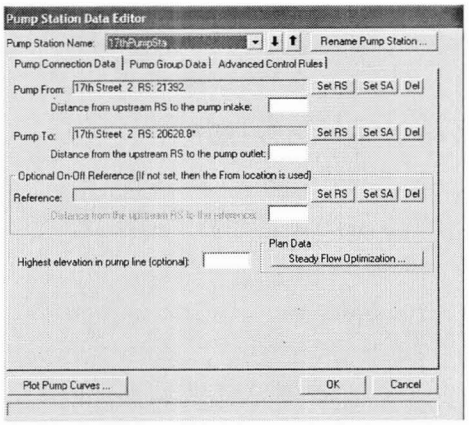
Pump Stations



H 10

To add a pump station, first click on the pump station button on the top row. Next, by clicking on the schematic, a pump station can be added wherever desired. When a pump station is added, the GUI will ask for a pump station name. After typing in the name, the pump station data can be entered by clicking on the pump station button on the left hand side and choosing the appropriate pump station from the list (if the geometry has more than one pump).

Pump Station Data Editor



- Set Connection info
  - Add Distance D/S if pump Inlet/Outlet not at exact cross section
- Optional On-Off Reference
- Optional Highest Elevation in Pump Line

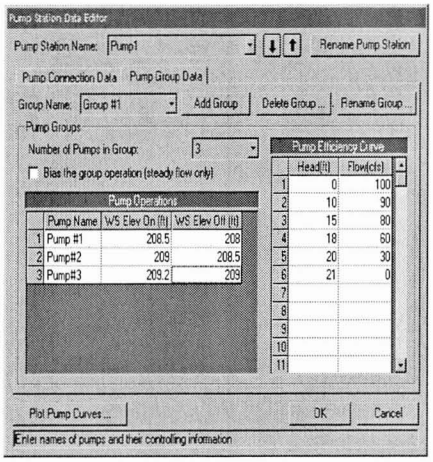
Hydrologic Engineering Center 11

By clicking on the <SET RS> buttons or the <SET SA> buttons, the pump can be connected from a cross section or storage area and then to a cross section or storage area. If the pump inlet or outlet is not located right at a cross section, a distance downstream can be added. The program will take this distance into account when computing the appropriate headwater or tailwater.

By default, the water surface at the inlet of the pump ("Pump From" location) is used to trigger the pump control (see next slide). The option "On-Off Reference" can be set to monitor an alternate water surface for triggering the pumps.

If the pump has a minimum elevation it has to pump to, the optional Highest Elevation in Pump Line can be entered. The program will use this elevation or the tailwater, whichever is greater, when computing the head that the pump has to pump against.

Pump Group Data

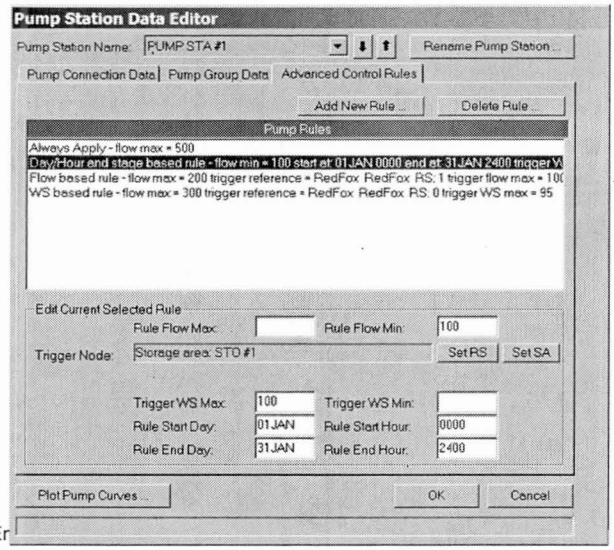


- Pump Efficiency Curve for each Pump Group
- Multiple Pumps within each Pump Group
  - Same Pump Curve
  - Different pump-on/ pump-off water surfaces

Hydrologic Engineering Center 12

A pump station can have multiple pump groups and each pump group can have multiple pumps. (This is analogous to how gates and culverts are organized.) All the pumps in a pump group must have the same Pump Efficiency Curve. Each individual pump can have its own water surfaces for turning on and off.

Advanced Control Rules



Clicking the <Advanced Control Rules> on the pump editor brings up the above editor.

The rules narrow the range of flow that is allowed from the pump by applying conditional max and min flows.

Rules are added by clicking the <Add New Rule> button and then selecting the rule type (next figure).

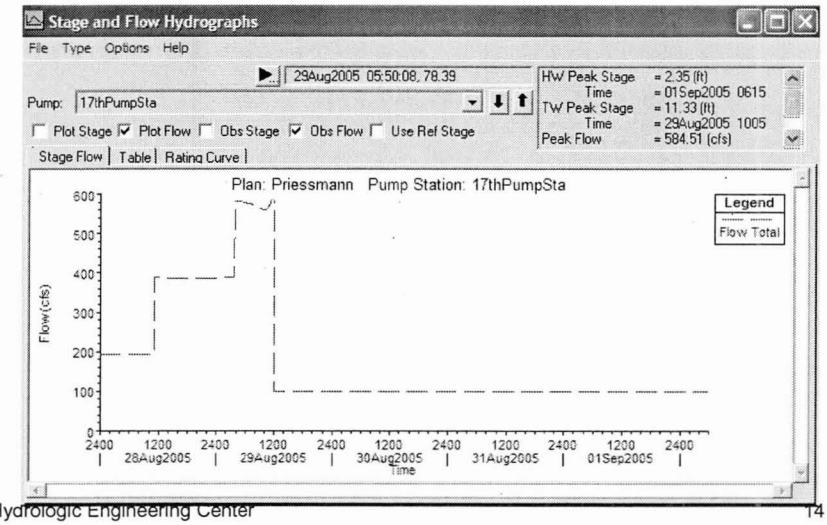
The first rule is always applied. It puts a maximum of 500 cfs from the pump. So the pump will never pump more than this amount.

The second rule (highlighted rule in figure) only applies in the month of January and only if the WSEL in storage area STO #1 is above 100 ft. If both of these conditions are met, the pump will have a minimum flow of 100. As currently programmed, this 100 cfs is a true minimum. Even if the pump curve shows that the pump can't pump this much based on current head, the pump will still have a minimum flow of 100 cfs.

The third rule lowers the maximum flow to 200 cfs if the flow at river station 1 is 100 cfs or more.

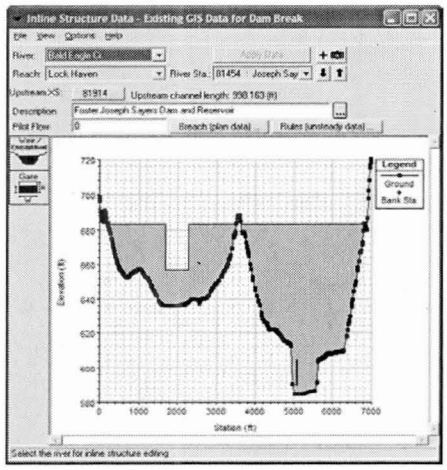
The fourth rule lowers the maximum flow to 300 cfs if the stage at river station 0 is 95 feet or more. If the maximum is already at 200 cfs (because of rule 3), then this rule has no effect. It won't override a maximum limit computed by previous rules.

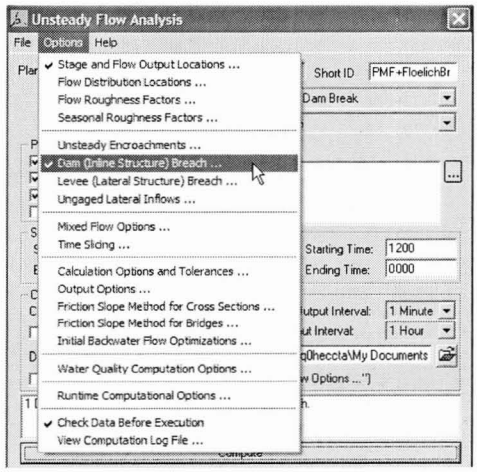
17th Street Pump Station Hydrograph



Hydrologic Engineering Center 14

Dam & Levee Breach





Hydrologic Engineering Center 15

A dam breach can be performed on an inline weir and a levee breach can be performed on a lateral weir. The breach editor can be accessed from either the unsteady flow analysis editor or by clicking the breach button on the inline or lateral weir editor. It is usually more convenient to just click on the button on the appropriate geometry editor, but the user should keep in mind that breach information is stored with the plan data not the geometry data. For instance, to compare a "breached" with a "not breached" run, the project should have two plans, but they can (and in many cases should) have the same geometry.

The dam breach data (next slide) is as follows:

Breach This Structure - This check box is used to decide if the program will perform the breach or not. This box was added to allow the user to turn certain breaches on or off, without loosing the user entered breach information.

Center Station - The centerline stationing of the final breach.

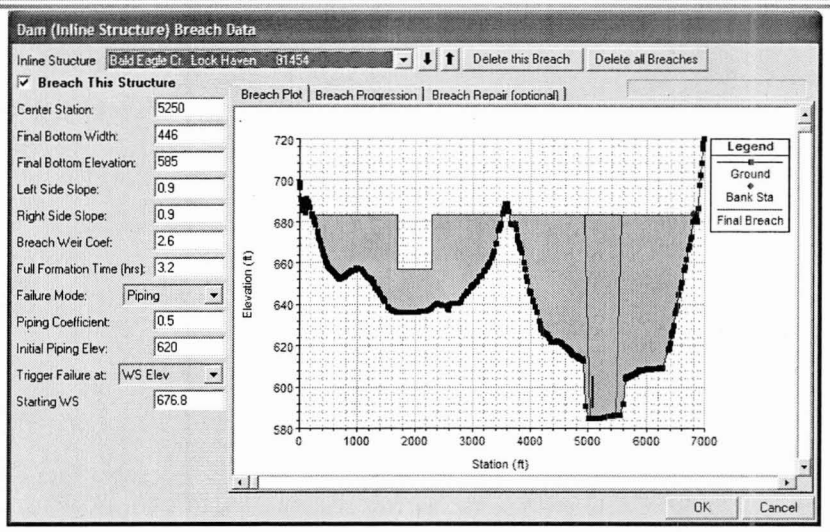
Final Bottom Width - The bottom width of the breach at its maximum size.

Final Bottom Elevation - The elevation of the bottom of the breach after it has been fully developed.

Left Side Slope - This is the left side slope of the trapezoidal breach.

Right Side Slope - This is the right side slope of the trapezoidal breach.

Dam Breach Data



Hydrologic Engineering Center 16

Full Formation Time (hrs) - The breach development time in hours.

Failure Mode - This option allows the user to choose between two different failure modes, an Overtopping failure and a Piping failure.

Piping Coefficient - If a piping failure mode is selected, the user must enter a piping coefficient.

Initial Piping Elev. - If a piping failure mode is selected, the user must enter an initial piping elevation.

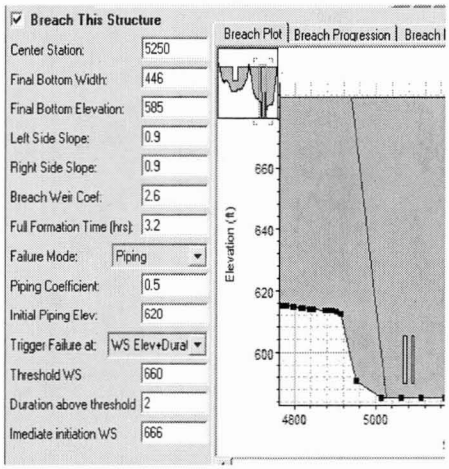
Trigger Failure At - This field is used to select one of three trigger methods for initiating the breach. The three trigger methods are a water surface elevation, a specific time and date, and a water surface elevation plus a duration.

Starting WS - If the user selects water surface elevation for the failure trigger mode, then this field must be entered. This field represents the water surface elevation at which the breach should begin to occur.

Start Date - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the date at which the breach will begin to occur.

Start Time - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the time at which the breach will begin to occur.

Breach Growth



- Trigger Methods
  - Water Surface
  - Water Surface plus Duration
  - Set Time
- Breach Can Grow Without Water
- Breach Can Grow Below Cross Section

Hydrologic Engineering Center 17

Threshold WS - If the user selects water surface elevation plus duration for the failure trigger mode, then this field must be entered. This field represents the water surface elevation at which the duration is triggered ("the clock starts ticking").

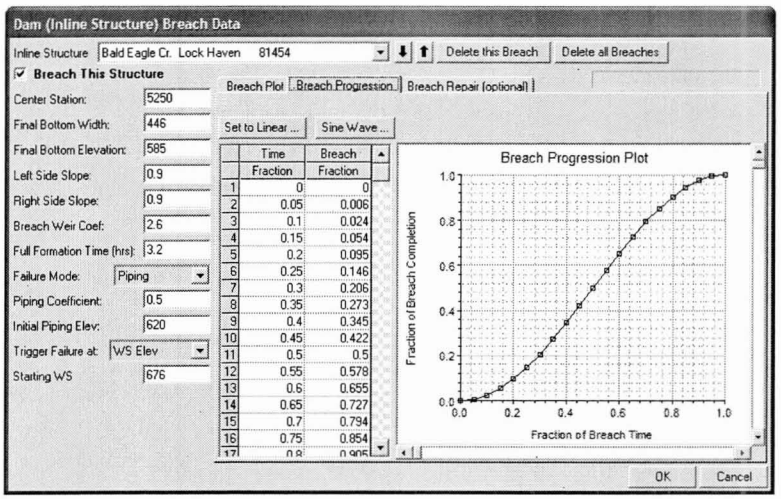
Duration above threshold - If the user selects water surface elevation plus duration for the failure trigger mode, then this field must be entered. This field is used to enter the amount of time that the water surface must be above the threshold for the breach to begin. In the above example, if the water surface is higher than 660 feet for 2 hours, then the breach will be triggered.

Immediate initiation WS - If the user selects water surface elevation plus duration for the failure trigger mode, then this field must be entered. This field is used to enter a water surface at which the breach will immediately begin to occur (regardless of the amount of time).

Note: once the breach has been triggered, it will continue to grow even if the water surface drops below the invert of the breach. The breach will reach its maximum size, based on the formation time, regardless of the hydraulics.

The original cross section shape does not limit the maximum size of the breach. The breach will reach the full size as entered by the user (and outlined in red on the plot) even if this is below (or past the edge) of the cross section.

Non-Linear Breach Growth

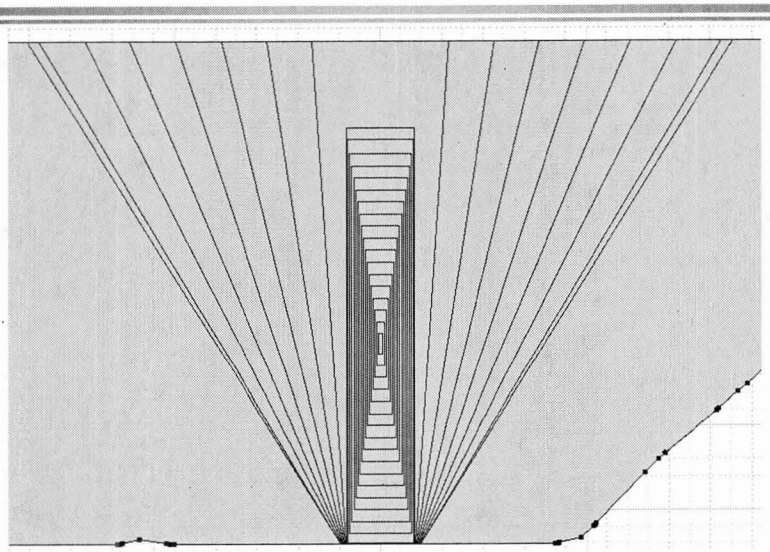


Hydrologic Engineering Center 18

By clicking on the <Breach Progression> button, the above editor will be shown. By default, the breach growth is assumed to be linear from start to maximum size (Full Formation Time). However, a non-linear breach growth curve can be entered. This is done by entering a Time Fraction (from zero to 1.0) and a Breach Fraction (from zero to 1.0). The user-entered data is plotted in the graphic next to the table. The breach progression curve is then used during the breach formation time to adjust the growth rate of the breach.

In addition to adjusting the curve in order to have a more realistic breach growth, the curve can sometimes be used in order to improve stability. If the program is having stability problems during part of the breach growth, the rate of growth during that time can be reduced by adjusting the curve.

Breach Growth Geometry

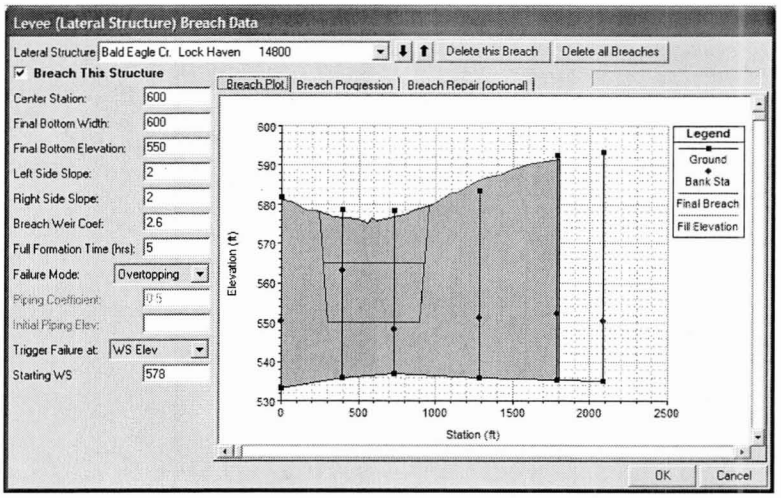


Hydrologic Engineering Center 19

A piping breach starts out as a point—at the centerline station and elevation as specified by the user. Its growth is modeled as a rectangle.

When the top of the piping breach is greater than the water surface, it is assumed that the levee sluffs and the breach is modeled as an open breach. The side slopes are gradually increased until they reach the user specified values.

Levee Breach Data



Hydrologic Engineering Center 20

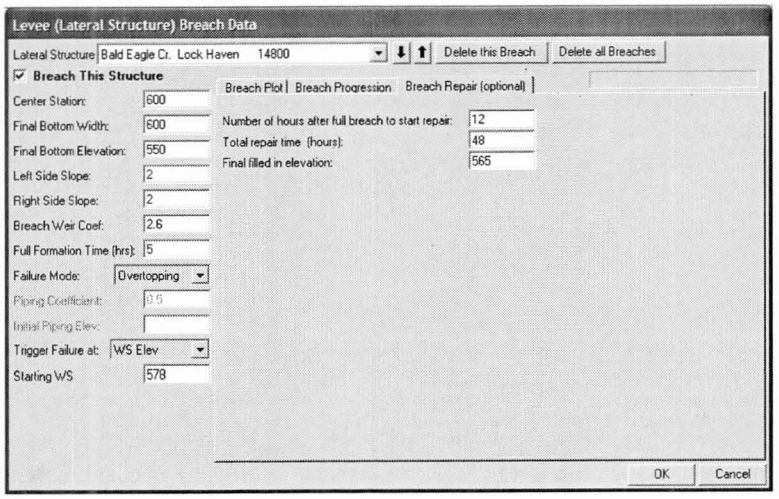
This option is very similar to the Dam Break option described previously. The only difference is that the breaching is performed on a levee. The options and data entered to describe the breach is the same as a Dam Break.

In order to use this option, the user must first define the levee as a lateral weir within HEC-RAS. The lateral weir profile is used to describe the top of the levee along the stream both at and between the cross sections. Second, a weir coefficient is entered for calculating the flow that may go overtop of the levee if the water surface gets high enough. Entering breach data for the levee can be accomplished from the lateral weir editor or from the **Levee (lateral weir) Breach** option from the unsteady flow simulation window. The levee breaching data is stored as part of the unsteady flow plan file, just as it is for a dam break. When the levee breach option is selected, a breach editor will appear as shown above.

The information entered in the fields is the same as for a Dam Break.

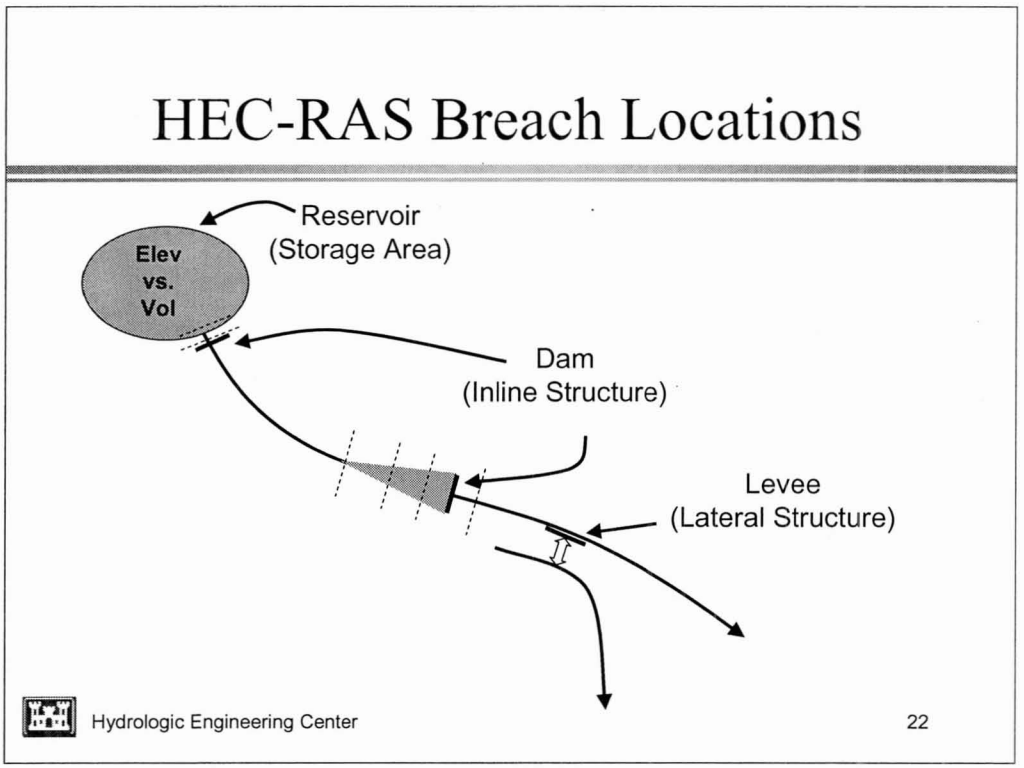
A levee breach can grow vertically below the level of the original cross section. However, (unlike a dam breach) it cannot grow horizontally past the edge of the original lateral weir. Again, the red outline will show the final shape of the breach.

Levee Breach Data



Hydrologic Engineering Center 21

A levee repair option is available by clicking on the <Breach Repair> button. Enter the number of hours after, the breach has fully formed, that the repair is to start. The total time for repair to take place and the elevation that the breach is repaired to. The elevation that the breach is repaired to is shown on the levee breach schematic (previous slide).



The formation of a breach can be simulated in two types of structures: inline and lateral. The parameters and hydraulic computations for the breaches are the same. A reservoir is usually modeled as "inline"; that is, its geometry is described with cross sections. This allows dynamic routing of the flood wave through the reservoir.

Alternatively, the reservoir can be described as a "storage area" for which the geometry is described by an elevation-volume relationship. Level-pool hydrologic routing within the reservoir is done for this situation. (See Goodell[1])

There must be at least two cross sections between the storage area and the dam. The first cross section is associated with the storage area and the second is associated with the inline structure that represents the dam. These two cross sections should be large, "reservoir-sized" cross sections.

A lateral structure may be a levee that overtops and does not fail, or possibly fails (depending on user input data). The levee is connected to a neighboring reach or storage area.

[1]Goodell, Christopher R. "Dam Break Modeling for Tandem Reservoirs – a Case Study using HEC-RAS and HEC-HMS," Presented at the ASCE World Water & Environmental Resources Congress 2005, EWRI, Anchorage, AK, 15-19 May 2005.

Estimating Dam Break Parameters

- USBR Dam Safety Research Report!
- Froehlich (1995b)
  - Earth dams
- MacDonald and Langridge-Monopolis
  - Earth dams
- Van Thun and Gillette (1990)
- Breach

Hydrologic Engineering Center 23

Determining the size and growth rate for breaches is a very inexact science. The Bureau of Reclamation's "Dam Safety Research Report," offers an excellent literature review of the subject (available online).

MacDonald and Langridge-Monopolis developed best-fit and envelope curves for both breach parameters and peak outflow from breached, earthfill dams as a function of a breach formation factor.

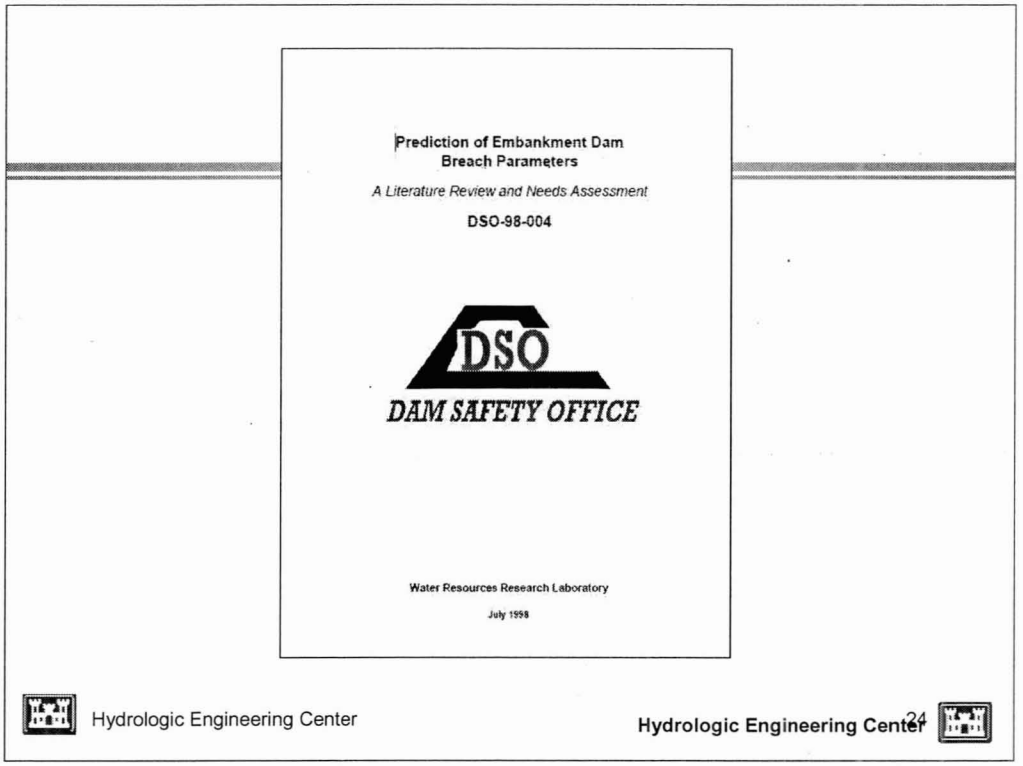
Froehlich developed a best-fit regression equation for peak flow based on a breach formation factor.

Breach is a physically based model that uses sediment transport equations.

The use of several different procedures, to get a range of values, is generally recommended.

Reference:

Water Resources Research Laboratory, "Prediction of Embankment Dam Breach Parameters", Dam Safety Office, Bureau of Reclamation, 1998.



Reference for breach parameters:

Wahl, Tony L., "Prediction of Embankment Dam Breach Parameters – A Literature Review and Needs Assessment," DSO-98-004, Dam Safety Research Report, U.S. Department of the Interior, Bureau of Reclamation, Dam Safety Office, July 1998.

(http://www.usbr.gov/pmts/hydraulics_lab/pubs/PAP/PAP-0735.DSO-98-004.pdfhttp://www.usbr.gov/pmts/hydraulics_lab/pubs/PAP/PAP-0735.DSO-98-004.pdf)

Castle Lake Example

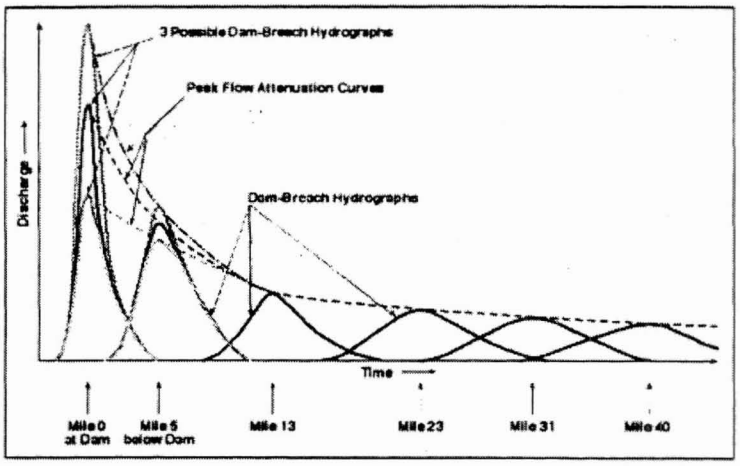
	Width	Time	Peak Q
Breach	480	0.25	1,180,000
Froehlich	305	0.36	761,300
M and L-M	25	0.50	147,600

Hydrologic Engineering Center 25

The above example is taken from a study HEC did on a debris avalanche dam that formed after the eruption of Mount St. Helens.

The MacDonald and Langridge-Monopolis equations probably do not give the most appropriate results for this situation, since their data did not include any landslide or debris blockage dams. However, even on a more typical embankment dam, breach parameters can vary considerably between different methodologies.

Dambreak Floodwave Progression



Hydrologic Engineering Center 26

How important the breach parameters are depend largely on how far downstream from the dam the study area is located. At the toe of the dam, the computed peak flow often varies wildly within the assumed limits of the breach parameters. However, as the study area is moved downstream, the breach parameters have less and less effect on the flow hydrograph. There are two main reasons for this. First, the total volume of water in each of the different hydrographs is basically the same (being the stored water behind the dam). Second, as the hydrographs move downstream, a sharp hydrograph will attenuate much more quickly than a flat hydrograph. Hydrographs from different assumed breach parameters can converge to produce the same peak flow (and for that matter, the same rise and fall) in a surprisingly quick distance. In the above example (Fread), the hydrographs have substantially converged within five miles and are indistinguishable by mile 13. It is still recommended that the user perform a sensitivity analysis on the breach parameters for all dam break studies.

Levee breaches in general and dam breaks in particular have rapidly varying water surfaces and flows. Because of this, a small time step is required to maintain stability and accuracy. Often on the order of a few seconds for a dam break problem. (See next slide).

Dam & Levee Breach Stability Considerations

- Small Time Step – Important!
- Appropriate Cross Section Spacing Downstream of Dam Breach (Courant <1.0)
- Increased Manning's n values just downstream of Dam
- "Critical" vs. "Total" Breach Time
- Non-Linear Growth usually more accurate
  - adjust, if needed
- "Mixed Flow" On (only if needed)

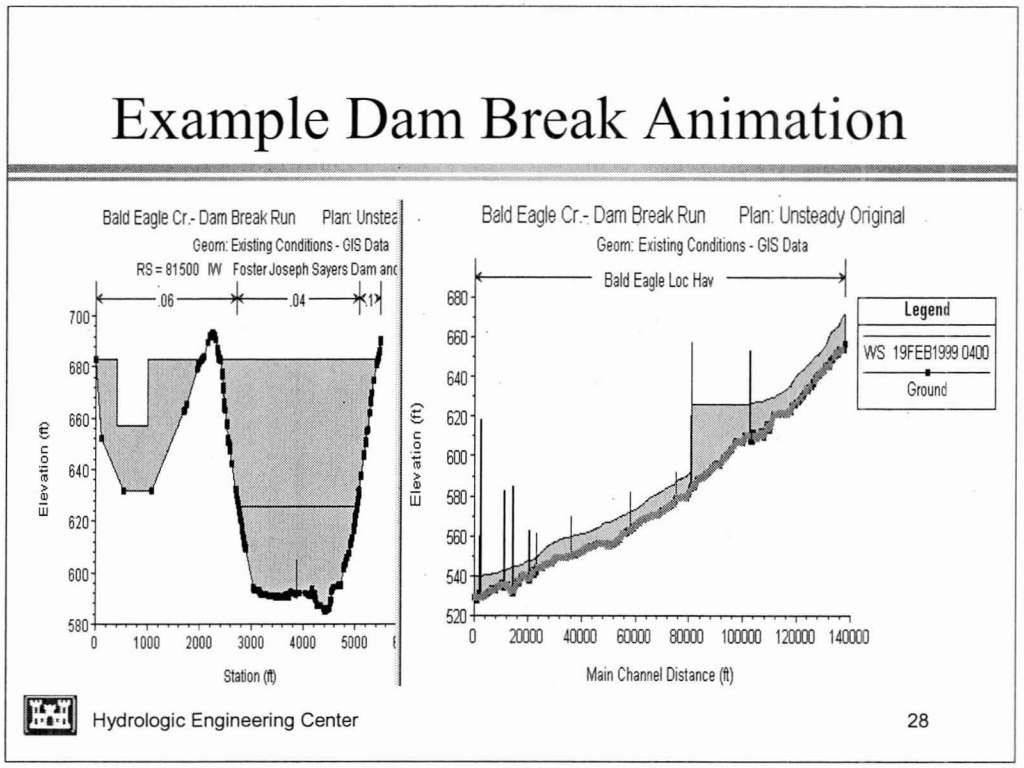
Hydrologic Engineering Center 27

Cross sections that are spaced too far apart below a dam break can cause stability and accuracy problems. However, the Courant condition should be kept close to or less than one (e.g., if the velocity is 20 feet/second and a ten second time step is being used, the cross section spacing should be no closer than around 200 feet).

Dam breaks are usually associated with large amount of sediment, debris and turbulence. The Manning's n value should be increased (particularly at cross sections near the dam) to account for these affects.

The reported breach time, especially for piping initiated breaches, often includes a significant period of time when the flow is, relatively speaking, a trickle. This time should not be included in the "Critical Breach Time" unless the user makes appropriate adjustments to the non-linear growth time.

If the dam break has stability problems, turning the "Mixed Flow" option on may help (it may help even if the flow is not going supercritical). This may reduce the accuracy of the water surfaces, particularly immediately downstream of the dam. However, this error is usually very minor compared with the error caused by the unknowns in the breach parameters.



Getting a dam breach model to be stable can be tricky. However, it is usually possible with enough adjusting. While close cross sections and small time steps usually work best, there are occasions when very short cross section spacing and tiny time steps can make stability worse. Trying a range of parameters is suggested.

Under the animation option in HEC-RAS, an animation of the profile plot or the cross section plot can be written out as an AVI.

As a final note on dam breaching, Dr. Gee and Gary Brunner have submitted a paper for publication, "Dam Break Flood Routing using HEC-RAS and NWS-FLDWAV." In it, they conclude that the computed breach outflow hydrograph is similar for the two models. The discrepancy being mostly due to the interpretation and approximation of the river geometry.

DAM AND LEVEE BREAK WORKSHOP

Introduction

This workshop will help students learn how to use HEC-RAS to: enter and edit dam and levee breach data; perform the unsteady breach run; make adjustments for stability; and review and understand the pertinent dam and levee breach output.

Problem

For this workshop we will be using the Bald Eagle data set. A project file (BaldEDmbrk.prj) with the title "Bald Eagle Cr.- Dam Break Run" has been loaded on your workshop computers.

A dam, located at river station 81500, is being modeled as an inline weir. The given unsteady flow hydrograph causes the dam to spill during the peak flows. For this flood event, a piping breach will be added. The floodwave from the dam failure causes the downstream levee to be overtopped. A lateral weir overtopping breach will be added to the upstream levee section, and then the lateral weir will be partly rebuilt using the levee repair option.

Students will be required to add the dam breach to the inline weir. The stability and accuracy of the resulting breach should be checked and any minor modifications performed. Next, the downstream lateral levee breach should be added. The following is a summary of the required tasks for each group:

1. Open the project, open the current plan and then save the plan with a new name (use the "save as" command). Give the new plan a descriptive name such as, "Dam Break."

2. Enter the dam break information, save the plan data, and perform the unsteady flow computations.

3. Review the results.

4. Save the current "Dam Break" plan with a new, descriptive name such as, "Dam Break 10s."

5. Change the time step, save the plan, and perform the unsteady computations.

6. Review the results.

7. Save the current "Dam Break 10s" with a new, descriptive name such as, "Dam & Levee Break."

8. Add the levee break information, save the plan data, and perform the unsteady computations.

9. Increase the length of the time window, save the plan data ("Long Run"), and perform the unsteady computations.

10. Add the levee break repair data, save the plan data ("Levee Repair"), and perform the unsteady computations.

11. Review the results.

Tasks

1. Create a new plan.

Open up the unsteady flow analysis editor. Under the File menu, choose "Save Plan As..." and save the plan with a new name. This plan will contain the dam break information. Use a descriptive name for the plan and for the short id.

2. Enter the dam break information.

The inline weir representing the dam is located at river station 81500. The dam break has the following characteristics:

Center Station	= 4200 ft
Final Bottom Width	= 400 ft
Final Bottom Elevation	= 592 ft
Left Side Slope	= 1
Right Side Slope	= 1
Full Formation Time	= 3 hrs
Failure Mode	= piping
Piping Coefficient	= 0.6
Initial Piping Elevation	= 620 ft
Trigger Failure at	use water surface
Starting Water Surface	= 664 ft

After the dam break information has been entered, save the plan.

3. Perform the unsteady flow simulation (with the original time step) and review the results.

Both the profile plot and the cross section plot can be animated to show the changing water surfaces with time. By choosing the inline weir cross section plot (river station

81500), and clicking on the animate button (the right arrow symbol), and then clicking on play, the changes in the water surface and the growth of the dam breach can be seen.

4. Create a new plan and then compute the dam break with a ten second time step.

Under the File menu, choose "Save Plan As..." and save the plan with a new name. This plan will contain the dam break information and it will be run with a ten second time step. Use a descriptive name for the plan and for the short id.

5. Perform the unsteady flow simulation and review the results.

Compare the results of the dam break with the ten second time step to the previous dam break results. This can be accomplished by clicking on the stage and flow hydrograph button (on the main HEC-RAS editor) and then choosing inline weir under type. Alternately, left click on the inline weir on the geometry editor (this may require zooming in) and choose plot stage and flow hydrograph. Under the options menu, choose plans and then select the two dam break plans (the current 10 second dam break plan and the previous dam break plan).

6. Save the plan with a new name and add the lateral weir levee breach.

At the downstream end of the reach, the levee is composed of 3 separate lateral weirs. Add a levee breach at the upstream lateral weir (river station 32100) with the following characteristics:

Center Station	= 700 ft
Final Bottom Width	= 500 ft
Final Bottom Elevation	= 558 ft
Left Side Slope	= 1
Right Side Slope	= 1
Full Formation Time	= 5 hrs
Failure Mode	= overtopping
Trigger Failure at	use water surface
Starting Water Surface	= 579 ft

7. Review the results.

8. Create a new plan with a longer time window. The simulation will now be run until 22Feb1999 at 1200. Increase the computation interval to 1 minute (this is mostly to save time running the workshop), and run the simulation.

9. Create a new plan ("yet again") and add a levee repair with the following characteristics:

Hours after full breach to start	12 hrs
Total repair time	12 hrs
Final filled in elevation	570 ft

10. Review the results. Compare the water surfaces in the two storage areas with and without the levee repair option.

11. More detailed output (optional)

If time allows, an additional run(s) can be made that has more detailed output. This will provide a much smoother animation. Save the plan (for the last time!) with a new name. On the unsteady flow analysis editor, in the **Detailed Output Interval:** window, change the time interval to a smaller increment—for instance 10 minutes. After performing the unsteady flow simulations, open up the profile plot and a cross section plot of the inline weir and run the animation.

Questions

1. How does the floodwave hydrograph from the ten-second time step compare to the original dam break hydrograph? Which do you think is more accurate? What additional time step checks could be made?

2. How much did the lateral weir breach change the maximum water surface and the peak flow into storage area STO #2? How much did the levee repair option reduce the water surface compared to the breach without the repair?

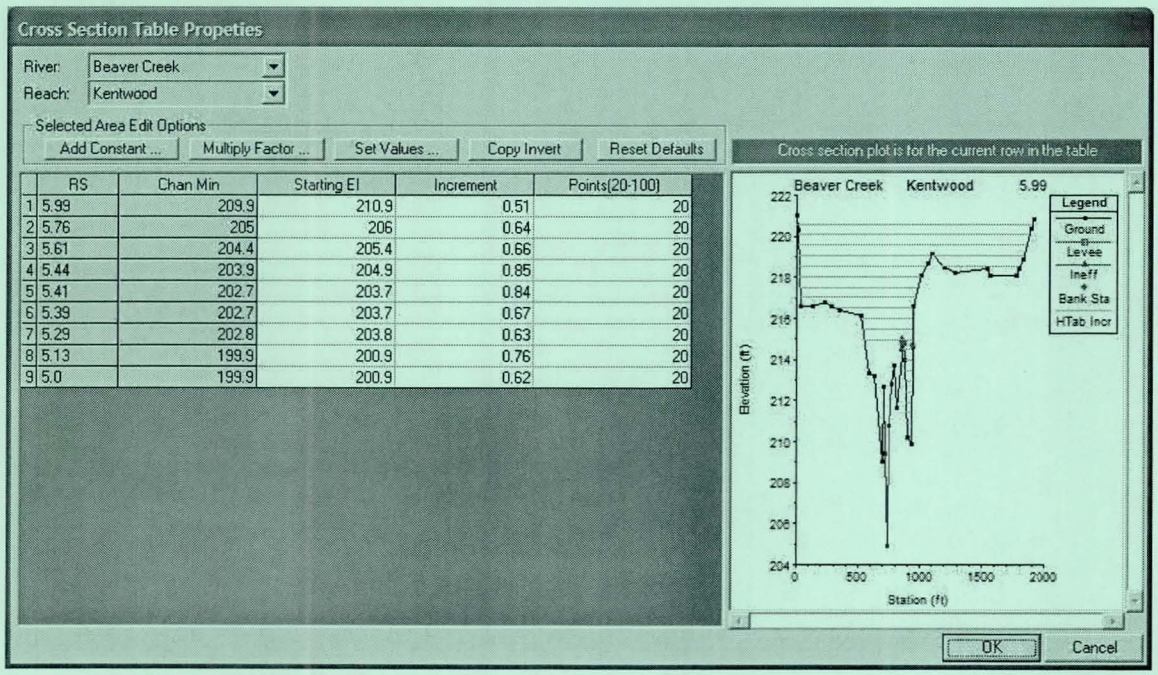
3. Does the data input and the model output both make "hydraulic sense?" What additional analysis and sensitivity checks could be done for this dam break problem?

Workshop Solution
Pre-Processing Geometric Data

Tasks

1. View HTAB Parameters and Analyze the Hydraulic Property Table

The **Cross Section Table Parameters** were viewed by depressing the HTAB Parameters button found in the **Geometric Data Editor**, located in the "Tools" section. Figure 1 displays the default parameters. The default parameters are sufficient for this example because the table extends to the top of each cross section with an incremental increase in elevation well below one foot.



	RS	Chan Min	Starting El	Increment	Points(20-100)
1	5.99	209.9	210.9	0.51	20
2	5.76	205	206	0.64	20
3	5.61	204.4	205.4	0.66	20
4	5.44	203.9	204.9	0.85	20
5	5.41	202.7	203.7	0.84	20
6	5.39	202.7	203.7	0.67	20
7	5.29	202.8	203.8	0.63	20
8	5.13	199.9	200.9	0.76	20
9	5.0	199.9	200.9	0.62	20

Figure 1 Default Cross Section Table Parameters

Next, (after running the **Geometry Preprocessor** on the Unsteady Flow Analysis window) Figure 2 was displayed by selecting **View** and then **Hydraulic Property Plots** from the main program window. As can be seen from the figure, there is a smooth increase in conveyance as elevation increases from the channel invert.

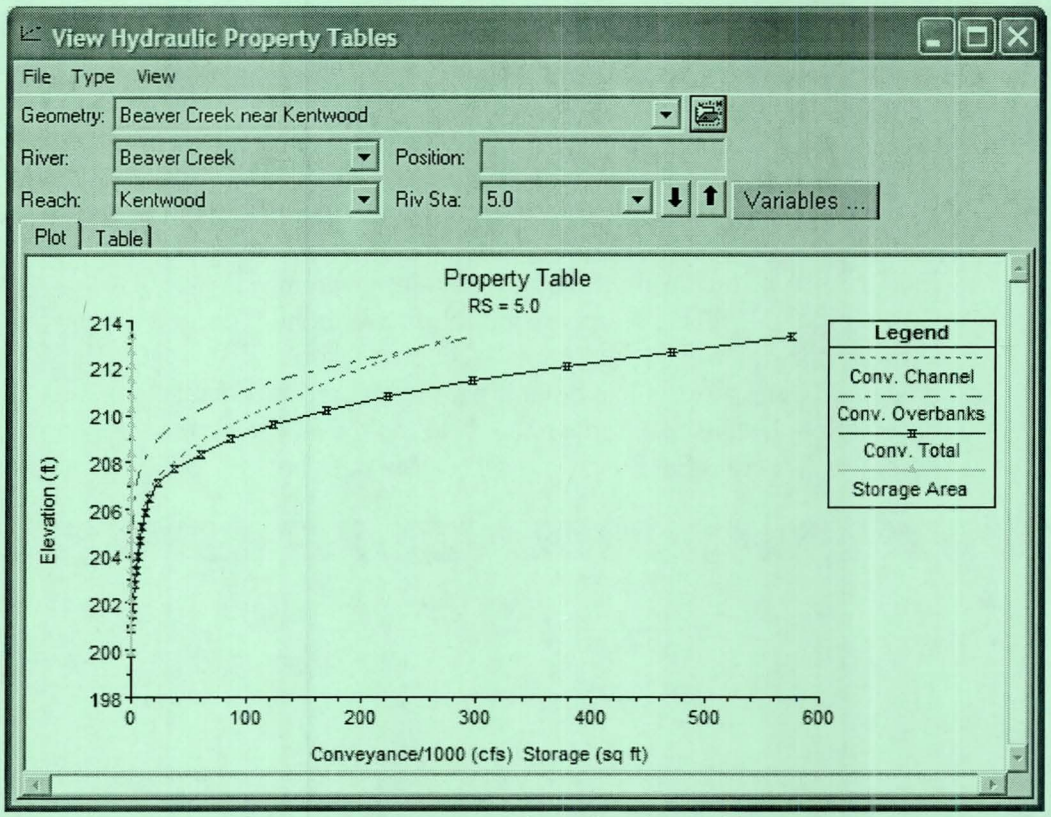


Figure 2 Hydraulic Property Plot at River Station 5.99

2. Place Levees at Necessary Locations

Levees were placed at the following locations using the **Graphical Cross Section Editor**.

River Station	Station	Elevation
5.99	866	214.8
5.76	906	214.3
5.29	223	209.6
5.13	358	209
5.00	518	209

The editor is selected from the **Geometric Data Editor** under the **Tools** menu. In the **Graphical Cross Section Editor, Add Levees** was selected from the **Options** menu.

Levees may also be placed using the **Cross Section Data Editor** found in the **Geometric Data Editor**, located in the "Editors" section. After depressing the **Cross Section** button the levees were added by selecting **Levees...** from the **Options** menu.

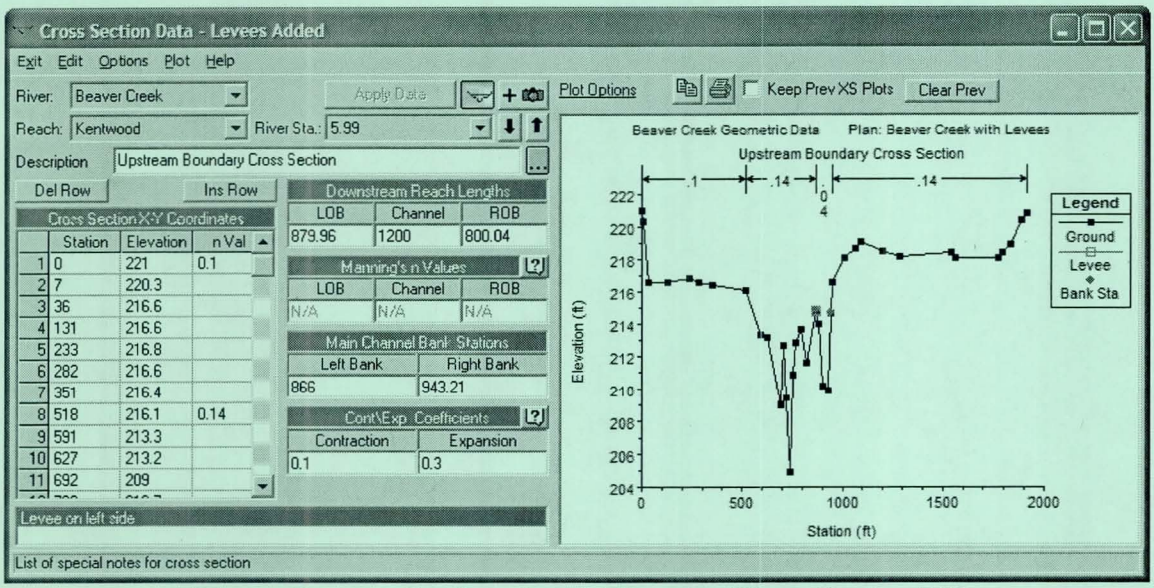


Figure 3 Cross Section Data Editor for River Station 5.99

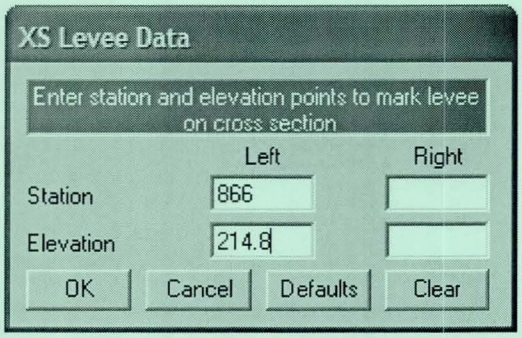


Figure 4 Adding Levees to River Station 5.99

After the levees were added the geometry was saved as a new file and the pre-processor was run. Next, the **Hydraulic Property Tables** were examined. Figure 5 displays the plot of River Station 5.99 with the addition of the levee. As seen in the figure, when the levee is overtopped there is an abrupt increase in conveyance with a small change in elevation. This occurs because RAS is a one-dimensional model, and as soon as the levee is overtopped the area protected by the levee is instantaneously filled with water. Viewing Figure 6, the area instantly filled is from Station 500 to 900, up to an elevation of 214.8 feet.

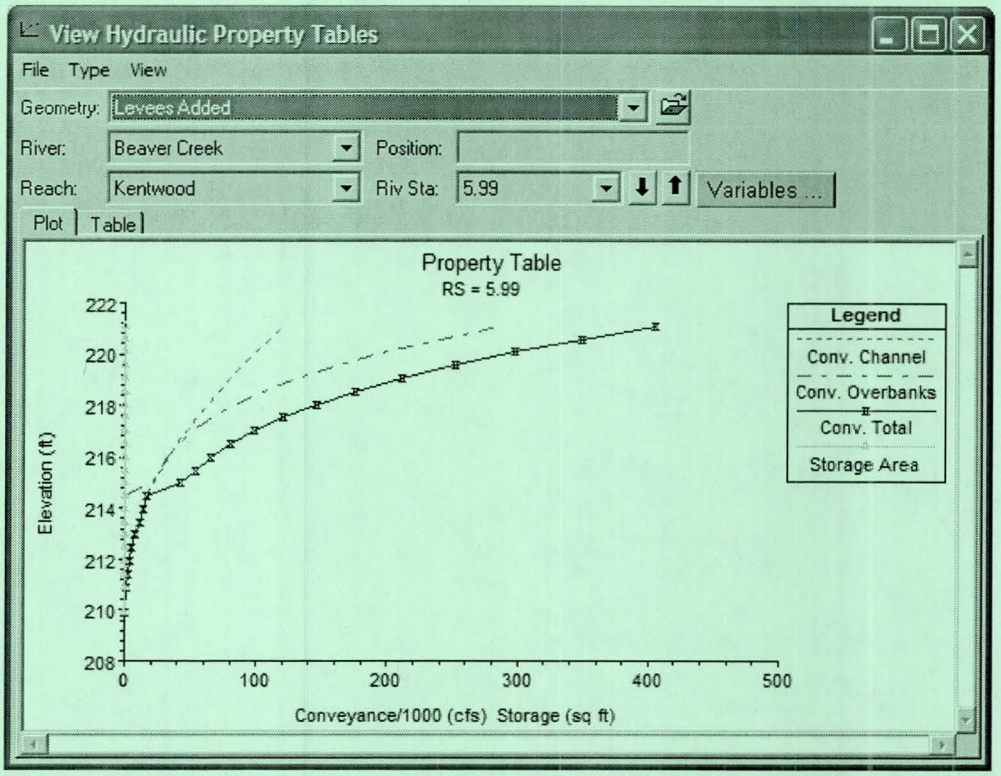


Figure 5 Hydraulic Property Plot of River Station 5.99 After Levee Addition

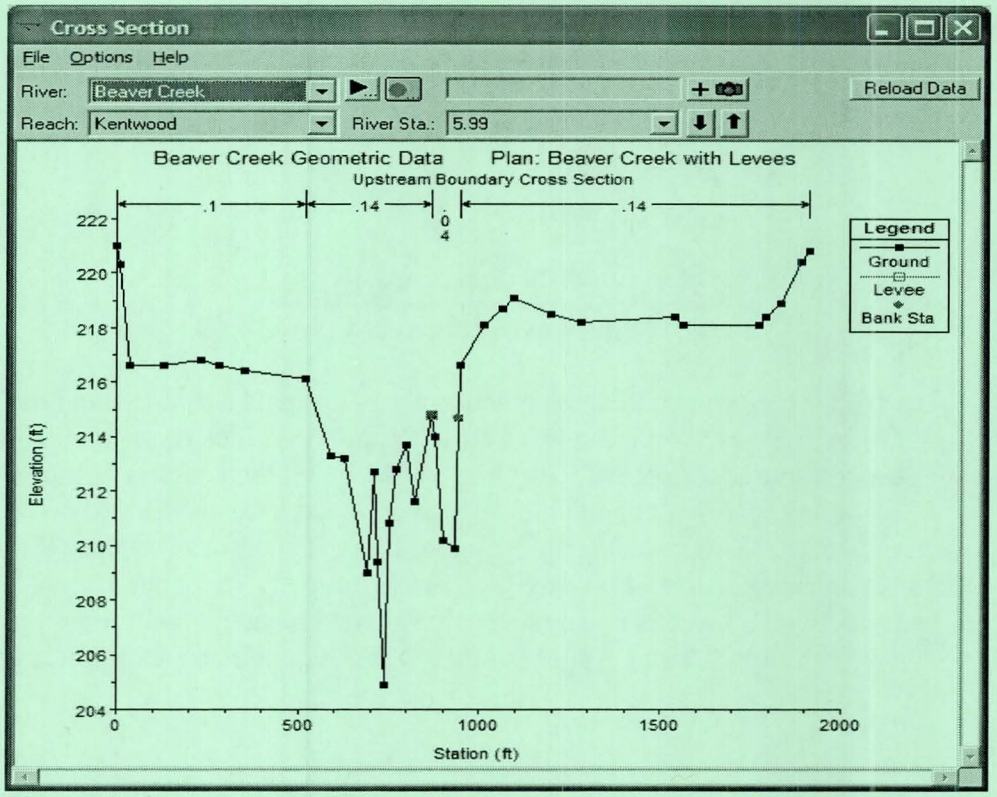


Figure 6 Cross Section 5.99

3. Place Normal Permanent Ineffective Flow Areas

Permanent ineffective flow areas were placed in generally the same manner as the levees. The **Graphical Cross Section Editor** can be used by selecting **Add Normal Ineffective Area (1 pt)** from the **Options** menu. The default setting for ineffective flow areas is non-permanent. After the ineffective flow areas are placed, they need to be changed from non-permanent to permanent. This was done by selecting **Set Ineffective Areas To Permanent Mode...** from the **Geometric Data Editor** under the **Tools** menu. The river stations with ineffective flow areas were selected, as shown in Figure 7.

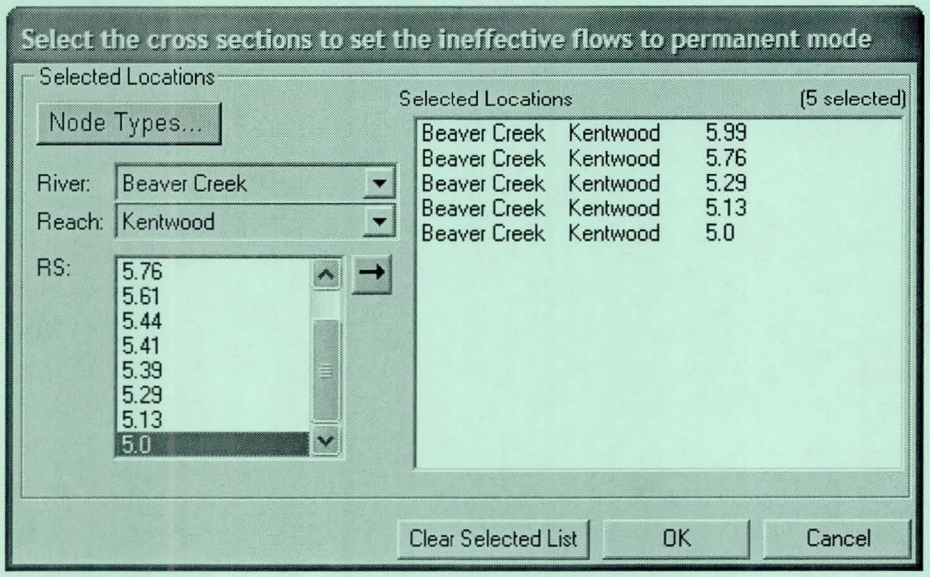


Figure 7 Ineffective Areas Changed to Permanent

Additionally, the ineffective flow areas may be added by selecting **Ineffective Flow Areas...** from the **Options** menu of the **Cross Section Data Editor**. This will display Figure 8. To set the ineffective flow areas to permanent, the box labeled "Permanent" was checked.

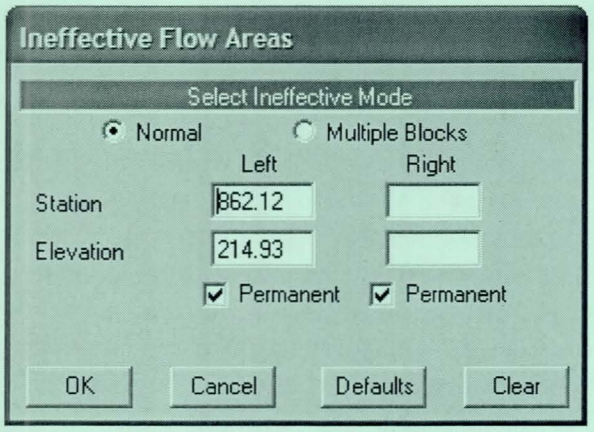


Figure 8 Permanent Ineffective Area

Next, the geometry was saved as a new file and the pre-processor was run. Finally, the **Hydraulic Property Tables** were viewed. Figure 9 displays the plot for River Station 5.99. As can be seen from the figure, the addition of the permanent ineffective area smoothed out the transition of increasing conveyance with increasing elevation. The permanent ineffective flow area behind the levee provides a better representation of what is physically occurring because as the levee is initially overtopped the conveyance in the overbanks will gradually increase.

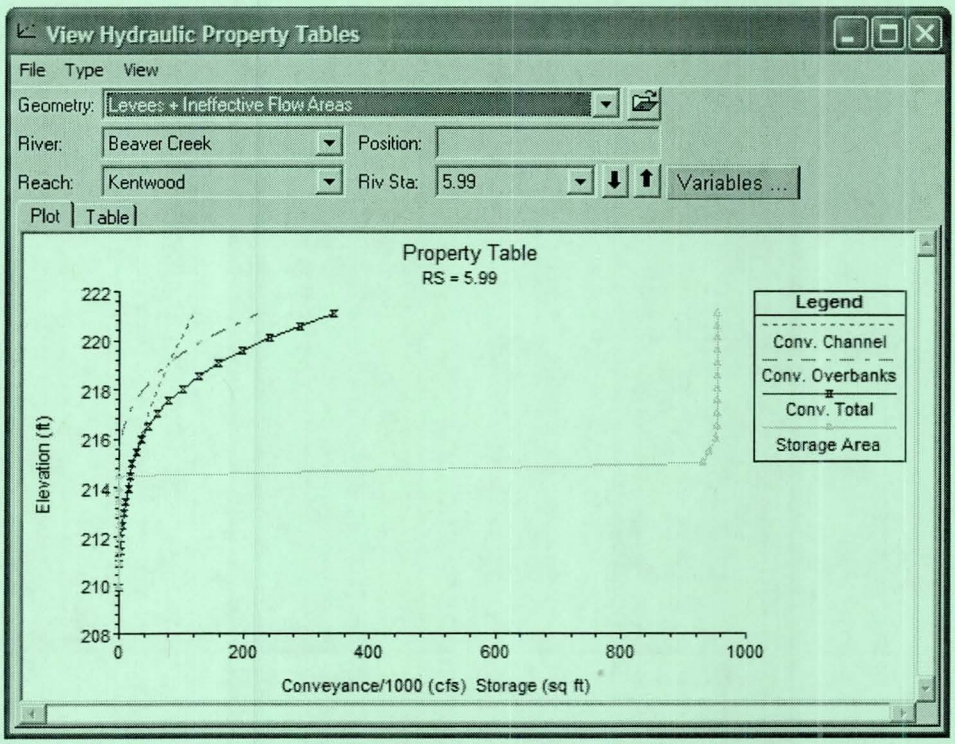


Figure 9 River Station 5.99 After Permanent Ineffective Area

Questions

1. Considering that HEC-RAS is a one-dimensional model, what problems will occur if a levee is overtopped during a simulation? What options in RAS can be used to model a levee being overtopped?

Because the program is one-dimensional, there can only be one water surface elevation. Thus, when a levee is overtopped the water surface level behind the levee will instantaneously be the same elevation as the water in the main channel. Of course, this is not actually occurring and must be taken into account when modeling. To model this phenomenon correctly a permanent ineffective flow area can be placed behind the levee. The sudden increase in water behind the levee will be seen by the program as storage, and not conveyance.

Another application would be to place high Manning's n values behind the levee. The high Manning's n value will hinder conveyance in the same manner as a permanent ineffective flow area.

2. Are there any anomalies with the conveyance in the overbanks at River Station 5.29? If so, what is it caused by, and how can it be fixed?

As seen on Figure 10, there is a break in the smooth relationship between conveyance and elevation. By examining the table it is found that this break occurs between the elevations of 213.9 and 214.5. Next, the cross section geometry is examined at this elevation. As seen on Figure 11, the right overbank geometry dramatically changes at this point.

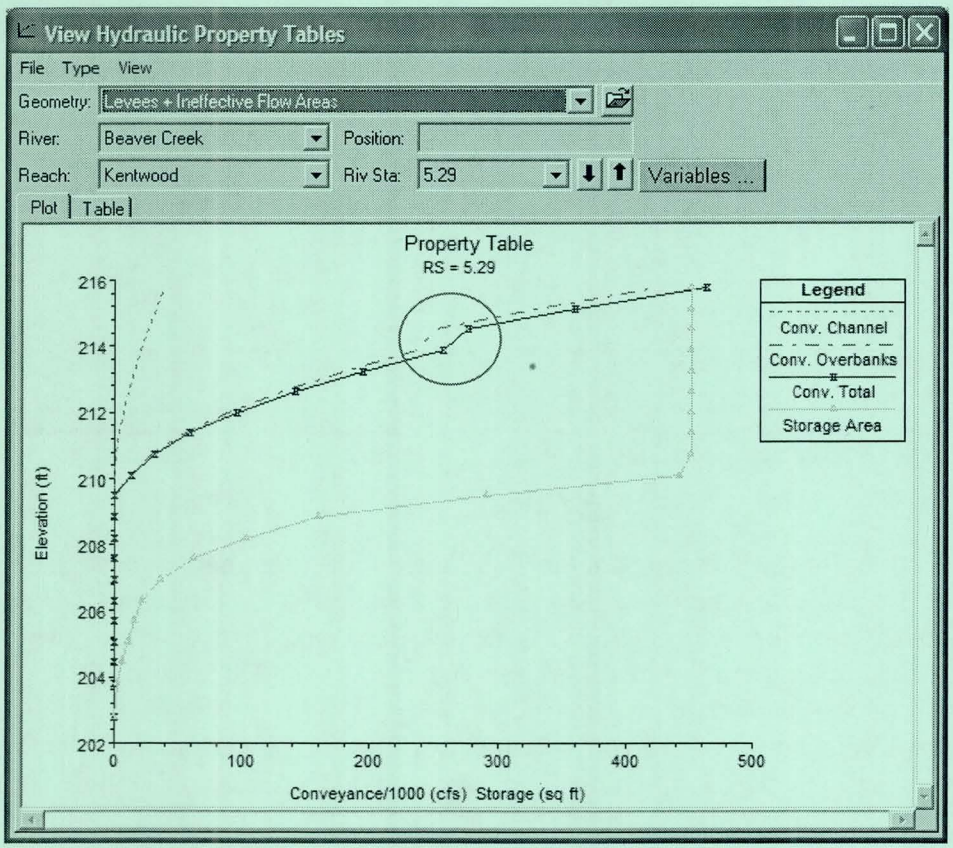


Figure 10 Hydraulic Property Plot at River Station 5.29

In HEC-RAS, the determination of total conveyance for a cross section requires that flow be subdivided into units for which the velocity is uniformly distributed. The approach used in RAS is to subdivide flow in the overbank areas using the locations where n-values change. The program sums up all the incremental conveyances in the overbanks to obtain a conveyance for the left overbank and the right overbank.

HEC-RAS uses Manning's equation to calculate conveyance. Imbedded in this equation are relationships for channel depth, channel area, and wetted perimeter. In this example, when the calculations go just above the elevation of 214 feet, Manning's equation sees an enormous increase in wetted perimeter for a minute increase in water depth. The conveyance answer found from this calculation is dramatically different from the conveyance answer found at the previous calculation below 214 feet.

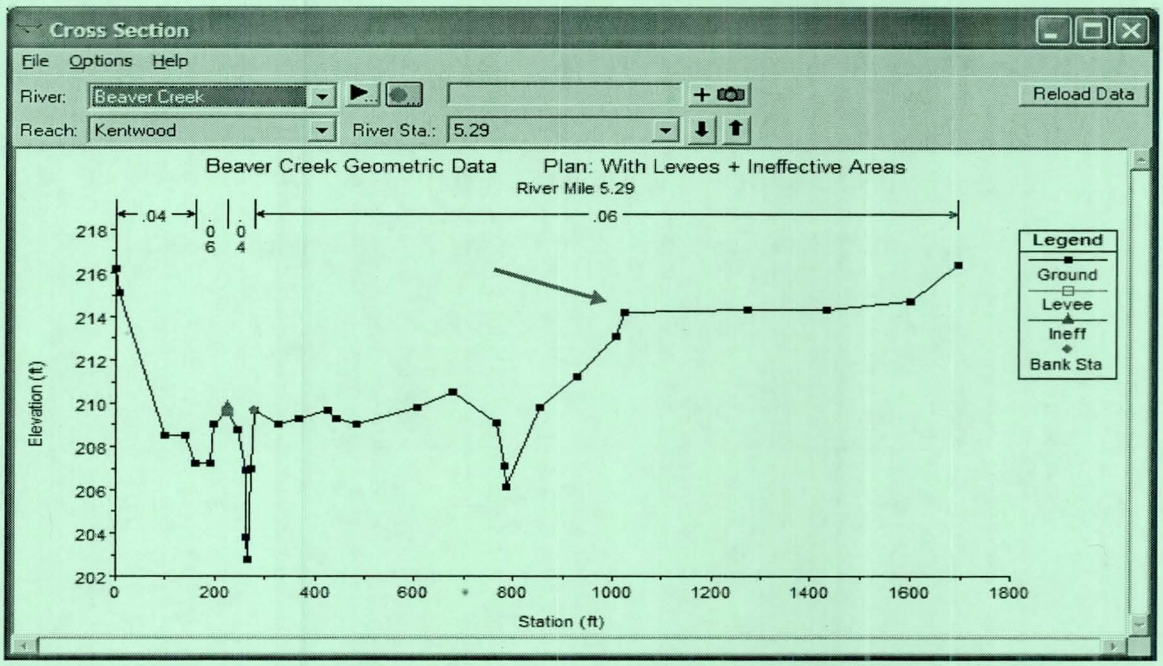


Figure 11 River Station 5.29

To account for this situation another subdivision for Manning's n must be added at the point of discontinuity. The value does not need to be different, as long as the subdivision is present, RAS will calculate separate conveyances for the geometry. As seen in Figure 12, a subdivision was added at Station 1024.

Figure 13 displays the **Hydraulic Properties Plot** for River Station 5.29 after the subdivision was added at Station 1024. As seen on the figure, this subdivision for the conveyance calculations smoothed the relationship between conveyance and elevation.

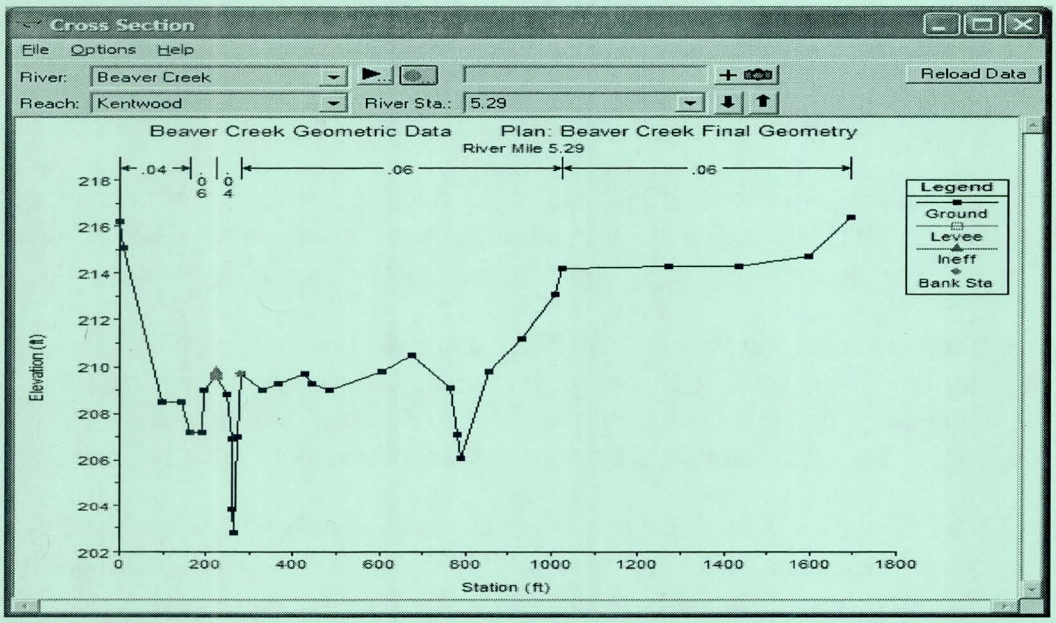


Figure 12 River Station 5.29 With Additional Manning's n Subdivision

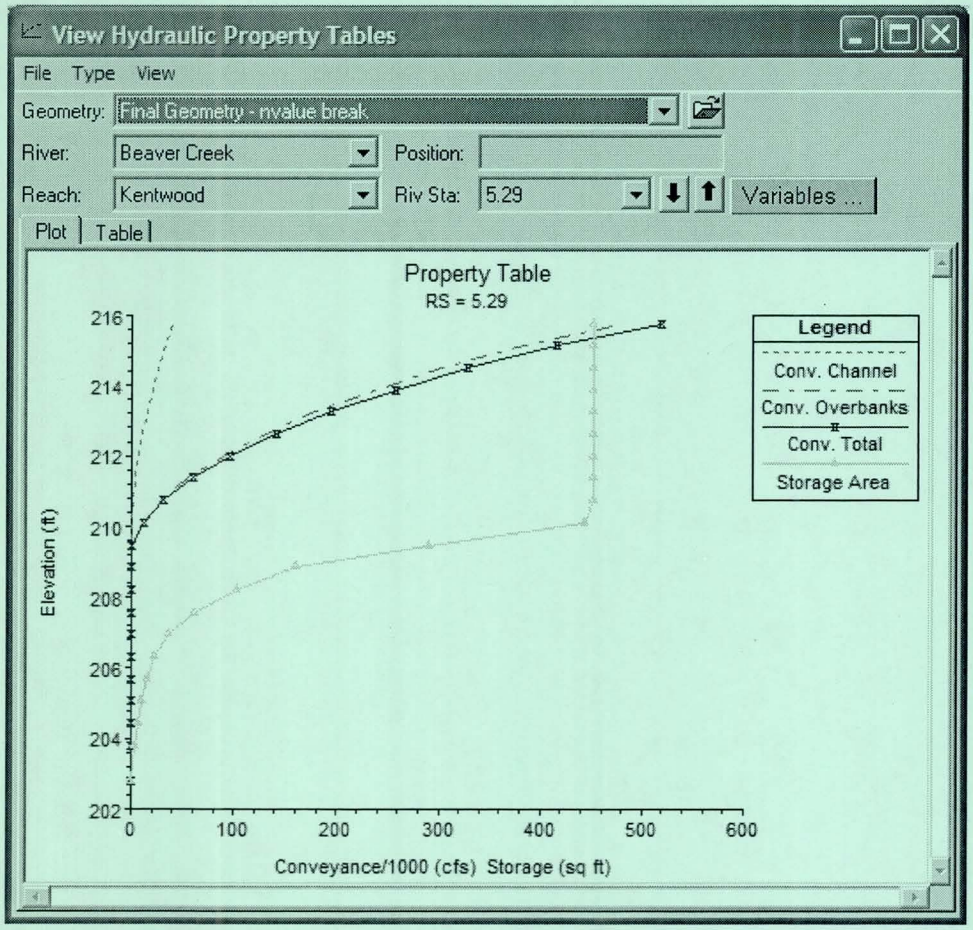


Figure 13 Hydraulic Property Plot of River Station 5.29 After Adding Subdivision

3. How does a permanent ineffective area affect storage within the cross section? When should a cross section be modeled with non-permanent ineffective areas and when should it be modeled with permanent ineffective areas?

As illustrated in Figure 9, a permanent ineffective storage area behind a levee will quickly add storage to the cross section once the levee is overtopped. If there was no levee, the area of storage would gradually increase with elevation.

If it is assumed that the entire ineffective area will begin to convey flow once a particular water surface elevation is reached, then the ineffective area should be modeled as non-permanent. If it is assumed from the geometry that a particular area will never convey flow then the area should be modeled as permanent.

Note: If the cross sectional geometry denotes a significant change in storage or conveyance when a levee is overtopped then this cross section should be altered. The cross section should be terminated at the levee and the area that was deleted should be modeled as a storage area. Abrupt and significant changes in storage and conveyance should be avoided because they can cause instabilities in the unsteady flow model.

Workshop 2 Solution:

Basic Unsteady Flow Modeling Using HEC-RAS

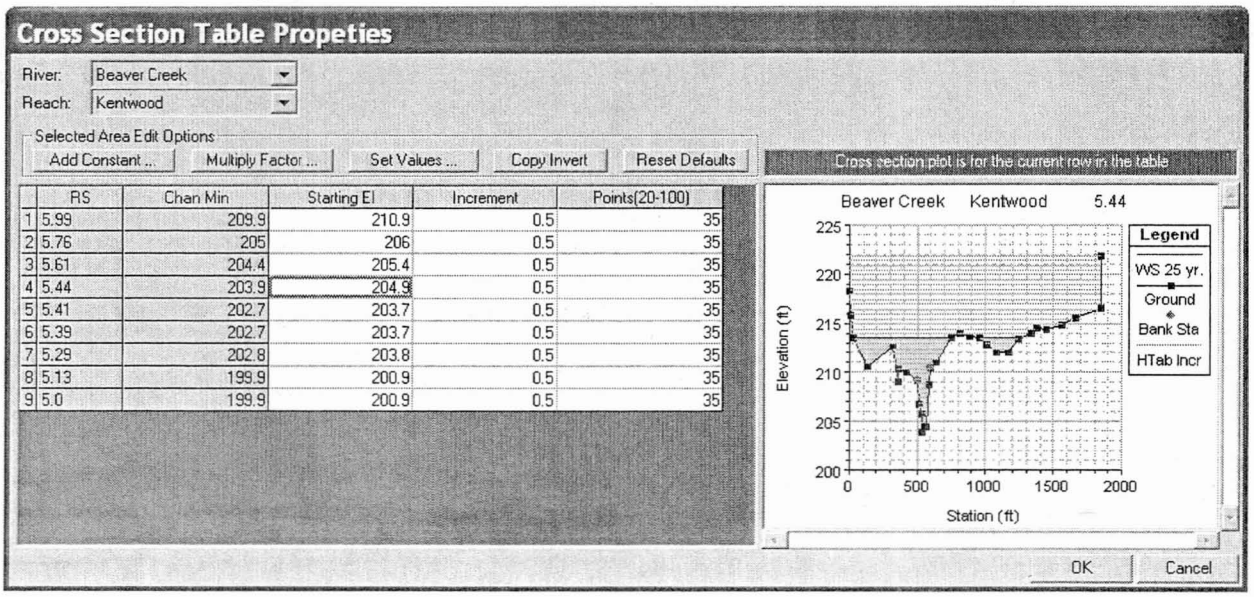
In this problem, the geometric data processed in Workshop 1 will be used to route a flood event. Unsteady flow data will be defined, a simulation performed and output will be reviewed.

Workshop Tasks:

1. Open Wrk2.prj. This data represents the data developed in Workshop 1

2. Open the Geometry file. Check the HTAB settings and adjust them, if necessary based on your Workshop 1 analyses. Save the file.

The HTAB settings, shown below are those established in workshop 1.

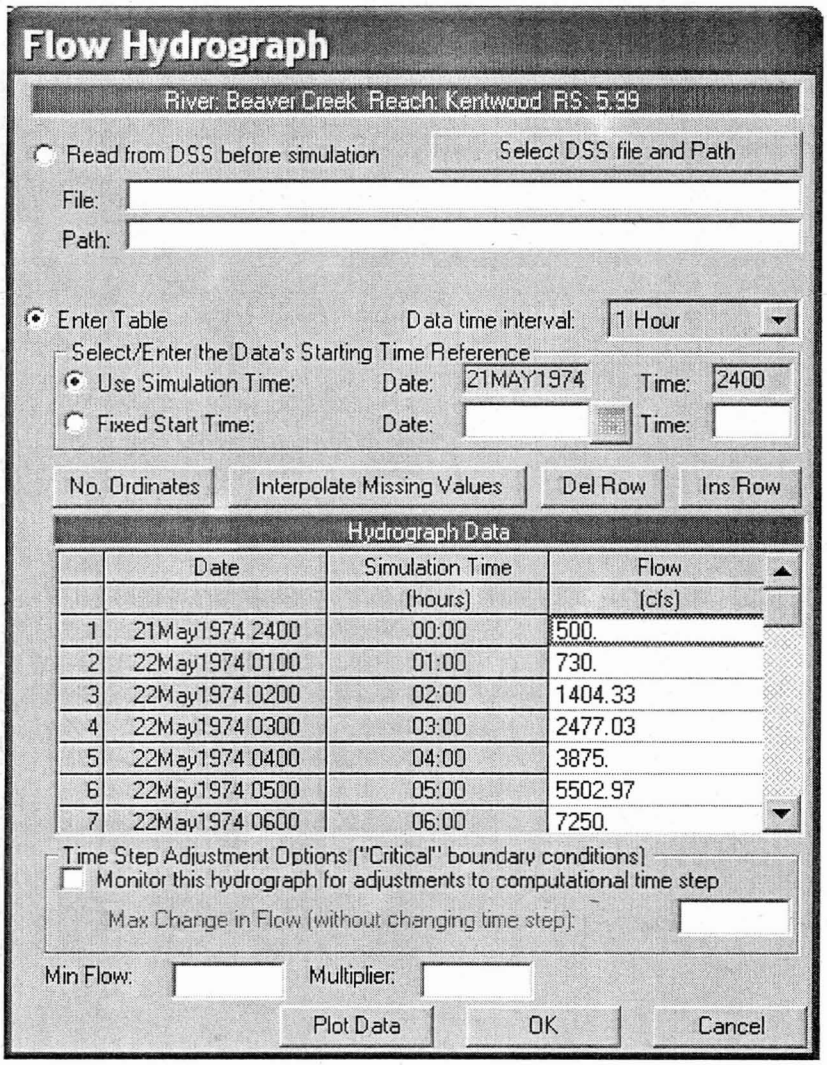


	RS	Chan Min	Starting El	Increment	Points(20-100)
1	5.99	209.9	210.9	0.5	35
2	5.76	205	206	0.5	35
3	5.61	204.4	205.4	0.5	35
4	5.44	203.9	204.9	0.5	35
5	5.41	202.7	203.7	0.5	35
6	5.39	202.7	203.7	0.5	35
7	5.29	202.8	203.8	0.5	35
8	5.13	199.9	200.9	0.5	35
9	5.0	199.9	200.9	0.5	35

3. Open the Unsteady Flow data editor and enter the required data for processing.

a. Upstream Boundary is a flow hydrograph. The data for the flood is in an excel file, and is listed in Hydrograph Table, on the next page.

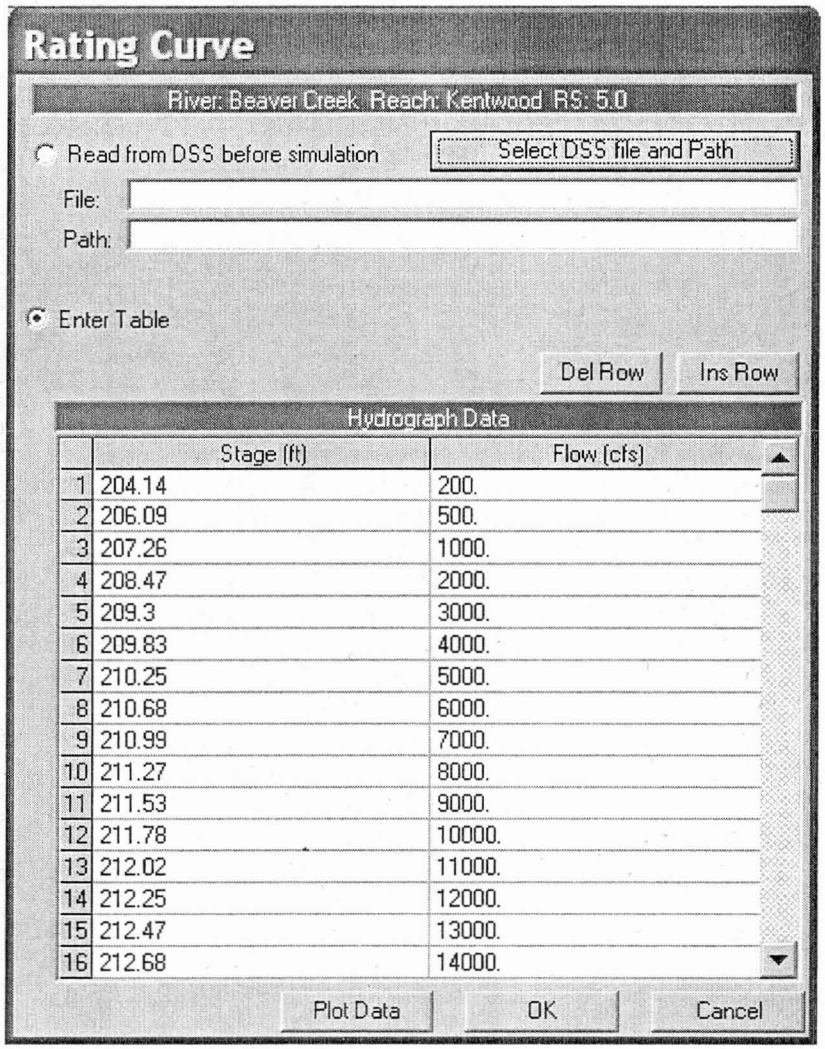
The Excel file was opened; the column of flow data was highlighted and copied. The Flow Column in the data editor was highlighted and the copied data pasted into the table (Ctrl V).



b. Downstream Boundary is a rating curve. The data are listed in the Rating Table, on the next page.

The downstream boundary Rating Curve was entered into the Table. An alternative would be to enter the data into the DSS file and then read it from DSS, using the option at the top of the table.

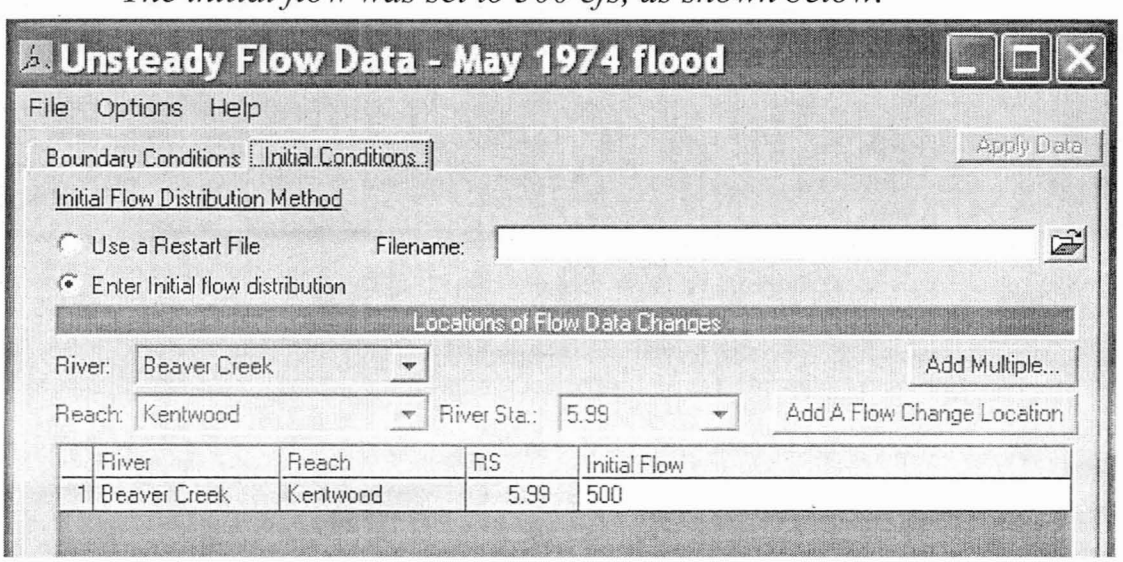
A zero flow stage was added at the minimum elevation to ensure stage is provided for low flows.



	Stage (ft)	Flow (cfs)
1	204.14	200.
2	206.09	500.
3	207.26	1000.
4	208.47	2000.
5	209.3	3000.
6	209.83	4000.
7	210.25	5000.
8	210.68	6000.
9	210.99	7000.
10	211.27	8000.
11	211.53	9000.
12	211.78	10000.
13	212.02	11000.
14	212.25	12000.
15	212.47	13000.
16	212.68	14000.

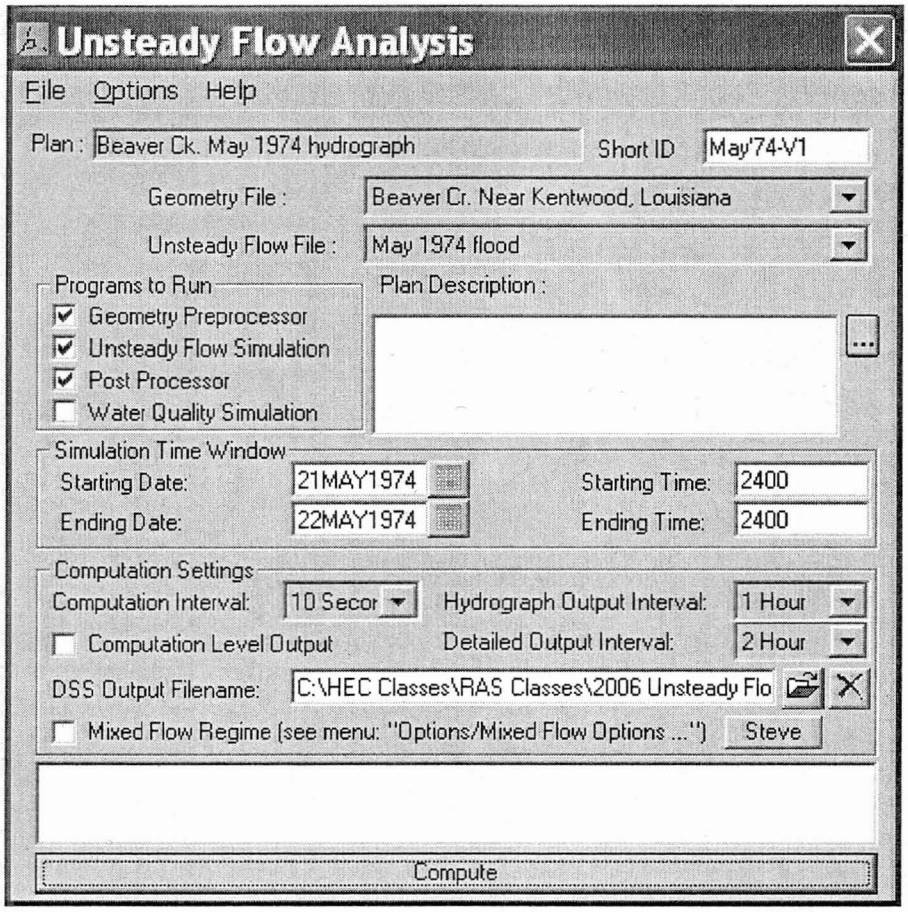
b. Initial Flow should start with the first discharge value from the flood hydrograph.

The initial flow was set to 500 cfs, as shown below.



	River	Reach	RS	Initial Flow
1	Beaver Creek	Kentwood	5.99	500

4. Set up the Unsteady Flow Analysis. Establish a new Plan, check Programs to Run, and set the times. What Computational Interval would be appropriate? Do not forget to select the cross sections for output.



The computation interval was initially set to 5 minutes. However, the simulation reported computational problems. The interval had to be reduced to 10 seconds before there was a fairly clean simulation. Still, maximum iterations were indicated at the beginning and end of the simulation. That is a very small time-step for this flow hydrograph and absence of hydraulic structures.

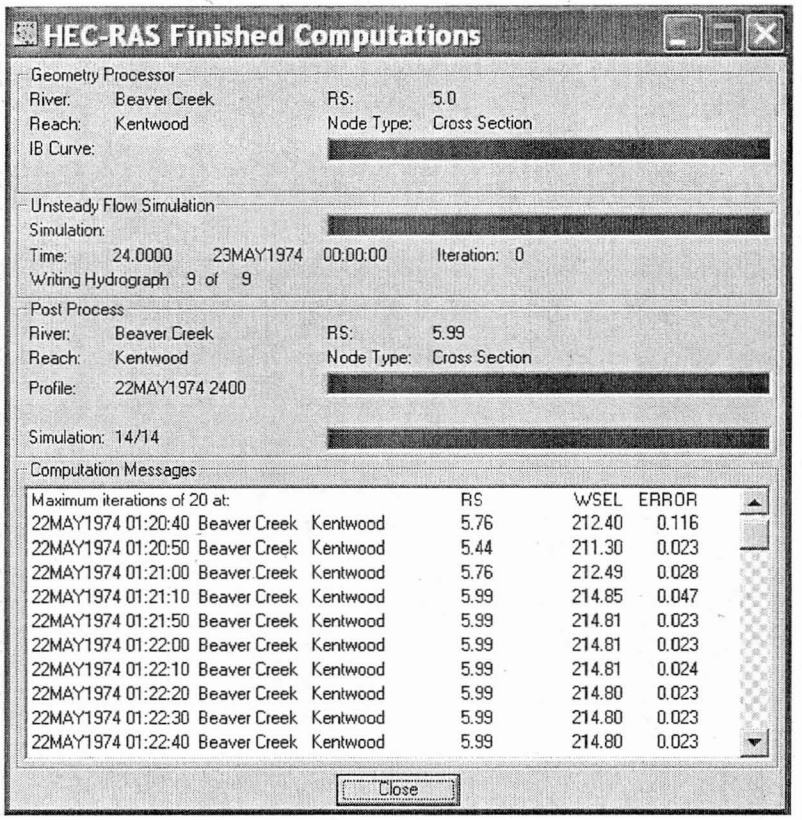
4. Evaluate the results.

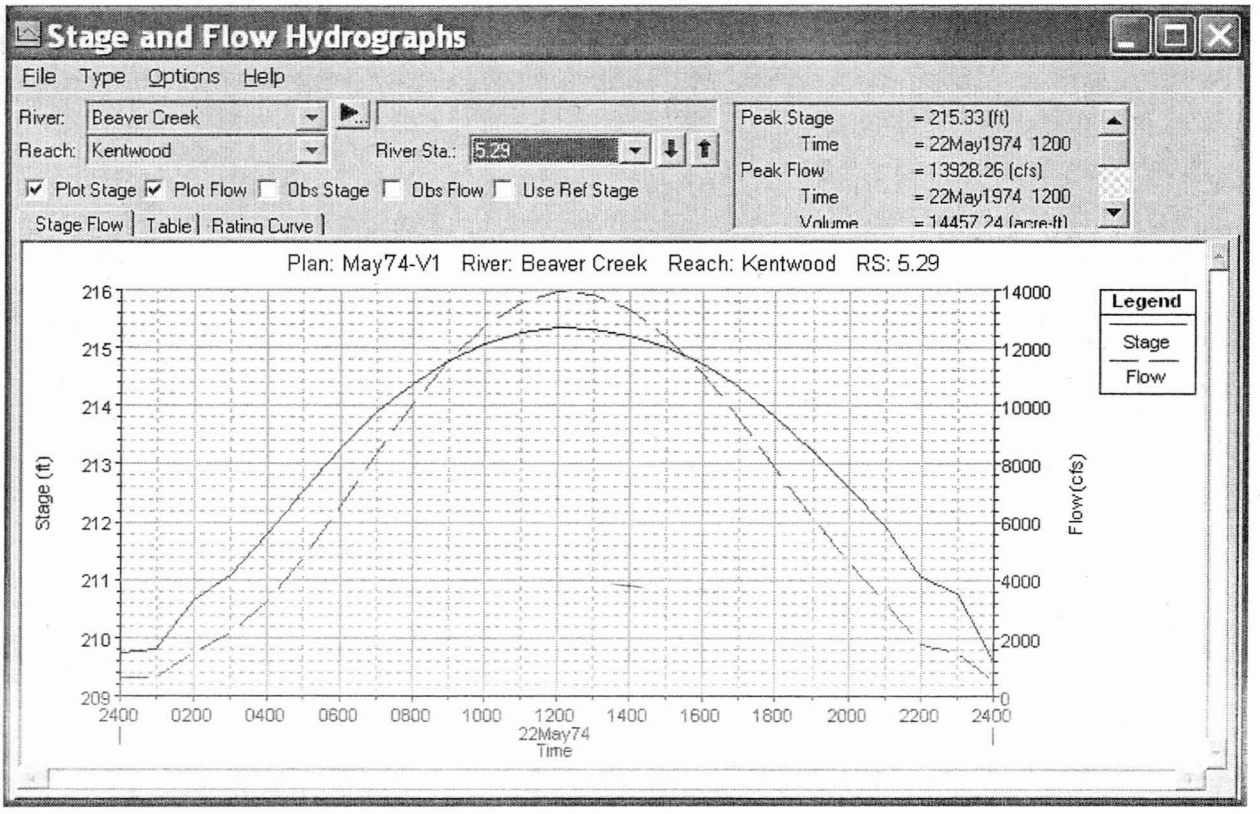
   a. Are there model results that suggest problems? What could be done to improve the solution?

The small computational interval, described above, is an obvious problem. Generally, shorter time and distance steps should improve the solution. Applying the cross-section interpolation routine can reduce the distance steps.

During the computations the Messages show Max iterations at time & location, along with ERROR. This indicates a failure to converge and the maximum error for that solution.

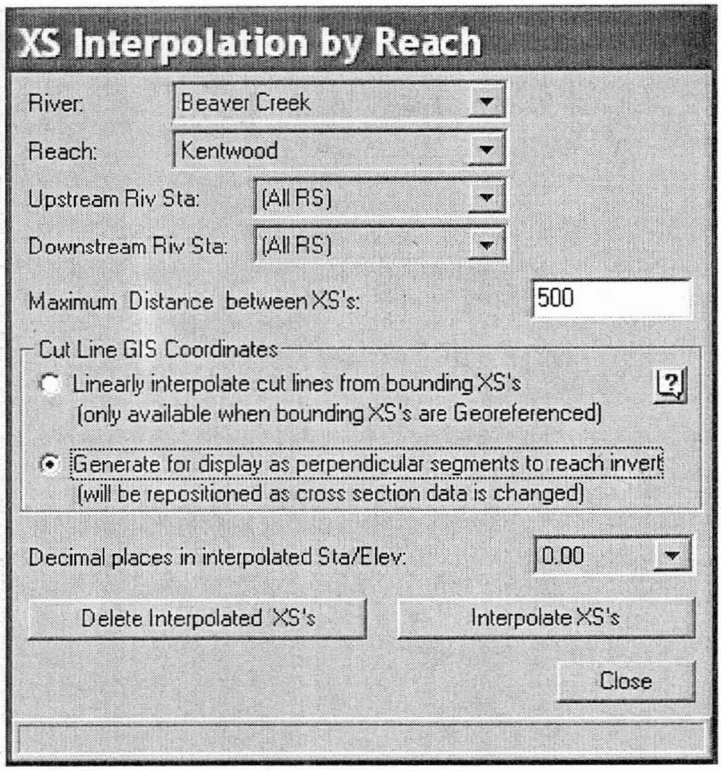
A review of the flow and stage hydrographs shows some irregularities at the lower end of the reach. Shown is the plot for section 5.29.





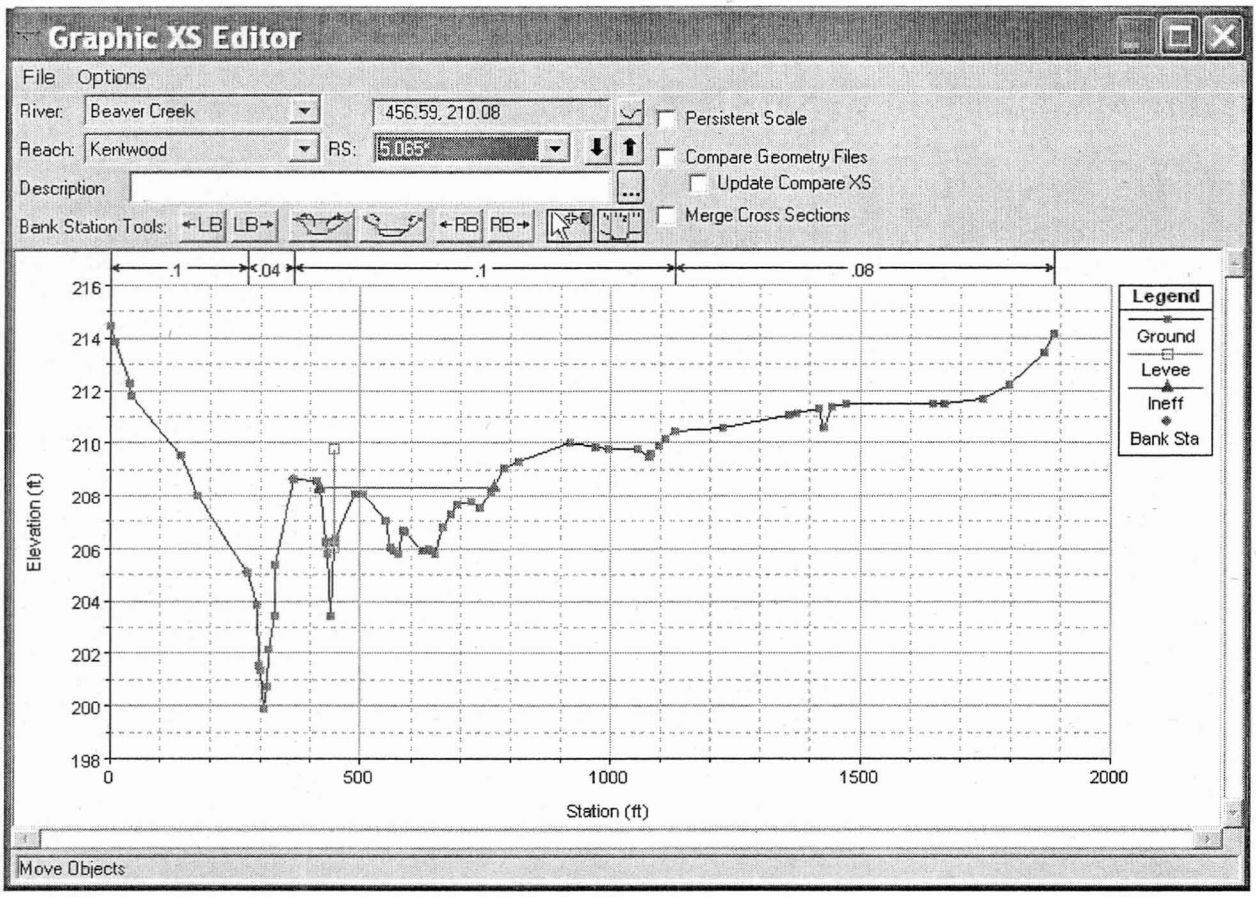
A review of the sections and the hydraulic properties shows the presence of levees and a sharp break in the conveyance when the levee is overtopped.

b. If time permits, make adjustments and run again.



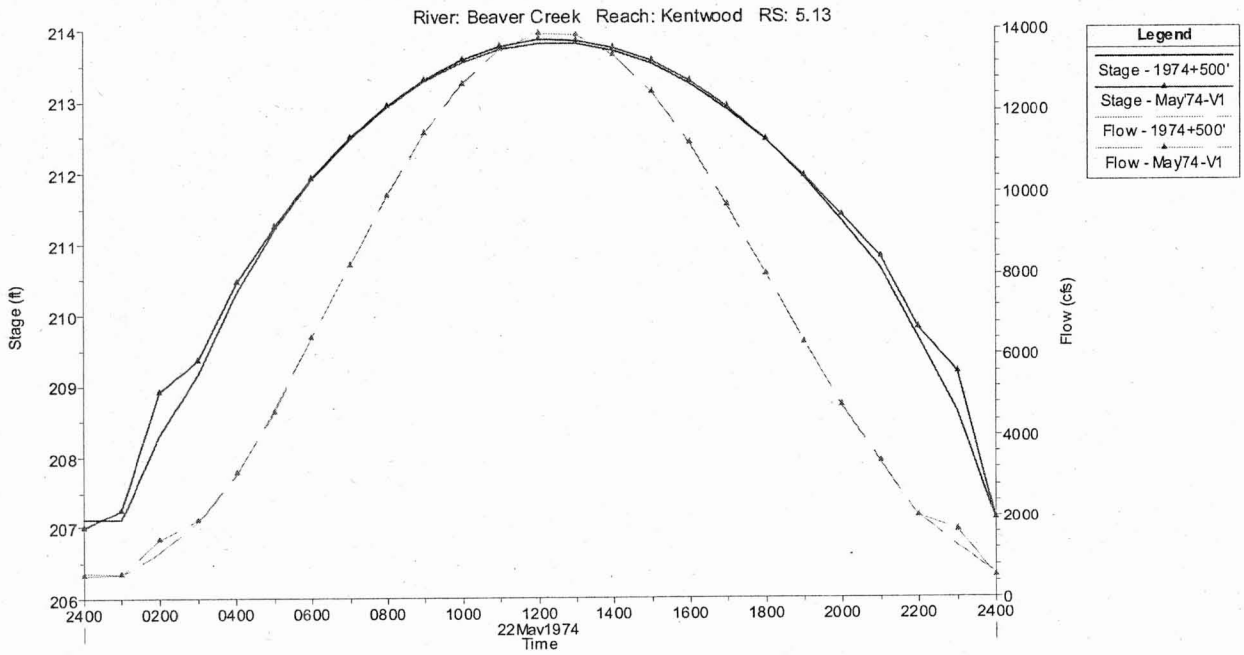
The Geometry file was modified to add interpolated sections with a maximum step of 500 feet. That means that any channel reach length greater than 500 feet will be subdivided by adding sufficient interpolated sections to reduce the step to no more than 500 feet. The geometry file was saved as a new file and a new plan was run with the modified geometry.
The interpolated sections should be carefully reviewed.

The levee option in the downstream sections did not appear reasonable on the interpolated sections. Using the Graphical Editor can easily modify this.



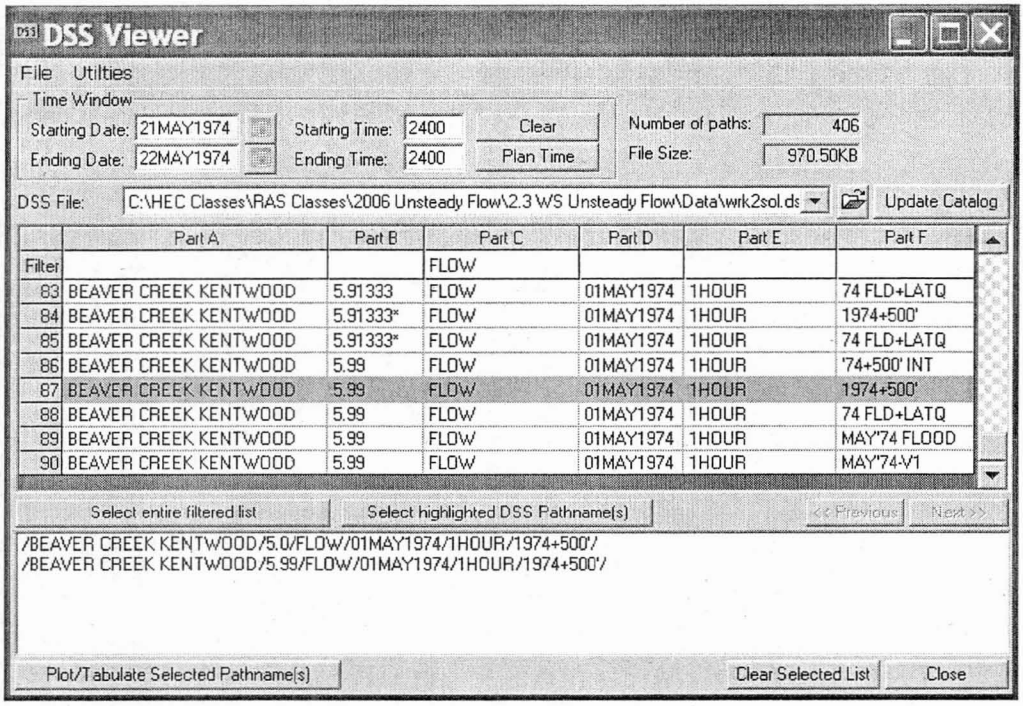
The levee location was moved for the interpolated sections and the geometry file was Saved As a new file name. The Simulation Editor was set to compute on a 30-second interval and the Output Locations was set for all Cross Sections. The simulation was run as a New Plan. The simulation sill showed some maximum iteration near the end of the simulation.

The results were compared to the original simulation. The comparisons for the two plans are shown below, upper is stage and lower is flow hydrograph. The lines without symbols are the new solution with interpolated cross sections. Note the smoother shape at the beginning and end of the hydrograph.



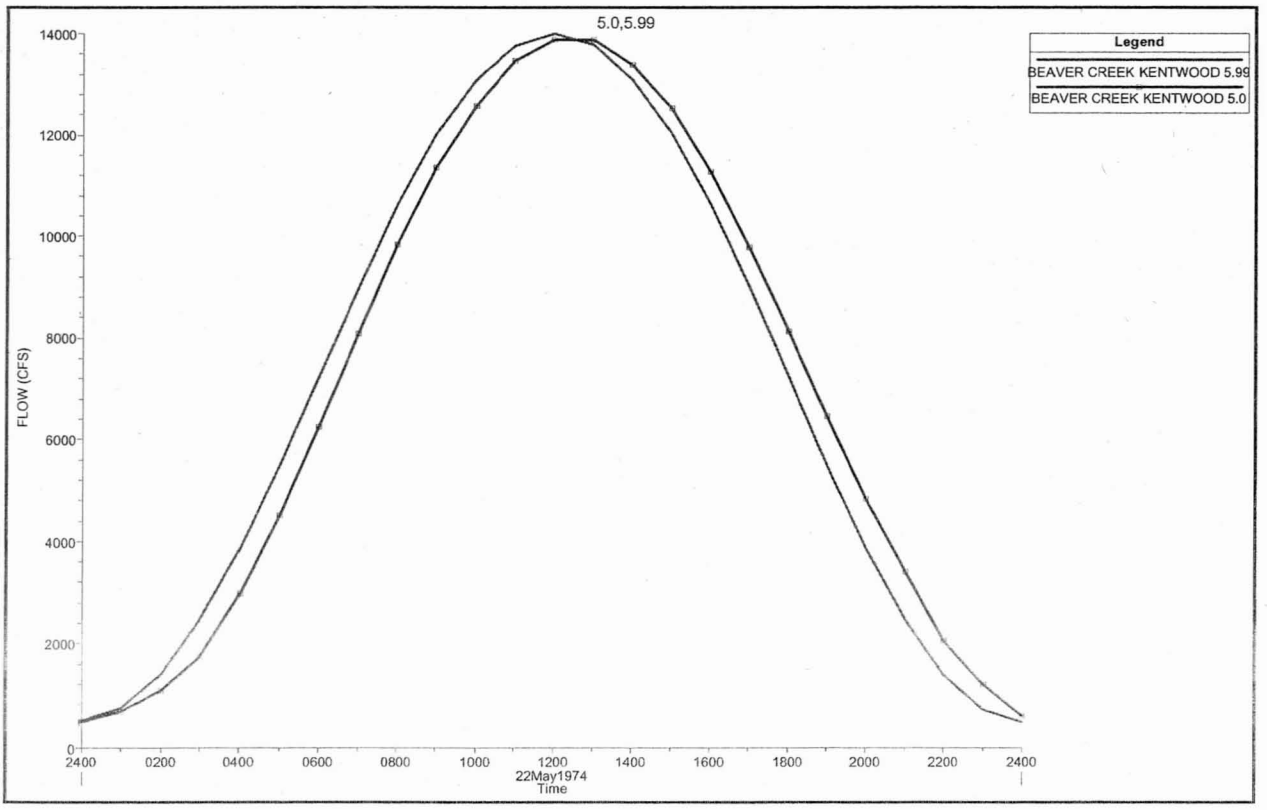
c. What was the attenuation in peak discharge?

The Flow and Stage hydrographs are available at any output location. By looking at the downstream section, one can see the peak discharge. However, to see the hydrographs at two, or more locations, you need to use the DSS Viewer. The flow hydrographs are shown for the upstream and downstream boundary cross sections.



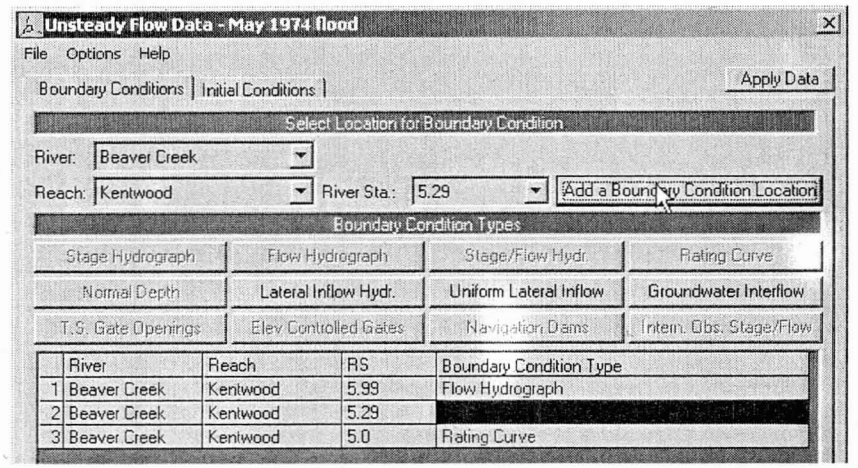
	Part A	Part B	Part C	Part D	Part E	Part F
Filter			FLOW			
83	BEAVER CREEK KENTWOOD	5.91333	FLOW	01MAY1974	1HOUR	74 FLD+LATQ
84	BEAVER CREEK KENTWOOD	5.91333*	FLOW	01MAY1974	1HOUR	1974+500'
85	BEAVER CREEK KENTWOOD	5.91333*	FLOW	01MAY1974	1HOUR	74 FLD+LATQ
86	BEAVER CREEK KENTWOOD	5.99	FLOW	01MAY1974	1HOUR	'74+500' INT
87	BEAVER CREEK KENTWOOD	5.99	FLOW	01MAY1974	1HOUR	1974+500'
88	BEAVER CREEK KENTWOOD	5.99	FLOW	01MAY1974	1HOUR	74 FLD+LATQ
89	BEAVER CREEK KENTWOOD	5.99	FLOW	01MAY1974	1HOUR	MAY'74 FLOOD
90	BEAVER CREEK KENTWOOD	5.99	FLOW	01MAY1974	1HOUR	MAY'74-V1

The Flow Hydrographs for the upstream and downstream locations are selected and then plotted. From the plot, you can see little change in peak flow. The table version of the display gives the actual values. The values around the peak were copied to the clipboard and are displayed on the next page.

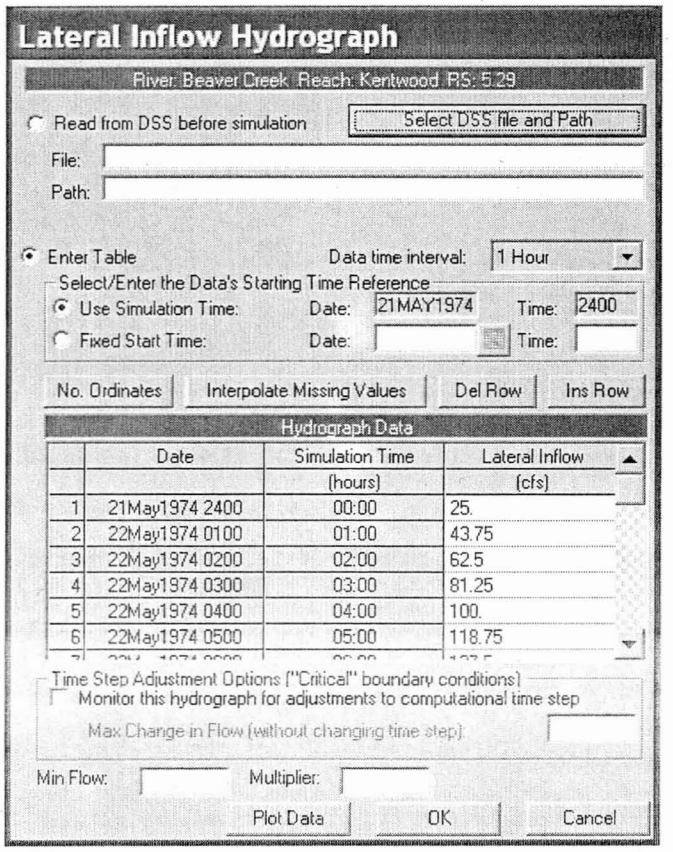


Data Date	KENTWOOD 5.0 INST-VAL CFS	KENTWOOD 5.99 INST-VAL CFS
22May1974 11:00	13464.15	13770.00
22May1974 12:00	13892.84	14000.00
22May1974 13:00	13865.86	13770.00

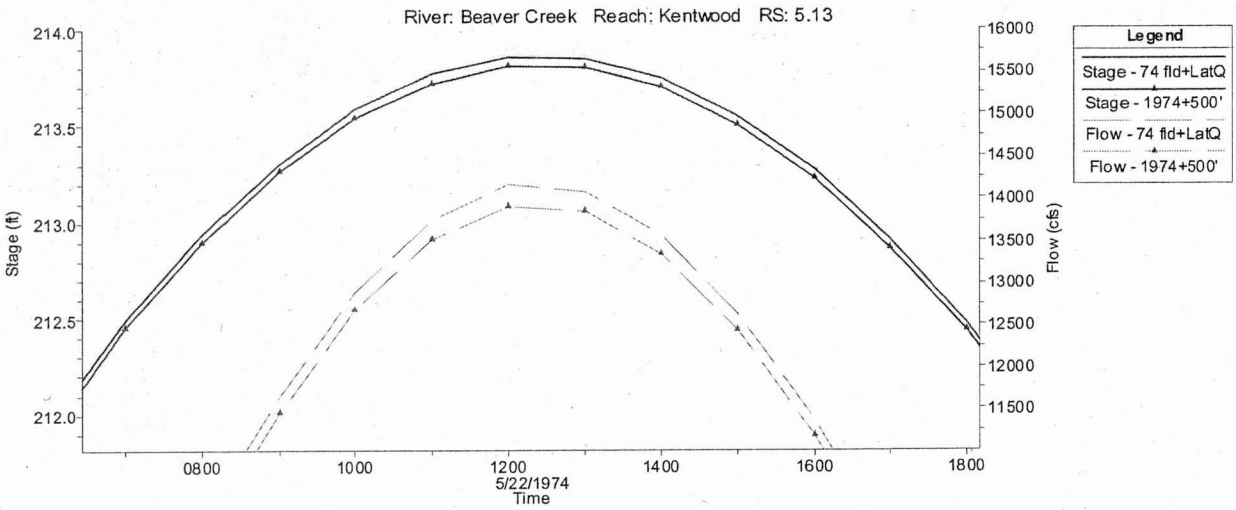
5. Add a lateral flow to River Station 5.29. The inflow starts at 25 cfs, peaks at 250 cfs at noon and subsides to 25 cfs at midnight. Save the Unsteady Flow file as a new name and run the simulation as a New Plan. How much and how far did the inflow change the peak profile?



The internal boundary location RS 5.29 was added. The Lateral Inflow Hydrograph option was selected. The starting, peak, and ending discharge were input and the "Interpolate Missing Values" option was used to fill in the lateral flow hydrograph. The flow file was saved under a new name and the unsteady flow simulation run under a new plan.



The flow is added below RS 5.29, so the largest change will be seen at the cross section immediately downstream, as shown in the hydrograph plot on the next page. (Stage is the upper plot and flow, the lower.) Although, there is some increase in stage both upstream and downstream from the added flows, as shown in the Profile Summary Table.



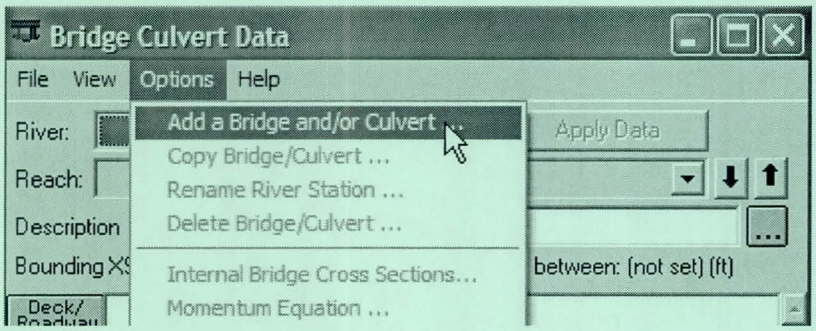
River: Beaver Creek Reach: Kentwood	River Sta	Profile	Plan	Q Total (cfs)	W.S. Elev (ft)
Kentwood	5.99	Max WS	74 fld+LatQ	13998.08	220.02
Kentwood	5.99	Max WS	1974+500'	13996.17	220.02
Kentwood	5.76	Max WS	74 fld+LatQ	13979.83	217.60
Kentwood	5.76	Max WS	1974+500'	13979.92	217.59
Kentwood	5.61	Max WS	74 fld+LatQ	13970.57	216.87
Kentwood	5.61	Max WS	1974+500'	13970.59	216.85
Kentwood	5.44	Max WS	74 fld+LatQ	13966.35	216.27
Kentwood	5.44	Max WS	1974+500'	13964.36	216.25
Kentwood	5.41	Max WS	74 fld+LatQ	13961.93	215.72
Kentwood	5.41	Max WS	1974+500'	13960.62	215.69
Kentwood	5.39	Max WS	74 fld+LatQ	13961.86	215.66
Kentwood	5.39	Max WS	1974+500'	13959.83	215.62
Kentwood	5.29	Max WS	74 fld+LatQ	13960.82	215.25
Kentwood	5.29	Max WS	1974+500'	13957.66	215.20
Kentwood	5.13	Max WS	74 fld+LatQ	14202.10	213.86
Kentwood	5.13	Max WS	1974+500'	13954.69	213.82
Kentwood	5.0	Max WS	74 fld+LatQ	14201.54	212.71
Kentwood	5.0	Max WS	1974+500'	13953.62	212.67

Workshop Solution
Modeling Bridges with Unsteady Flow

Tasks

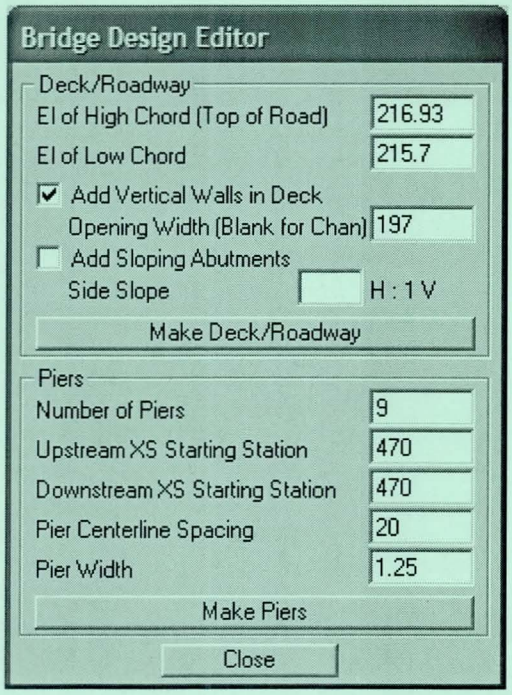
1. Enter the Bridge Information

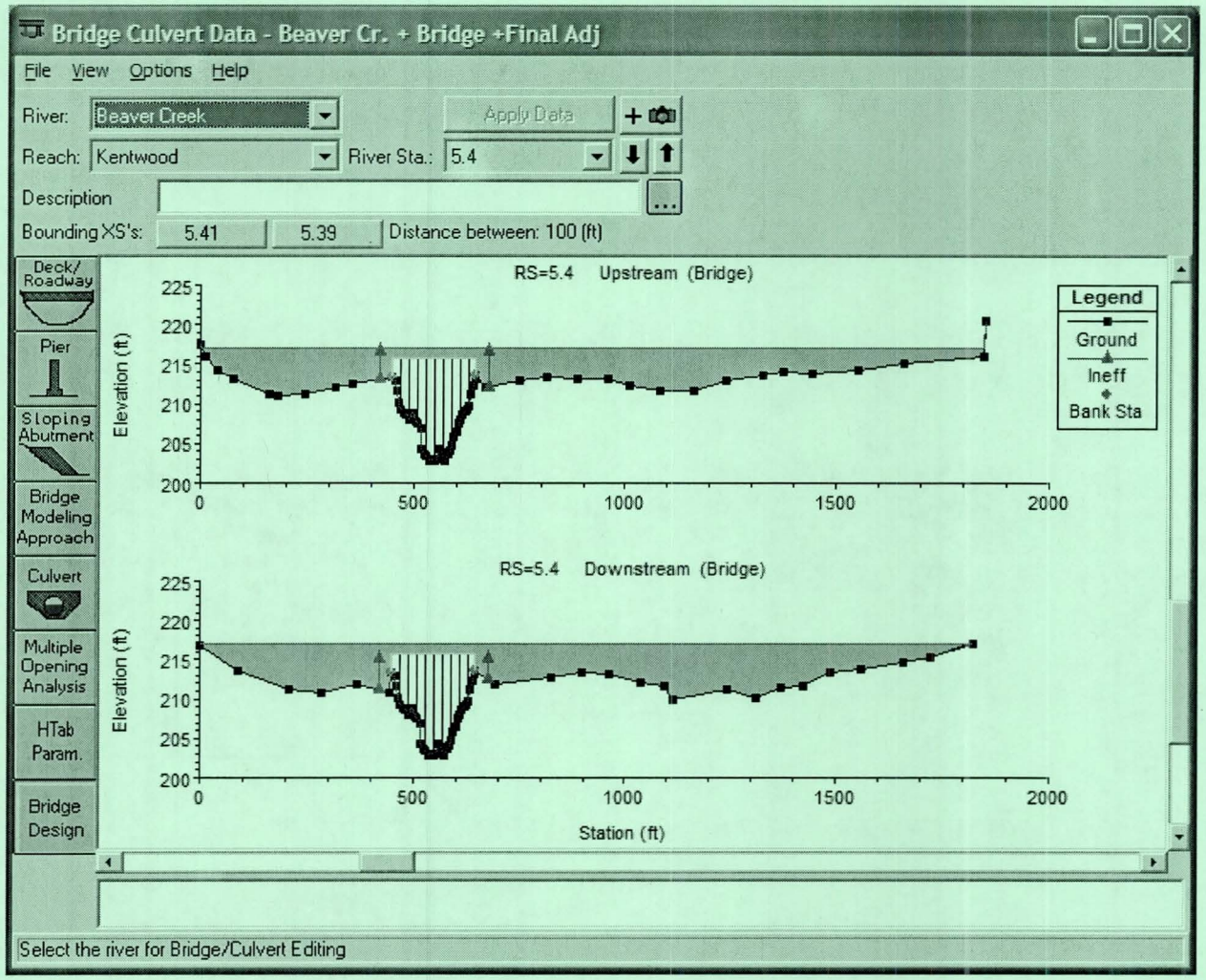
The bridge is located at river station 5.4 of Beaver Creek. The **Bridge Design Editor** can be used to enter the deck and road embankment and the pier data.



The Deck/Roadway and Pier data are entered in the Design Editor.

- Road embankment is at a constant elevation of 216.93 feet.
- The bridge low chord is at elevation of 215.7 feet.
- The bridge opening has vertical walls at 450 and 647 feet.
- The bridge has 9 piers. The piers are 1.25 ft. wide each and have a square nose. The piers are spaced 20.0 ft. apart on center, starting with the first pier at station 470.0 ft.
- The bridge deck is 40 feet wide, and the upstream side of the bridge deck is 30 feet from cross section immediately upstream of the bridge.





The Deck Distance must be entered and the location of the abutments and other roadway data should be checked to ensure they represent the bridge correctly. Check the pier data too.

Deck/Roadway Data Editor

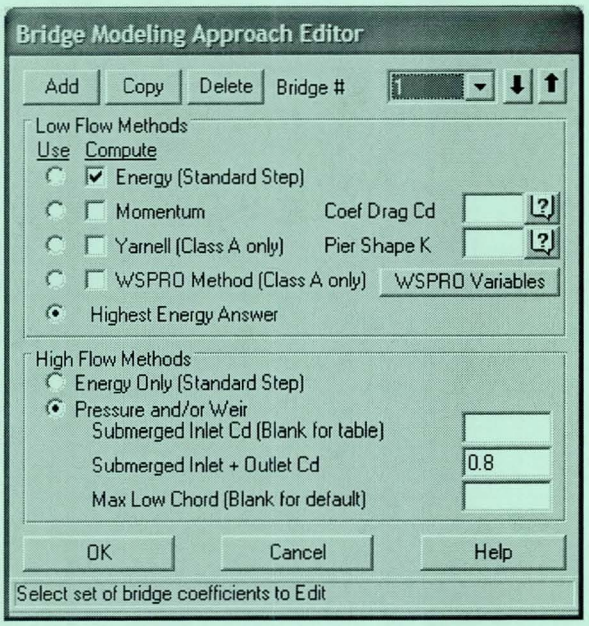
Del Row
Ins Row

Distance	Width	Weir Coef
30	40	2.6

	Upstream			Downstream		
	Station	high chord	low chord	Station	high chord	low chord
1	0.	216.93	202.7	0.	216.93	202.7
2	450.	216.93	202.7	450.	216.93	202.7
3	450.	216.93	215.7	450.	216.93	215.7
4	647.	216.93	215.7	647.	216.93	215.7
5	647.	216.93	202.7	647.	216.93	202.7
6	1850.	216.93	202.7	1824.	216.93	202.7
7						
8						

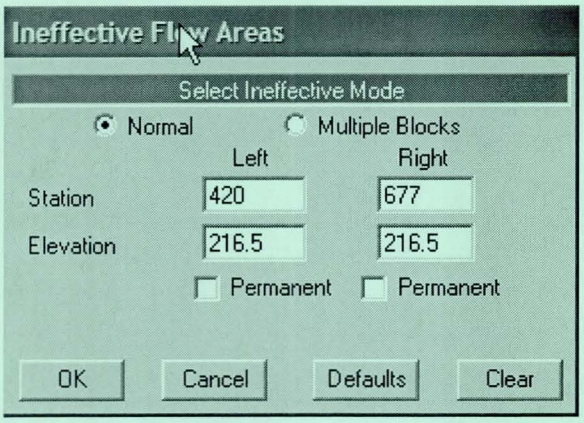
Select a bridge modeling approach for both low flow and high flow. Enter the necessary coefficients for the selected modeling approaches.

Multiple methods can be used initially; however, a single method should be selected prior to unsteady flow modeling. Based on previous modeling with this data, the shown combination is the best choice for this bridge.



Also, remember to add ineffective flow areas at the cross sections that bound the bridge. **SAVE THE GEOMETRIC DATA**.

Version 3.1.3 provides for direct access to the effective area option for the bounding sections. The initial model applied the Normal Effective Flow option, laterally offset from the abutment locations based on a rapid contraction and expansion of flow in the vicinity of the bridge. The elevation for the ineffective areas was set to just below the top of weir elevation.



2. Compute Steady-flow Water Surface Profiles

Once you have saved the geometric data, create a new steady flow plan. The steady-flow file is set for three profiles: 5000, 10000, and 14000 cfs. Give the plan a title and remember to enter a short ID for the plan, and save the data.

3. **Review The Results**

Review the results to ensure that the computed water surface profiles are reasonable. Make necessary adjustments to the model.

The Bridge Comparison table shows that for the 1974 flood event the pressure/weir flow solution was used.

Profile Output Table - Bridge Comparison

File Options Std. Tables Locations Help

HEC-RAS Plan: Init Bridge River: Beaver Creek Reach: Kentwood (Reload Data)

Reach	River Sta	Profile	E.G. US.	W.S. US.	Br Sel Method	Energy EG	Momen. EG	Yarnell EG	WSPRO EG	Prs O EG	Prs/Wr EG	Energy/Wr EG
			(ft)	(ft)		(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)
Kentwood	5.4	PF 1	213.45	213.19	Energy only	213.45						
Kentwood	5.4	PF 2	215.65	215.06	Energy only	215.65						
Kentwood	5.4	1974 flood	217.24	217.10	Press/Weir	216.81				217.51	217.24	

The bridge only table shows weir flow for the third profile

Profile Output Table - Bridge Only

File Options Std. Tables Locations Help

HEC-RAS Plan: Init Bridge River: Beaver Creek Reach: Kentwood (Reload Data)

Reach	River Sta	Profile	E.G. US.	Min El Prs	BR Open Area	Prs O WS	Q Total	Min El Weir Flow	Q Weir	Delta EG
			(ft)	(ft)	(sq ft)	(ft)	(cfs)	(ft)	(cfs)	(ft)
Kentwood	5.4	PF 1	213.45	215.70	1600.36		5000.00	216.94		0.22
Kentwood	5.4	PF 2	215.65	215.70	1600.36		10000.00	216.94		0.47
Kentwood	5.4	1974 flood	217.24	215.70	1600.36	217.39	14000.00	216.94	814.11	1.44

Check the cross sections around the bridge. The Six XS Bridge table shows that there was more overbank flow in the upstream of the bridge than downstream. This may be a result of the ineffective flow options being off upstream and turn on downstream. Also, the overbank flow at the upstream section is far greater than weir flow.

Profile Output Table - Six XS Bridge

File Options Std. Tables Locations Help

HEC-RAS Plan: Init Bridge River: Beaver Creek Reach: Kentwood Profile: 1974 flood (Reload Data)

Reach	River Sta	Profile	E.G. Elev	W.S. Elev	Crit W.S.	Frctn Loss	C & E Loss	Top Width	Q Left	Q Channel	Q Right	Vel Chnl
			(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(cfs)	(cfs)	(cfs)	(ft/s)
Kentwood	5.44	1974 flood	217.46	217.35		0.21	0.01	1843.69	3728.48	4428.00	5843.52	4.24
Kentwood	5.41	1974 flood	217.24	217.10	212.25			1844.17	2093.51	7673.70	4232.79	3.87
Kentwood	5.4 BR U	1974 flood	217.24	217.10	212.52			1844.17	198.65	13273.65	534.88	7.99
Kentwood	5.4 BR D	1974 flood	217.24	217.10	212.52			1824.00	198.65	13273.65	534.88	7.99
Kentwood	5.39	1974 flood	215.80	215.58	212.29	0.25	0.04	1699.42	1704.62	8228.83	4066.55	4.89
Kentwood	5.34*	1974 flood	215.51	215.37		0.25	0.01	1661.83	3211.67	4716.74	6071.59	4.63

A look at the all of the profiles shows that the flow data are consistent for

the lower flows that do not overtop the bridge.

Profile Output Table - Six XS Bridge

File Options Std. Tables Locations Help

HEC-RAS Plan: Init Bridge River: Beaver Creek Reach: Kentwood (Reload Data)

Reach	River Sta	Profile	E.G. Elev (ft)	W.S. Elev (ft)	Crit W.S. (ft)	Frctn Loss (ft)	C & E Loss (ft)	Top Width (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Vel Chnl (ft/s)
Kentwood	5.44	PF 1	213.79	213.54		0.34	0.00	1055.43	1101.28	3360.66	538.06	4.85
Kentwood	5.44	PF 2	216.14	216.01		0.35	0.14	1745.42	2758.55	4060.70	3180.75	4.42
Kentwood	5.44	1974 flood	217.46	217.35		0.21	0.01	1843.69	3728.48	4428.00	5843.52	4.24
Kentwood	5.41	PF 1	213.45	213.19	208.91	0.05	0.00	1090.16	0.02	4990.53	9.45	4.11
Kentwood	5.41	PF 2	215.65	215.06	210.95	0.09	0.02	1643.92	50.38	9842.97	106.65	6.23
Kentwood	5.41	1974 flood	217.24	217.10	212.25			1844.17	2093.51	7673.70	4232.79	3.87
Kentwood	5.4 BR U	PF 1	213.40	213.09	209.13	0.11	0.00	178.22		5000.00		4.48
Kentwood	5.4 BR U	PF 2	215.55	214.80	211.18	0.22	0.01	185.75		10000.00		6.98
Kentwood	5.4 BR U	1974 flood	217.24	217.10	212.52			1844.17	198.65	13273.65	534.88	7.99
Kentwood	5.4 BR D	PF 1	213.30	212.97	209.12	0.05	0.01	176.23		5000.00		4.56
Kentwood	5.4 BR D	PF 2	215.33	214.51	211.18	0.10	0.04	185.75		10000.00		7.25
Kentwood	5.4 BR D	1974 flood	217.24	217.10	212.52			1824.00	198.65	13273.65	534.88	7.99
Kentwood	5.39	PF 1	213.23	212.95	208.91	0.29	0.07	1192.45	38.63	4960.59	0.78	4.25
Kentwood	5.39	PF 2	215.18	214.50	210.95	0.36	0.27	1576.18	151.14	9805.53	43.33	6.67
Kentwood	5.39	1974 flood	215.80	215.58	212.29	0.25	0.04	1699.42	1704.62	8228.83	4066.55	4.89
Kentwood	5.34*	PF 1	212.87	212.74		0.28	0.03	1171.77	1065.15	2652.25	1282.60	3.86
Kentwood	5.34*	PF 2	214.55	214.42		0.25	0.02	1512.80	2283.09	3857.26	3859.64	4.29
Kentwood	5.34*	1974 flood	215.51	215.37		0.25	0.01	1661.83	3211.67	4716.74	6071.59	4.63

The effective area controlling elevation was lowered at the downstream section to allow overbank flow for the 1974 flood profile. Also, the overbank 'n' values were doubled at the bounding sections to reduce overbank conveyance. The weir coefficient was lowered to 2.5 to reflect the low head and the long weir length which will tend to be too efficient. The model was rerun and checked against the observed data.

Looking at the bridge tables we find that there is still a lot of flow in the overbanks compared with the computed weir flow.

Profile Output Table - Six XS Bridge

File Options Std. Tables Locations Help

HEC-RAS Plan: 1st Adj River: Beaver Creek Reach: Kentwood Profile: 1974 flood (Reload Data)

Reach	River Sta	Profile	E.G. Elev (ft)	W.S. Elev (ft)	Crit W.S. (ft)	Frctn Loss (ft)	C & E Loss (ft)	Top Width (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Vel Chnl (ft/s)
Kentwood	5.44	1974 flood	217.62	217.53	214.48	0.24	0.06	1844.49	3704.44	4300.54	5995.02	4.06
Kentwood	5.41	1974 flood	217.31	217.04	212.24			1843.59	1342.53	9964.47	2693.00	5.06
Kentwood	5.4 BR U	1974 flood	217.32	217.04	212.51			1843.59	284.42	12987.74	769.97	8.01
Kentwood	5.4 BR D	1974 flood	217.32	217.04	212.51			1820.26	284.15	12992.84	765.15	8.01
Kentwood	5.39	1974 flood	216.15	215.74	212.26	0.37	0.09	1713.61	1103.38	10245.66	2650.96	5.98
Kentwood	5.34*	1974 flood	215.69	215.46		0.32	0.06	1668.97	3155.60	5875.29	4969.11	5.70

We need to increase the n values in the overbanks and rerun until the weir flow and overbank flow balance. N values were increased to 0.6. Weir flow approximated the overbank flows.

Profile Output Table - Bridge Only

File Options Std. Tables Locations Help

HEC-RAS Plan: Final Steady River: Beaver Creek Reach: Kentwood Profile: 1974 flood

Reach	River Sta	Profile	E.G. US. (ft)	Min El Prs (ft)	BR Open Area (sq ft)	Prs O WS (ft)	Q Total (cfs)	Min El Weir Flow (ft)	Q Weir (cfs)	Delta EG (ft)
Kentwood	5.4	1974 flood	217.39	215.70	1600.36	217.23	14000.00	216.94	1473.65	1.07

Profile Output Table - Six XS Bridge

File Options Std. Tables Locations Help

HEC-RAS Plan: Final Steady River: Beaver Creek Reach: Kentwood Profile: 1974 flood

Reach	River Sta	Profile	E.G. Elev (ft)	W.S. Elev (ft)	Crit W.S. (ft)	Frctn Loss (ft)	C & E Loss (ft)	Top Width (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Vel Chnl (ft/s)
Kentwood	5.44	1974 flood	217.82	217.73		0.26	0.16	1845.42	3677.73	4163.30	6158.97	3.86
Kentwood	5.41	1974 flood	217.39	216.79	212.23			1841.40	509.71	12649.63	840.65	6.59
Kentwood	5.4 BR U	1974 flood	217.39	216.79	212.52				352.47	12681.92	948.18	7.50
Kentwood	5.4 BR D	1974 flood	217.39	216.79	212.51				352.47	12681.92	948.18	7.50
Kentwood	5.39	1974 flood	216.33	215.52	212.24	0.36	0.33	1694.32	429.86	12677.61	892.53	7.59
Kentwood	5.34*	1974 flood	215.63	215.49		0.23	0.02	1671.45	3283.69	4747.71	5968.60	4.59

Flow in right overbank.

4. Compare the Computed Results With Observed Data

The flow rate used in the third profile (Q=14,000 cfs), is the same as an observed event that occurred on May 22, 1974. The USGS obtained high water marks and average values were estimated from the high water marks.

Observed Water Surfaces for Comparison

River: Beaver Creek Delete Row

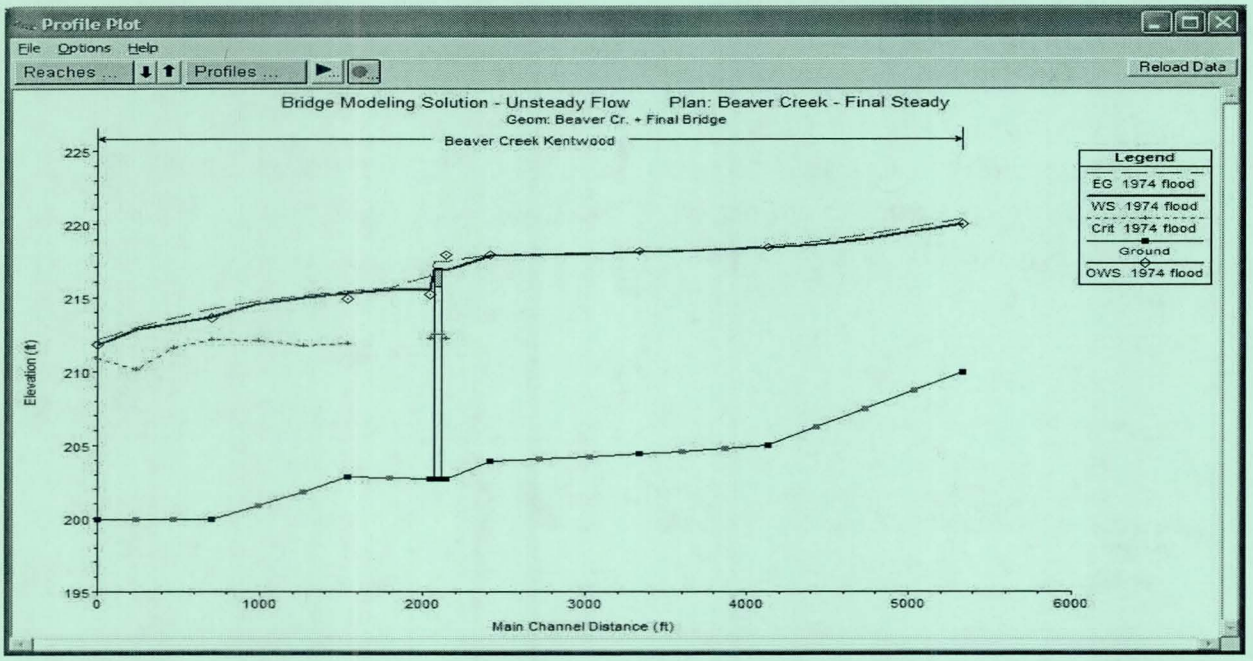
Reach: Kentwood River Sta.: 5.99 Add an Obs. WS Location

	Observed WS Location				Observed Water
	River	Reach	RS	PF 2	1974 flood
1	Beaver Creek	Kentwood	5.99		220
2	Beaver Creek	Kentwood	5.76		218.4
3	Beaver Creek	Kentwood	5.61		218.1
4	Beaver Creek	Kentwood	5.44		217.8
5	Beaver Creek	Kentwood	5.41		217.8
6	Beaver Creek	Kentwood	5.39		215.2
7	Beaver Creek	Kentwood	5.29		214.9
8	Beaver Creek	Kentwood	5.13		213.6
9	Beaver Creek	Kentwood	5.0		211.8

OK Cancel Help

The observed stages can be entered for the third profile by selecting ***Steady Flow Options****, and selecting* ***Observed WS****.*

The combination of changes produced a good reproduction of the observed profile data. The profile plot shows the entered observed water surface elevations for the third profile. Standard Table 1 was modified to show the observed data along with the computed water surface elevations. The results are shown below. Generally, the profile is within 0.1 ft.



Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

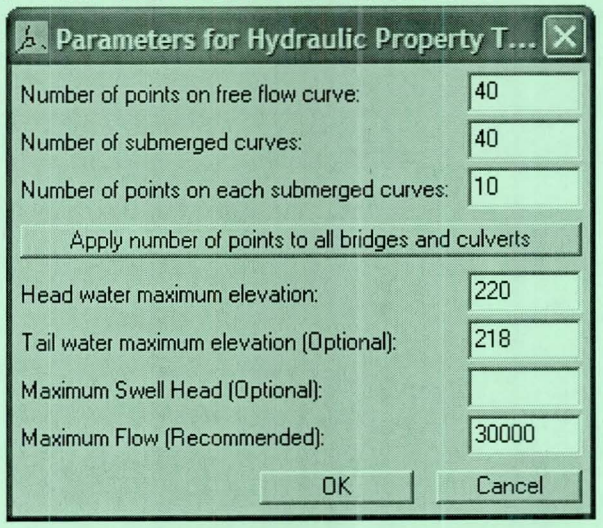
HEC-RAS Plan: Steady Final River: Beaver Creek Reach: Kentwood Profile: 1974 flood (Reload Data)

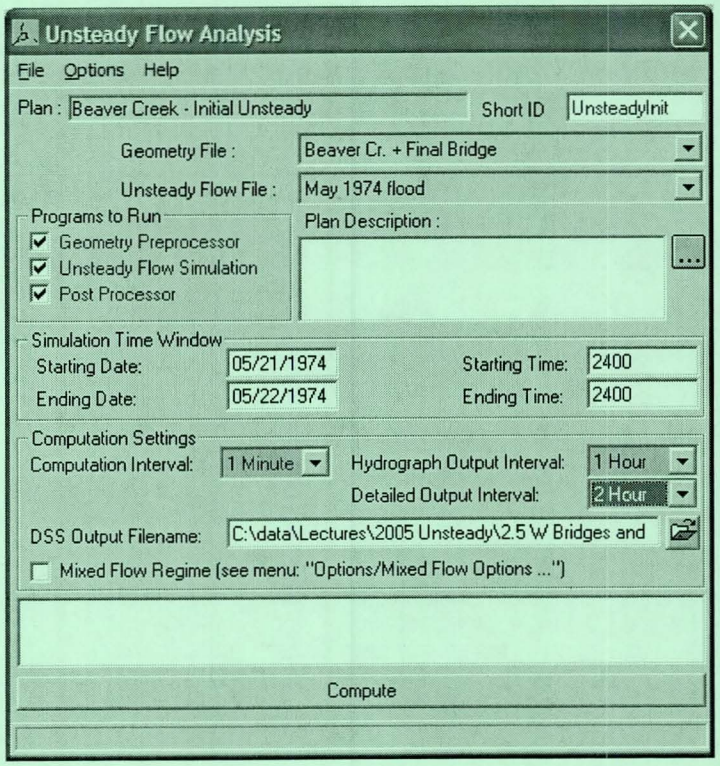
Reach	River Sta	Profile	Q Total	Min Ch El	W.S. Elev	Obs WS	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area	Top Width	Froude # Chl
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)	(ft)	
Kentwood	5.99	1974 flood	14000.00	209.90	220.03	220.00	220.33	0.002285	7.10	6667.30	1864.07	0.44
Kentwood	5.76	1974 flood	14000.00	205.00	218.39	218.40	218.54	0.000925	4.38	8969.35	1758.36	0.28
Kentwood	5.61	1974 flood	14000.00	204.40	218.01	218.10	218.07	0.000575	4.10	9338.38	1976.32	0.21
Kentwood	5.44	1974 flood	14000.00	203.90	217.73	217.80	217.82	0.000416	3.86	9146.09	1845.42	0.20
Kentwood	5.41	1974 flood	14000.00	202.70	216.79	217.80	217.39	0.001543	6.59	7785.32	1841.40	0.37
Kentwood	5.4		Bridge									
Kentwood	5.39	1974 flood	14000.00	202.70	215.52	215.20	216.33	0.002465	7.59	6465.78	1694.32	0.46
Kentwood	5.29	1974 flood	14000.00	202.80	215.28	214.90	215.38	0.000918	4.05	5912.75	1624.26	0.26
Kentwood	5.13	1974 flood	14000.00	199.90	213.59	213.60	214.10	0.002810	8.91	4910.23	1657.39	0.49
Kentwood	5.0	1974 flood	14000.00	199.90	211.80	211.80	212.19	0.003223	7.37	5626.93	1925.36	0.50

5. Set up an Unsteady Flow Simulation.

Based on your review of the steady-flow simulation, determine the appropriate settings for the bridge processing (HTAB). Set the HTAB values and save the geometry file. You don't have to rename it.

The range of elevations for the processing of bridge data should exceed the expected range of flow and elevations. The tailwater and headwater elevations could be set one-foot higher than the values for the 1974 flood. Also, the maximum flow could be set to the maximum expected in future applications.





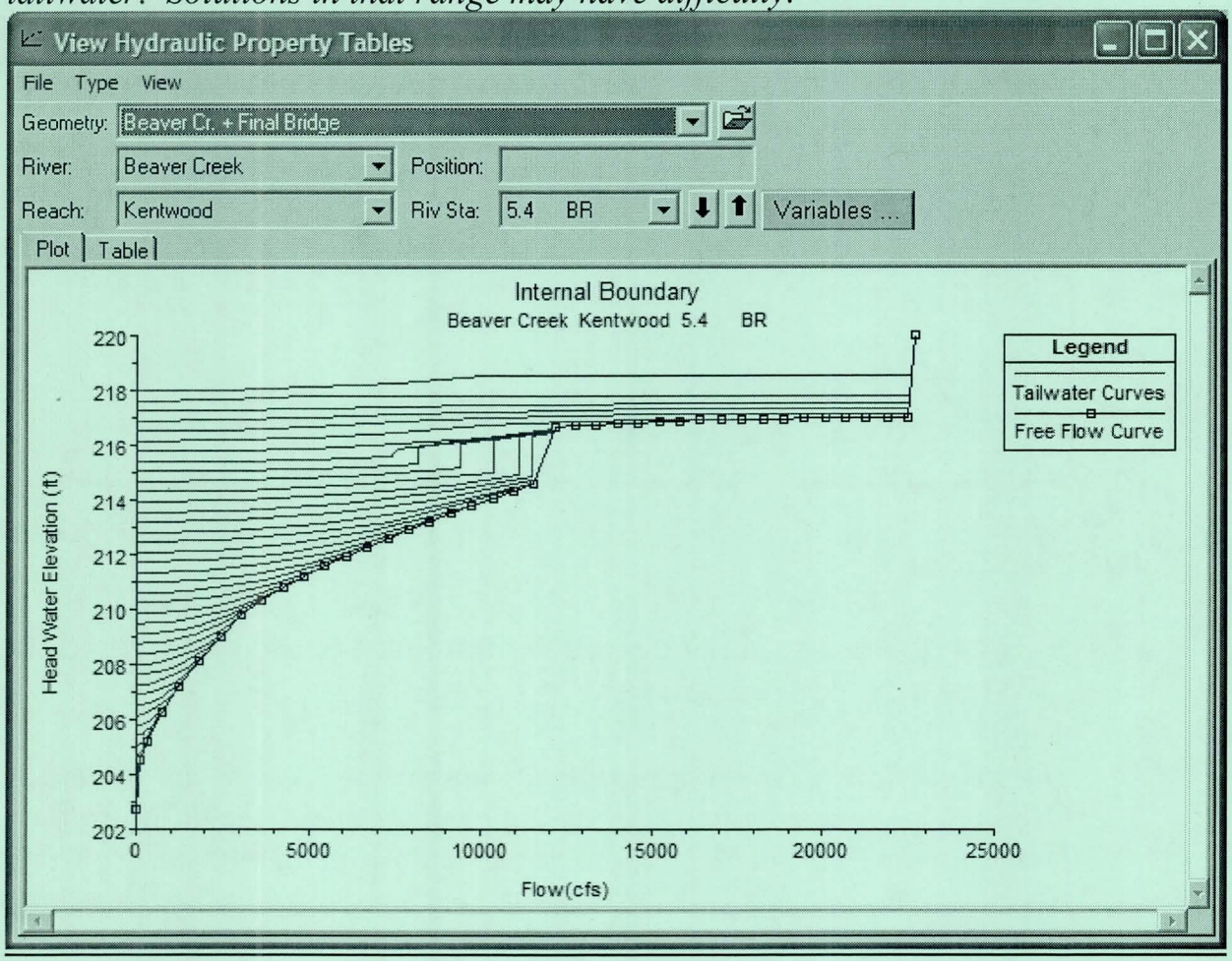
The flood hydrograph used in the previous workshop will be applied here, along with the downstream boundary and initial flow. The hydrograph represents the May 1974 flood.

Set up an unsteady flow simulation. Decide the appropriate time interval and set any required parameters. Save the plan as a new name and run the simulation.

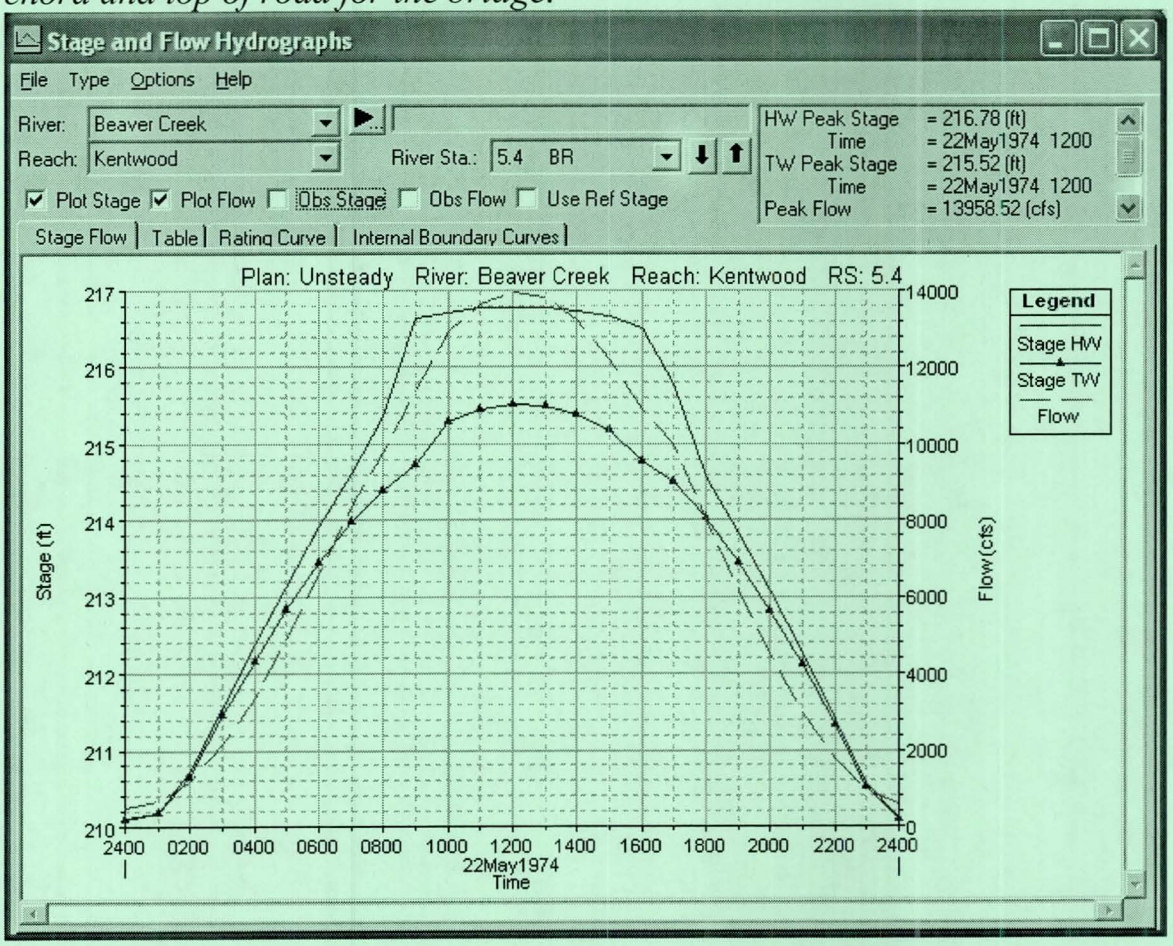
6. Review the Results for the Unsteady-flow Simulation.

Review the results to ensure that they are reasonable. Compare the results for the maximum stage to those from the third profile with the observed data. Make any necessary changes and run the simulation again, until results look reasonable. Keep track of the changes you make.

The Hydraulic Properties Plot for the bridge is shown above. The family of curves looks reasonable, except for the zone around elevation 216. The low-chord elevation is 215.70, where pressure flow using the gate equation will begin. The sharp breaks for several curves reflect the upstream water surface elevation based on the gate equation, which is independent of tailwater. Solutions in that range may have difficulty.



The stage and flow hydrograph plot, for the bridge solution, is shown below. Generally, the hydrographs appear reasonable up to the low-chord elevation. Then there is a break in the hydrographs as they hit the low chord and top of road for the bridge.



Questions

1. How do your steady-flow results compare to the observed data? What methods did you use?

The steady flow results were a little low immediately upstream from the bridge. The pressure and weir equation were used for the third profile. Lowering the weir coefficient to 2.5 and increasing the overbank n values to reduce overbank conveyance around the bridge gave reasonable results, closer to the observed data.

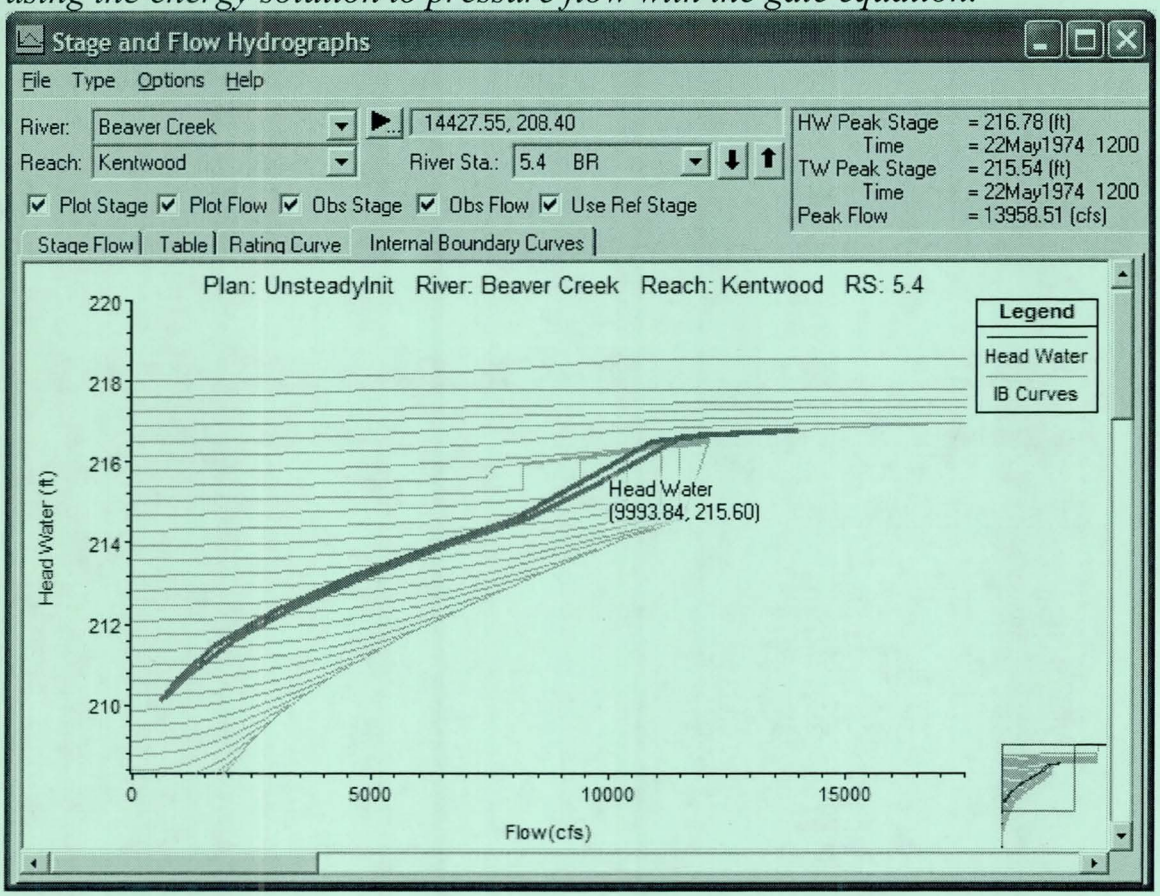
2. What settings did you use for HTAB on bridge model processing?

The HTAB settings limited the tailwater and headwater to values above the 1974 flood. Also, the maximum flow was set to 30,000 – twice the 1974

peak. The number of curves were the default setting.

3. What changes did you make to the unsteady flow model, after you were satisfied with the steady flow solution?

The headwater stage hydrograph is a little ragged around the bridge deck. This can be expected when pressure and weir flow is present; however, it might be improved with a few adjustments. A new feature is a plot to track the solution through the bridge rating, as shown in the Internal Boundary Curve below. The ragged zone is through the bridge when flow goes from using the energy solution to pressure flow with the gate equation.



If we were not satisfied with the internal boundary curves at the transition from the energy solution to the pressure solution, we could adjust the gate coefficient to smooth out the flow transition. If a value of 0.40 was used a lower headwater would result from the same tailwater (at a given flow). A comparison of the results showed no significant improvement in the solution and was not used.

The Bridge Only table is shown below for profiles around the peak flow.

Profile Output Table - Bridge Only

File Options Std. Tables Locations Help

HEC-RAS Plan: Unsteady River: Beaver Creek Reach: Kentwood — Reload Data

Reach	River Sta	Profile	E.G. US.	Min El Prs	BR Open Area	Prs O WS	Q Total	Min El Weir Flow	Q Weir	Delta EG
			(ft)	(ft)	(sq ft)	(ft)	(cfs)	(ft)	(cfs)	(ft)
Kentwood	5.4	22MAY1974 0600	214.28	215.70	1600.36		6586.95	216.94		0.34
Kentwood	5.4	22MAY1974 0800	215.90	215.70	1600.36		9762.22	216.94		0.51
Kentwood	5.4	22MAY1974 1000	217.24	215.70	1600.36	216.91	12898.07	216.94	799.92	1.22
Kentwood	5.4	22MAY1974 1200	217.39	215.70	1600.36	217.21	13958.52	216.94	1446.66	1.07
Kentwood	5.4	22MAY1974 1400	217.29	215.70	1600.36	217.00	13220.96	216.94	990.45	1.15
Kentwood	5.4	22MAY1974 1600	217.02	215.70	1600.36	216.51	10898.96	216.94		1.35
Kentwood	5.4	22MAY1974 1800	215.02	215.70	1600.36		8021.49	216.94		0.40

Upstream energy grade elevation at bridge or culvert (specific to that opening, not necessarily the weighted average).

The weir flow values from post-processing can be compared to the flow transitions upstream and down from the bridge. The Six XS Bridge Table shows those results for the weir profiles, as shown below. Flow is conserved in the overbanks and overbank flow is comparable to the computed weir flow.

Profile Output Table - Six XS Bridge

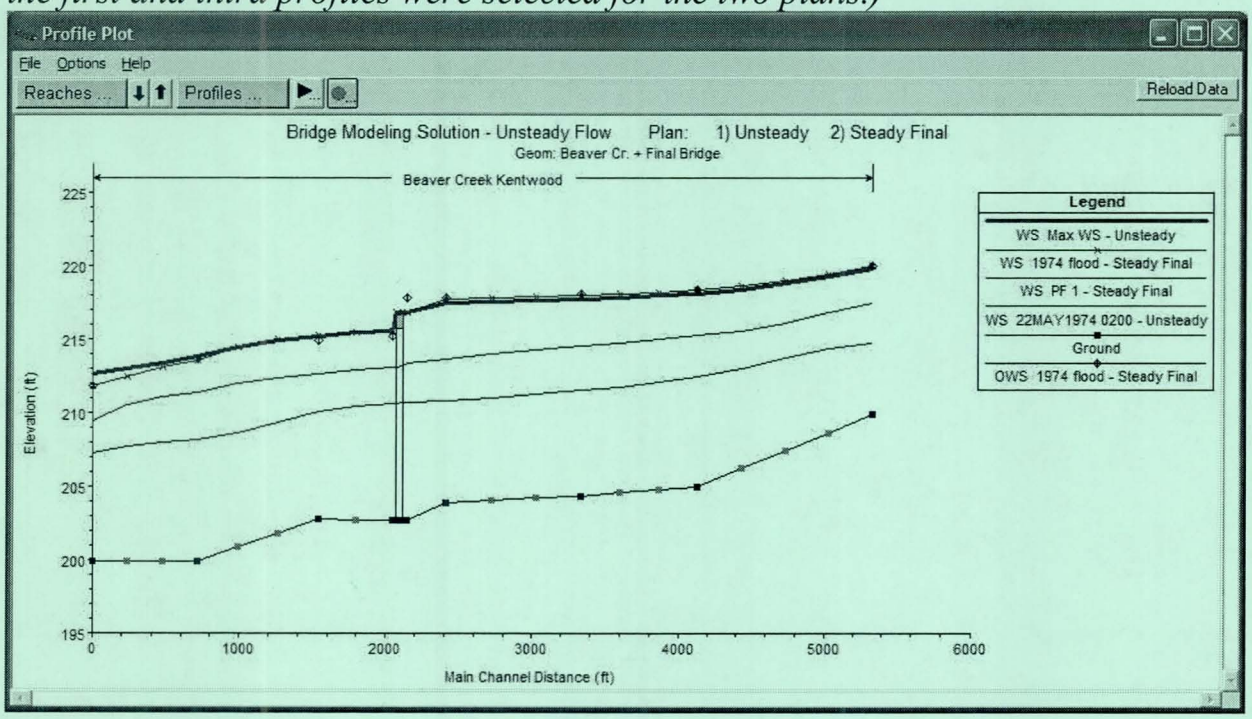
File Options Std. Tables Locations Help

HEC-RAS Plan: Unsteady River: Beaver Creek Reach: Kentwood — Reload Data

Reach	River Sta	Profile	E.G. Elev	W.S. Elev	Crit W.S.	Frctn Loss	C & E Loss	Top Width	Q Left	Q Channel	Q Right	Vel Chnl
			(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(cfs)	(cfs)	(cfs)	(ft/s)
Kentwood	5.44	22MAY1974 1000	217.39	217.29		0.27		1843.41	3450.07	4131.21	5343.31	3.98
Kentwood	5.44	22MAY1974 1200	217.54	217.44		0.29		1844.07	3707.36	4355.00	5901.62	4.14
Kentwood	5.44	22MAY1974 1400	217.43	217.34		0.27		1843.62	3518.67	4188.01	5497.33	4.02
Kentwood	5.41	22MAY1974 1000	217.24	216.71	211.90			1840.79	463.89	11676.89	757.29	6.12
Kentwood	5.41	22MAY1974 1200	217.39	216.78	212.22			1841.39	508.04	12612.83	837.66	6.57
Kentwood	5.41	22MAY1974 1400	217.29	216.74	212.00			1840.98	477.38	11961.69	781.89	6.26
Kentwood	5.4 BR U	22MAY1974 1000	217.24	216.72	212.18				197.05	12185.20	530.58	7.33
Kentwood	5.4 BR U	22MAY1974 1200	217.39	216.78	212.51				348.12	12665.51	936.50	7.49
Kentwood	5.4 BR U	22MAY1974 1400	217.29	216.74	212.27				241.36	12337.11	649.70	7.38
Kentwood	5.4 BR D	22MAY1974 1000	217.24	216.72	212.18				197.05	12185.20	530.58	7.33
Kentwood	5.4 BR D	22MAY1974 1200	217.39	216.78	212.50				348.12	12665.51	936.50	7.49
Kentwood	5.4 BR D	22MAY1974 1400	217.29	216.74	212.27				241.36	12337.11	649.70	7.38
Kentwood	5.39	22MAY1974 1000	216.02	215.28		0.37		1670.11	375.04	11754.57	768.46	7.24
Kentwood	5.39	22MAY1974 1200	216.32	215.52		0.37		1694.60	428.89	12638.96	890.67	7.56
Kentwood	5.39	22MAY1974 1400	216.14	215.39		0.36		1682.69	394.26	12014.35	812.35	7.30
Kentwood	5.34*	22MAY1974 1000	215.31	215.16		0.25		1641.18	3026.65	4570.66	5277.19	4.60
Kentwood	5.34*	22MAY1974 1200	215.55	215.40		0.24		1664.69	3275.02	4787.21	5891.23	4.67
Kentwood	5.34*	22MAY1974 1400	215.42	215.27		0.24		1654.16	3109.84	4626.25	5499.92	4.59

4. How did the unsteady-flow simulation compare to the steady-flow profile with 14,000 cfs?

A comparison plot is shown below for the Unsteady flow Maximum Water Surface (Dark Solid line) and the 14,000 cfs steady flow (line with X). The profile compare well, but in general, the unsteady flow profile is lower. We can expect the unsteady flow profile answer to be slightly lower because the unsteady flow solution does not use the contraction/expansion coefficients for computing energy losses. At the downstream boundary, it is evident that the unsteady solution started at a higher water surface elevation than the steady flow solution. (The lower two profiles are the third profile unsteady-flow profile from post processing and the first profile steady flow because the first and third profiles were selected for the two plans.)



Lateral Spillways and Weirs, Storage Areas, and SA Connections Workshop Solution

Tasks

1. **Develop Storage Areas, Lateral Structures, and Storage Area Connections**

   a. Construct storage areas for Site 1 and 2.

   Storage areas were drawn over Site 1 and 2 utilizing the "Storage Area" button in the "Tools" row of the **Geometric Data Editor**. The storage areas were named Upstream SA and Downstream SA, denoted their location on the schematic.

   The data was entered in the **Geometric Data Editor** by selecting the "Storage Area" button, located in the "Editors" column. The **Area Times Depth Method** was selected for both storage areas. Upstream SA minimum elevation was set to 209 feet and the area to 80 acres. The Downstream SA minimum elevation was set to 207 feet and the area to 85 acres.

   b. In the **Unsteady Flow Data Editor** set the initial water surface elevations of the storage areas.

   In the editor the **Initial Conditions** tab was selected and the elevations were set to 209 feet and 207 feet for the upstream and downstream storage areas, respectively. These elevations are the minimum elevations of both storage areas, thus the storage areas are dry at the beginning of simulation.

   c. Connect the storage areas to each other with a storage area connection. The connection should consist of a weir and culvert.

   The connection was drawn utilizing the "SA Conn" tool found in the **Geometric Data Editor**, located in the "Tools" row. By beginning the connection within the upstream storage area and ended the connection in the downstream storage area the connection is linked to these two storage areas. The connection was named SA Transfer.

   The connection editor was opened with the "Storage Area Conn." button on the **Geometric Data Editor**, located in the "Editors" column. By selecting this option Figure 1 was displayed.

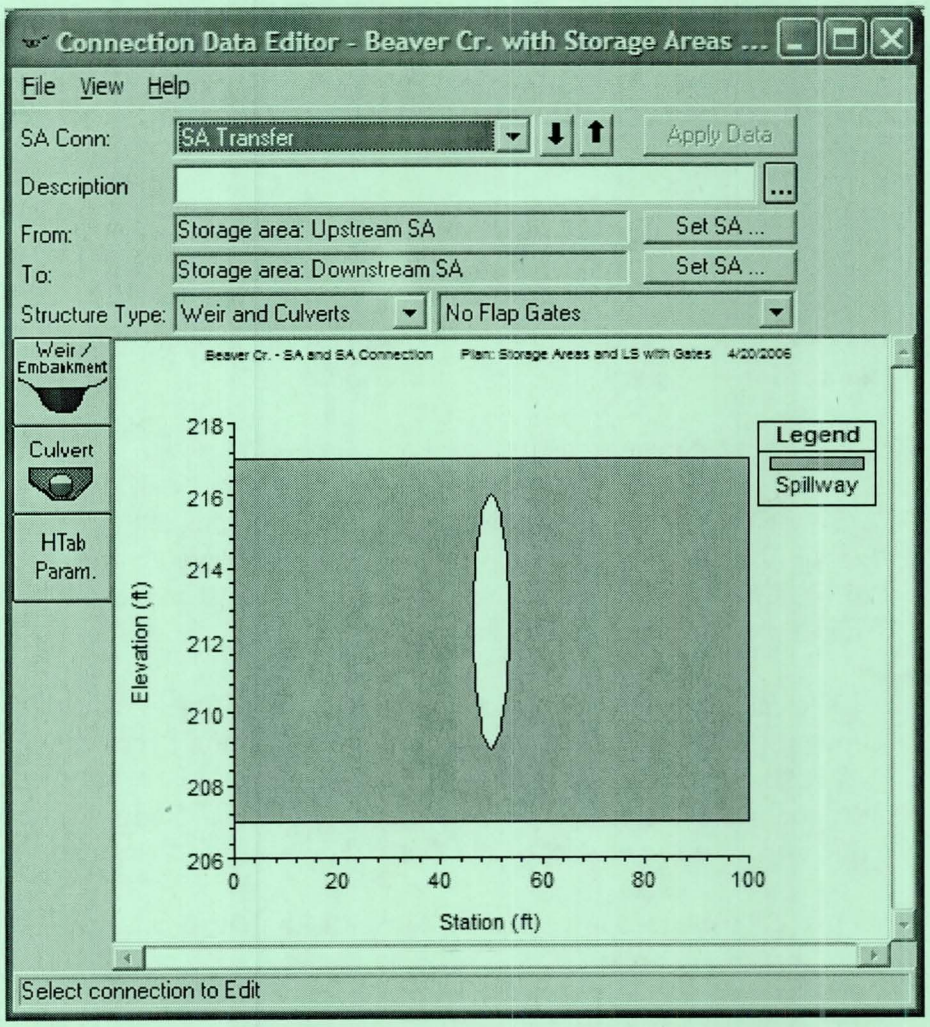


Figure 1 SA Connection Editor

From this editor the **Weir/Embankment** tool was selected. Data was entered for the weir geometry, as shown in Figure 2.

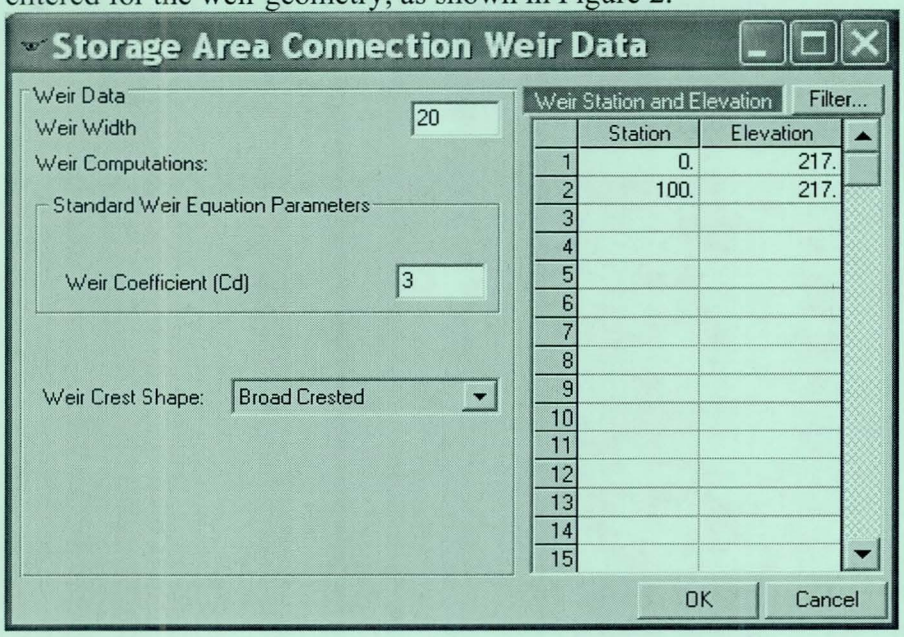


Figure 2 Weir Editor

After editing the weir, the **Culvert** tool was selected from the **SA Conn Data Editor**. Data was entered for a single culvert as shown in Figure 3.

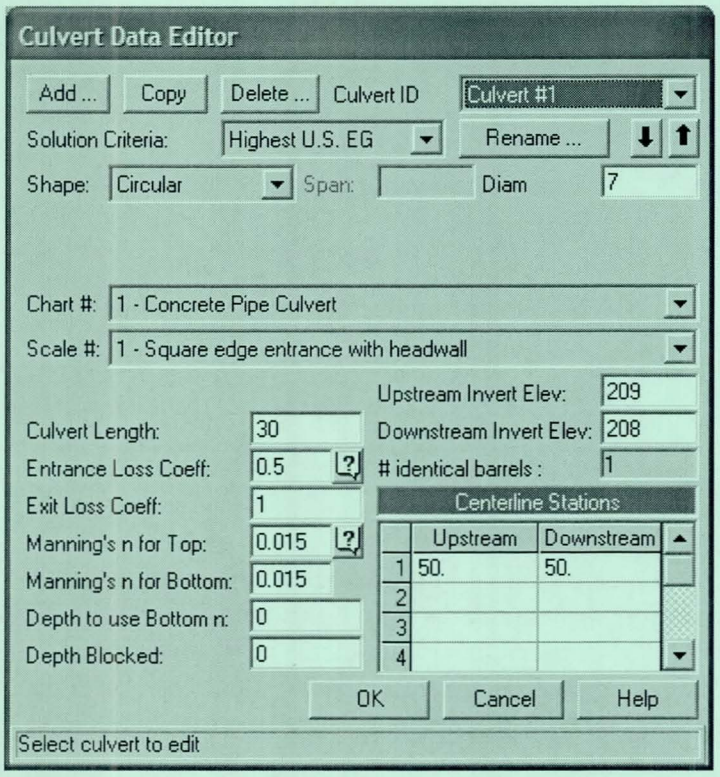


Figure 3 Culvert Data Editor

d. Connect the two storage areas to the reach via lateral structures.

The lateral structures were added using the lateral weir tool found in the **Geometric Data Editor**, located in the "Editors" section. After pressing this button the **Lateral Structure Editor** shown in Figure 4 is displayed.

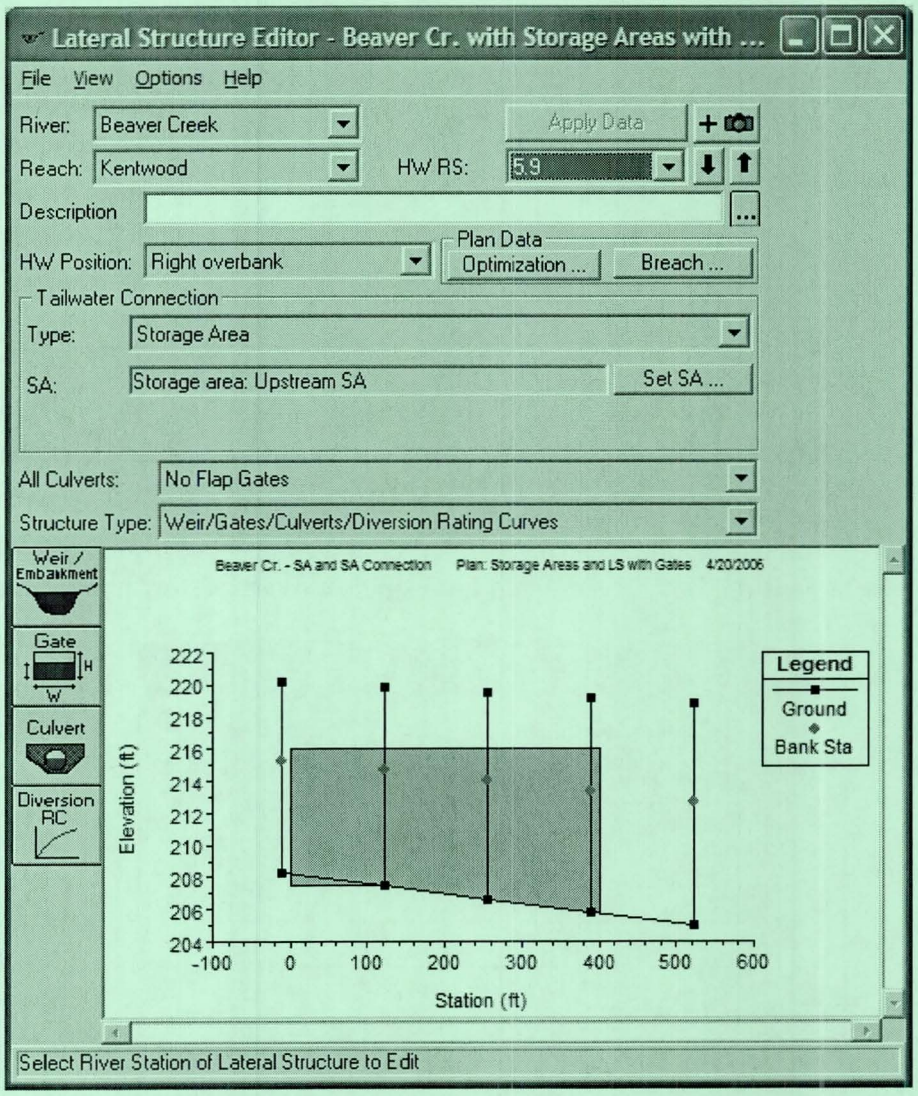


Figure 4 Lateral Structure Editor

Under the **Options** menu **Add a Lateral Structure** option was selected. The River Station was entered as 5.9. Next, the **Weir/Embankment** button was selected. The dimensions of the lateral weir were entered as shown in Figure 5. The weir was connected to SA Upstream by selecting the **Set SA** button and choosing "SA Upstream."

The same method was used to add the second lateral weir at River Station 5.570. The dimensions of this weir are shown in Figure 6. After completing the geometry, the file was saved and the unsteady flow simulation was run.

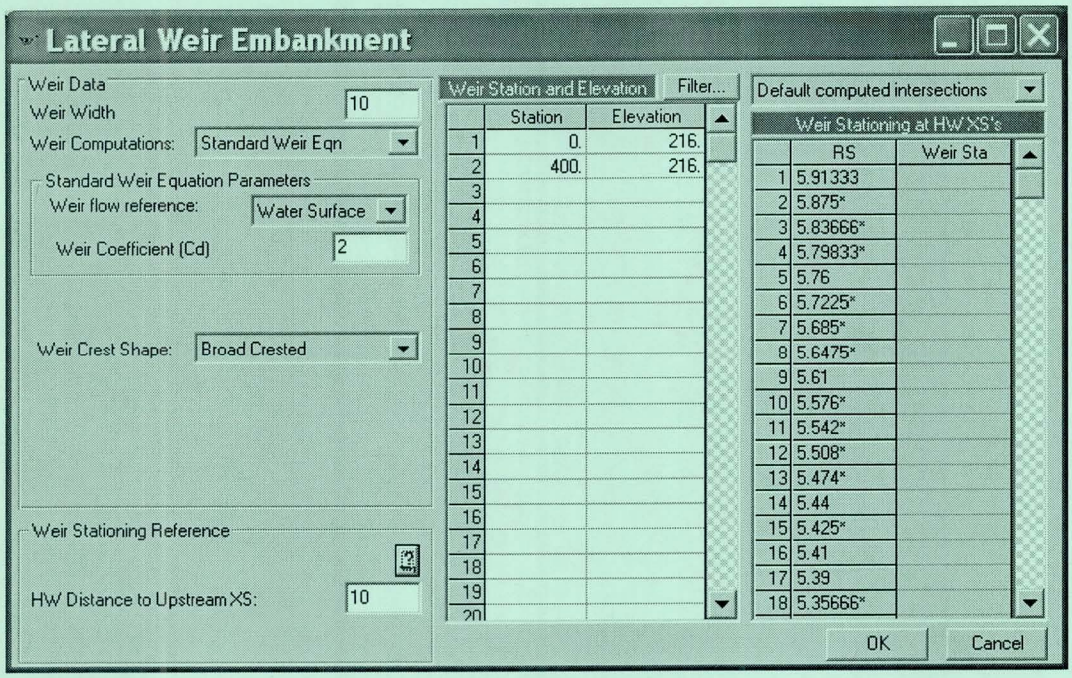


Figure 5 Lateral Weir Located at RS 5.9

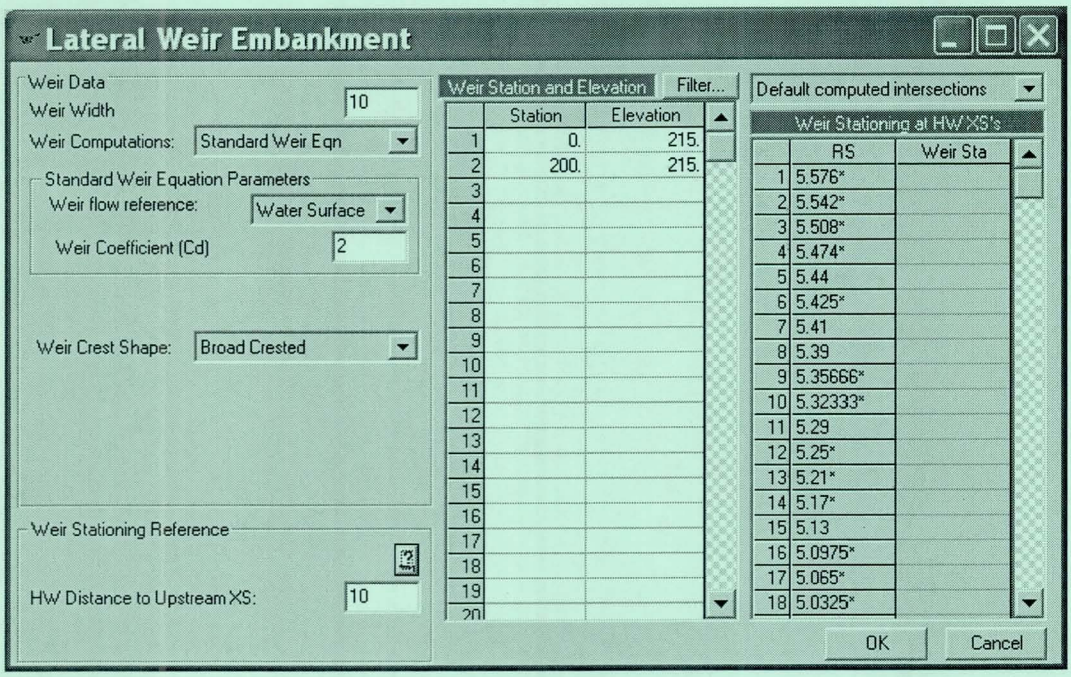


Figure 6 Lateral Weir Located at RS 5.570

e. Save a new plan for Task 1 with the new geometry file, then proceed with the simulation.

In the **Unsteady Flow Analysis Window,** a new plan with the name "Lateral Weirs and Storage– Task 1" and the short ID "Task 1." was saved. The simulation was then initiated.

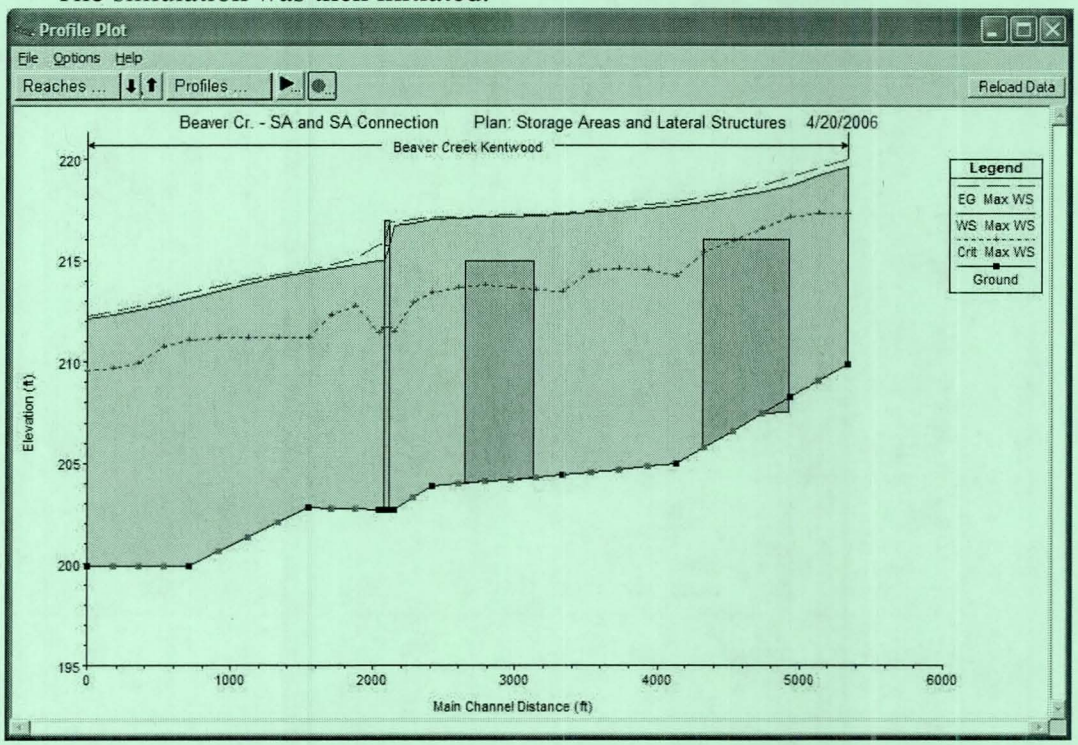


Figure 7 Profile Plot of Maximum Water Surface Profile

Profile Output Table - Bridge Only

File Options Std. Tables Locations Help

HEC-RAS Plan: Task 1 River: Beaver Creek Reach: Kentwood [Reload Data]

Reach	River Sta	Profile	E.G. US.	Min El Prs	BR Open Area	Prs O WS	Q Total	Min El Weir Flow	Q Weir	Delta EG
			(ft)	(ft)	(sq ft)	(ft)	(cfs)	(ft)	(cfs)	(ft)
Kentwood	5.4	Max WS	216.92	215.70	1600.36	216.65	11447.43	216.94		1.10
Kentwood	5.4	21MAY1974 2400	209.61	215.70	1600.36		516.50	216.94		0.02
Kentwood	5.4	22MAY1974 0200	210.62	215.70	1600.36		1188.30	216.94		0.04
Kentwood	5.4	22MAY1974 0400	212.42	215.70	1600.36		3378.41	216.94		0.15
Kentwood	5.4	22MAY1974 0600	214.14	215.70	1600.36		6411.08	216.94		0.32
Kentwood	5.4	22MAY1974 0800	215.31	215.70	1600.36		8841.77	216.94		0.44
Kentwood	5.4	22MAY1974 1000	215.97	215.70	1600.36		10417.19	216.94		0.50
Kentwood	5.4	22MAY1974 1200	216.39	215.70	1600.36	216.53	10828.41	216.94		1.22
Kentwood	5.4	22MAY1974 1400	216.89	215.70	1600.36	216.64	11396.06	216.94		1.10
Kentwood	5.4	22MAY1974 1600	216.21	215.70	1600.36		10621.94	216.94		0.50
Kentwood	5.4	22MAY1974 1800	215.26	215.70	1600.36		8641.06	216.94		0.42
Kentwood	5.4	22MAY1974 2000	213.45	215.70	1600.36		4952.94	216.94		0.22
Kentwood	5.4	22MAY1974 2200	211.46	215.70	1600.36		1925.09	216.94		0.07
Kentwood	5.4	22MAY1974 2400	210.01	215.70	1600.36		706.14	216.94		0.02
Kentwood	5.4	23MAY1974 0200	209.45	215.70	1600.36		443.75	216.94		0.01
Kentwood	5.4	23MAY1974 0400	208.90	215.70	1600.36		290.80	216.94		0.01
Kentwood	5.4	23MAY1974 0600	208.80	215.70	1600.36		267.05	216.94		0.01

Figure 8 Profile Output Table for Bridge Only

f. View output and adjust geometry until the bridge is not overtopped.

Figure 7 displays the profile plot with the maximum water surface profile displayed. From this plot it can be seen that the maximum profile does not overtop the bridge for the given geometry. Additionally, Figure 8 displays the **Profile Output Table** for the **Bridge Only**. Under the **Options** menu **Profiles** was selected, and every time step was selected. The data shows that there was no weir flow.

2. Place Gates on the Downstream Lateral Structure

a. Construct the gates along the downstream lateral weir.

From the **Lateral Structure Data Editor** the downstream lateral weir at RS 5.570 was selected. Next, the **Gate** button was selected. Figure 9 displays the data entered for the gates. Ten gates were entered; each gate is 4 feet apart. After entering all gate information the geometry file was saved as "Beaver Cr. with Gates."

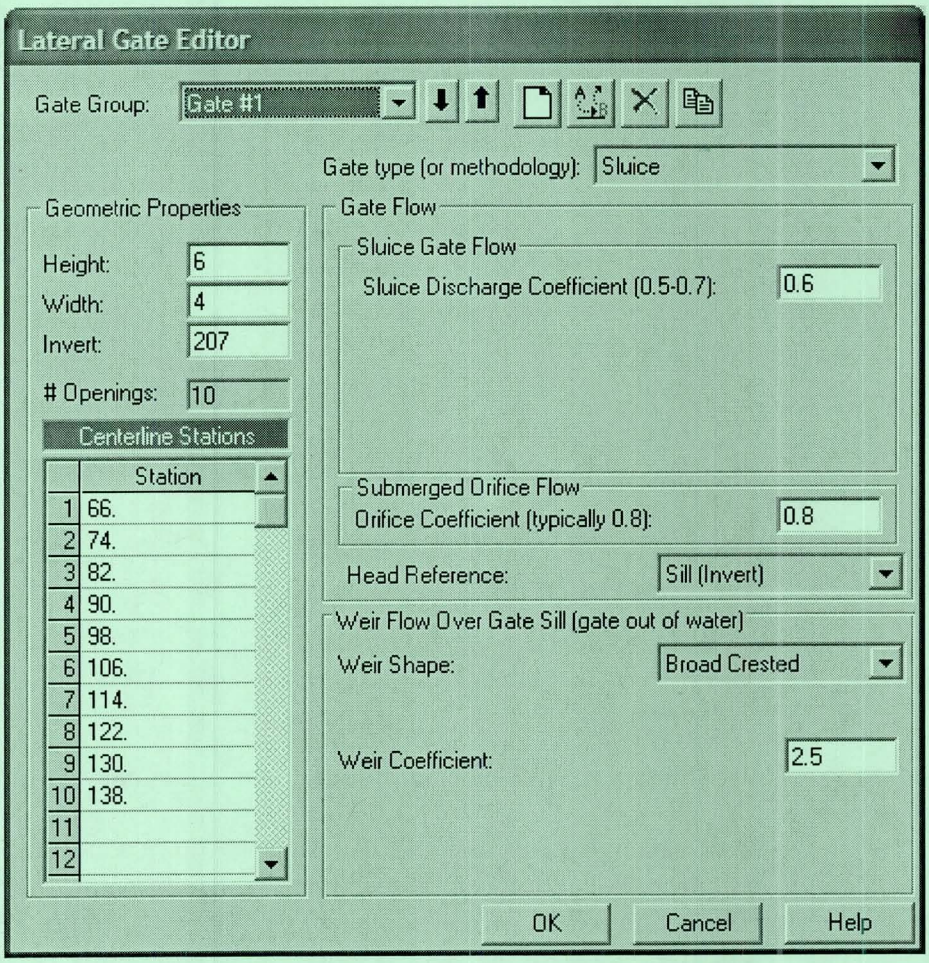


Figure 9 Lateral Gate Editor

b. In the **Unsteady Flow Data Editor** develop a time series of operation for the gates that will release the water in the downstream storage area after the flood wave.

In the **Unsteady Data Editor** RS 5.570 was selected and then the **Add a Boundary Location** button was depressed. A time series was developed that opened the gates after the flood wave had passed. After entering the times series information the unsteady flow file was saved as "Unsteady Flow with Gates." Figure 10 displays a plot of the time series for gate openings.

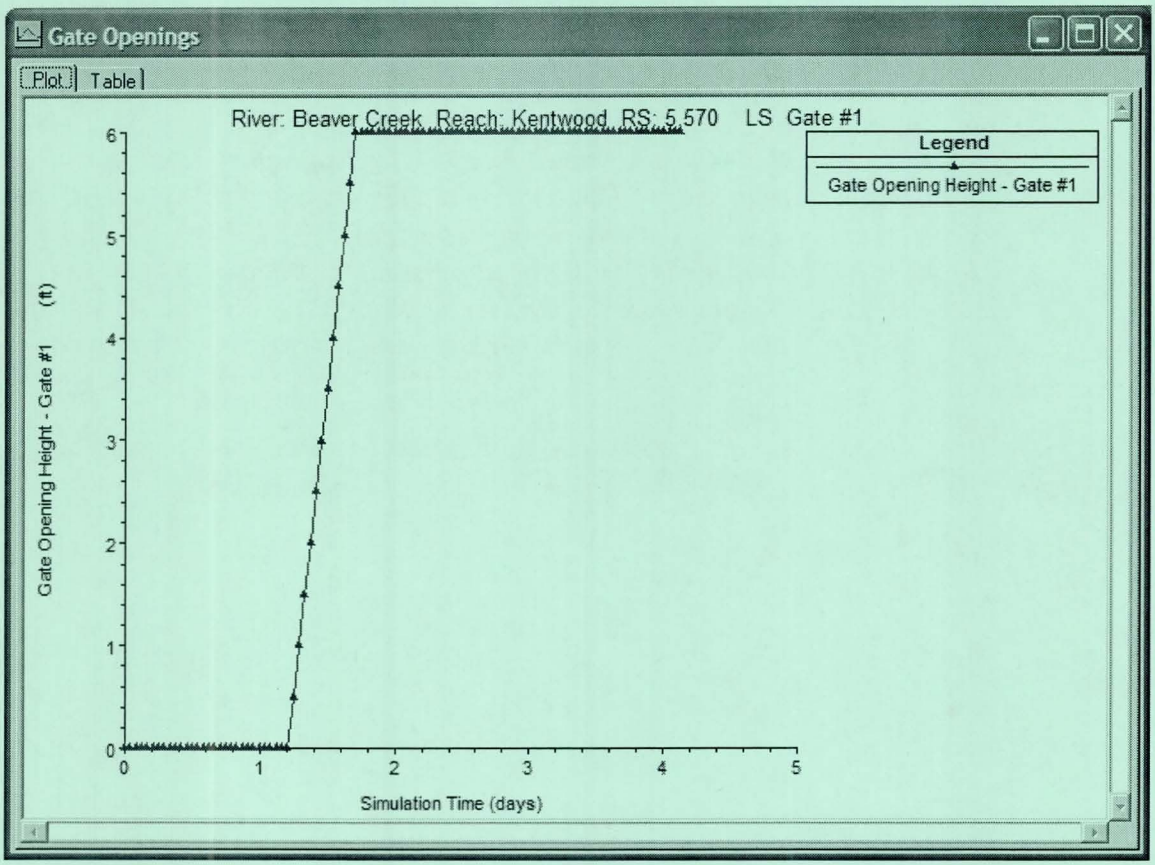


Figure 10 Times Series of Gate Openings

c. Save a new plan for Task 2 with the new geometry and unsteady flow files and then proceed with the simulation.

In the **Unsteady Flow Analysis Window,** a new plan with the name "Gates – Task 2" and the short ID "Task 2." was saved. The simulation was then initiated.

Questions

1. Did your design meet the requirement to prevent weir flow over the highway bridge?

As shown in Figures 7 and 8 the design met the requirement to prevent weir flow over the highway bridge. Pressure flow did occur during the simulation but the bridge was never overtopped.

2. When did the maximum flowrate occur in the storage areas? What was the maximum stage of both storage areas? What was the maximum flowrate between the two storage areas? What was the maximum flowrate over the lateral structure located at River Station 5.9?

Figure 11 and 12 display the ***Stage and Flow Hydrographs*** *for the upstream and downstream storage areas. As seen from the figures, the maximum flowrate occurred in the upstream storage area on the May 22nd, 1974 at 1100 hours (1,843 cfs). The maximum flow in the downstream storage area occurred on May 22nd, 1974 at 1400 hours (1,899 cfs). The maximum stages of the storage areas were 218.1 feet and 216.7 feet for upstream and downstream, respectively.*

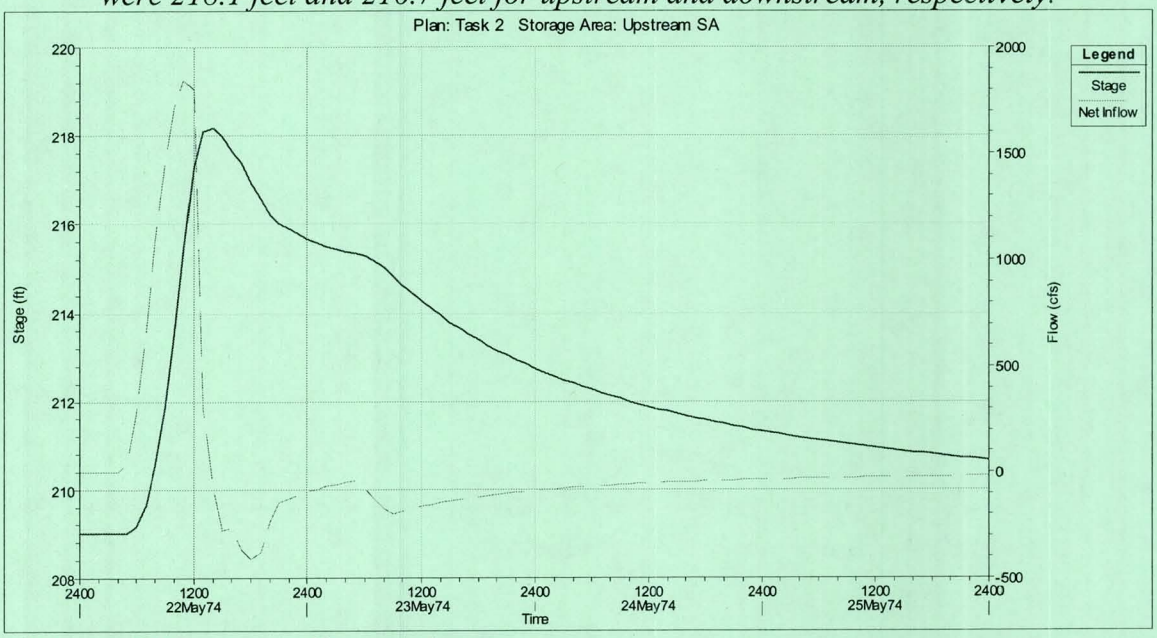


Figure 11 Upstream Storage Area

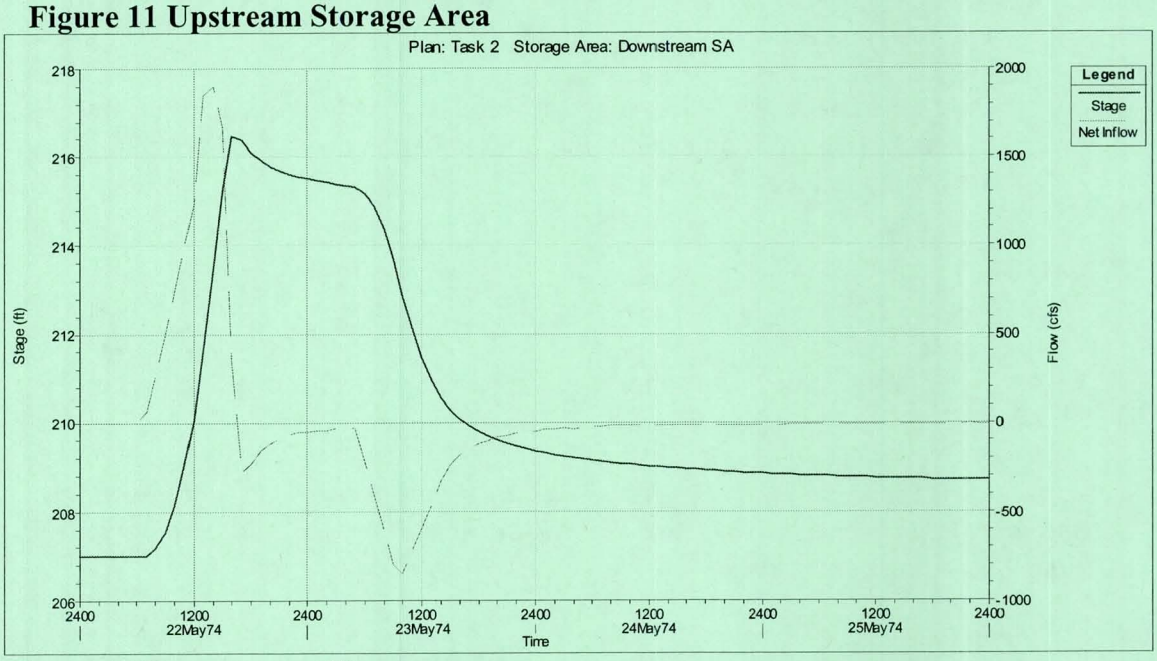


Figure 12 Downstream Storage Area

The maximum flowrate between the two storage areas can be viewed from the ***Stage and Flow Hydrograph*** *of the hydraulic connection, as shown in Figure 13. The maximum flowrate was 671 cfs.*

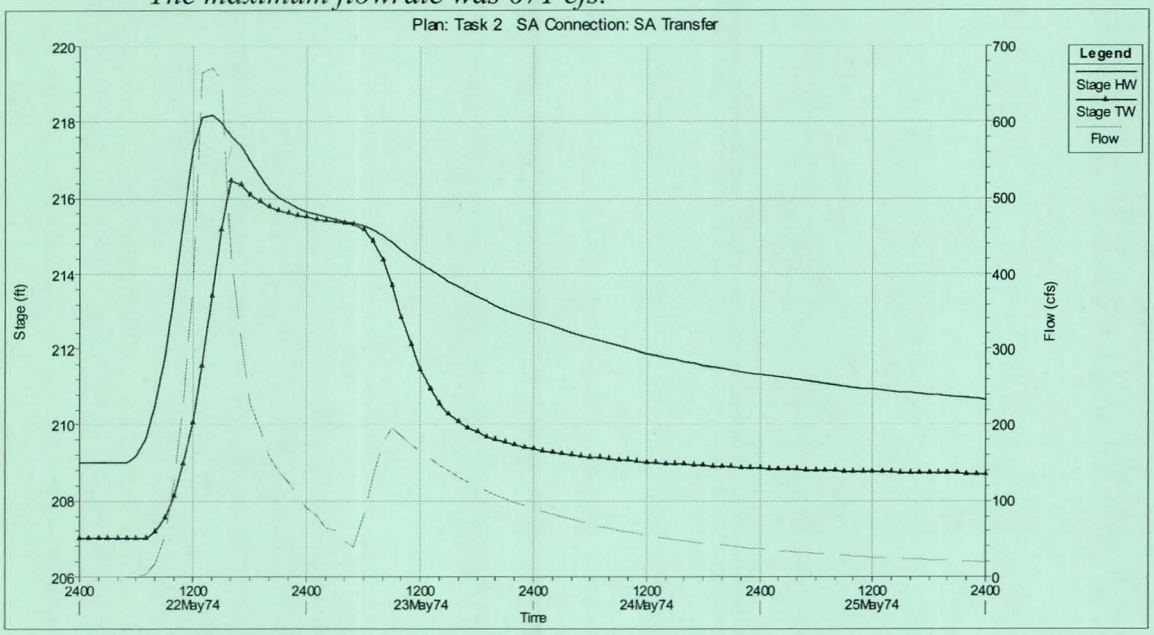


Figure 13 Storage Area Connection

The maximum flow over the lateral weir can be found by examining the ***Stage and Flow Hydrograph*** *plot of the lateral weir located at River Station 5.9, as shown in Figure 14. This figure can display the flow and stage information for headwater upstream and downstream of the lateral weir. Additionally, the figure can display flow and stage information for tailwater of the lateral weir (flow entering or leaving the storage area). To make the figure easier to read, only the flow information is included. As seen on the figure, the maximum flowrate over the weir (flow leaving) is 2,188 cfs.*

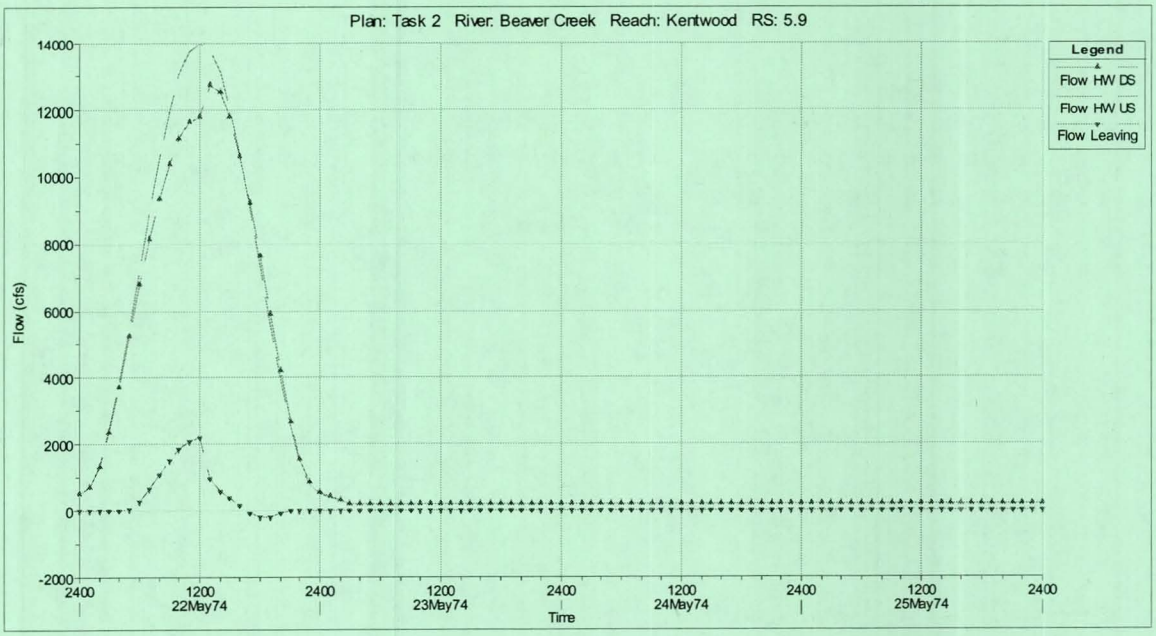


Figure 14 Lateral Structure at River Station 5.9

3. How did the stage and flow hydrographs change from Task 1 to Task 2? Did any numerical oscillations occur in the stage and flow hydrographs? If so, how can these oscillations be smoothed?

Figure 15 displays the downstream storage area before and after the control gates were added.

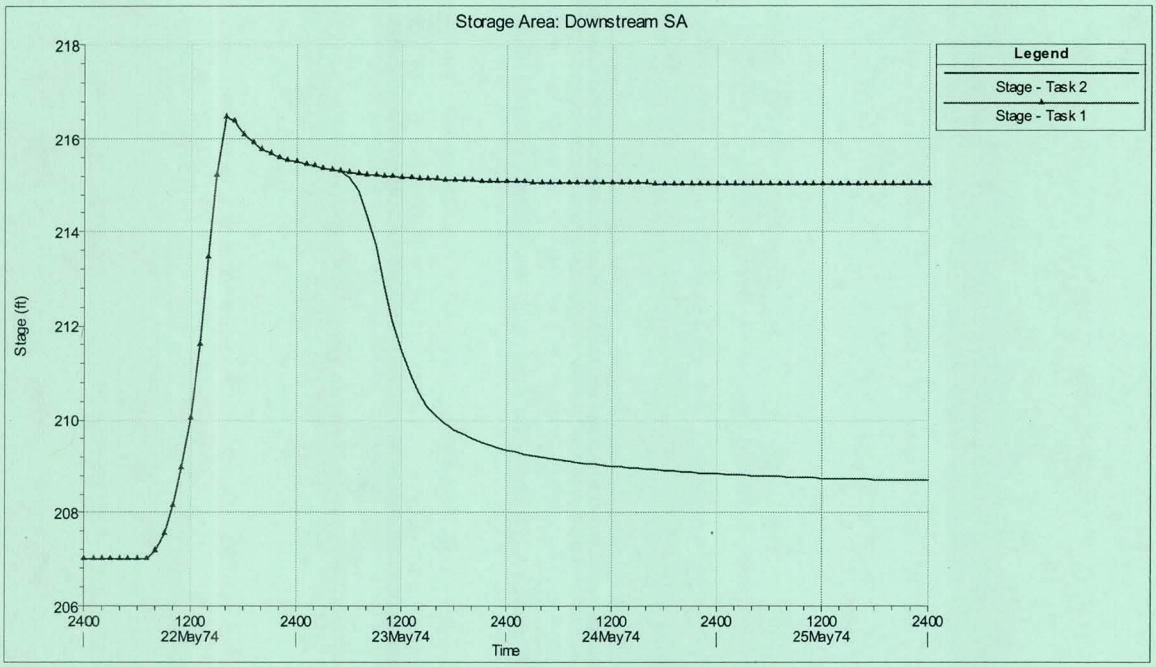


Figure 15 Downstream Storage Area With and Without Gates

Numerical oscillations can be observed, particularly in the downstream storage area hydrograph when the water surface elevations of the two storage areas approach each other (see circle on Figure 12). Oscillations become more prevalent and pronounced at coarse time steps (e.g.>30 seconds). Some oscillations can be smoothed by using a smaller time step, lowering the ***Storage Area Elevation Tolerance*** *and raising the* ***Weir Flow Submergence Decay Exponent****. These tolerance factors are found in the* ***Unsteady Flow Analysis Window*** *under the* ***Options*** *menu, under* ***Calculation Options and Tolerances…*** *For further discussion on these calculation options please read pages 8-23 and 8-24 of the RAS 3.1 User's Manual.*

4. What is another alternative, besides off-stream storage areas, to keep the bridge from being overtopped?

The bridge could be designed with multiple openings besides the main channel to pass larger flows. Multiple culverts could be placed along the bridge.

Trouble Shooting Workshop Solution

Task 1

As seen in Figure 1, the water surface profile for Beaver Creek is unstable, and eventually the errors become so severe that the program crashes.

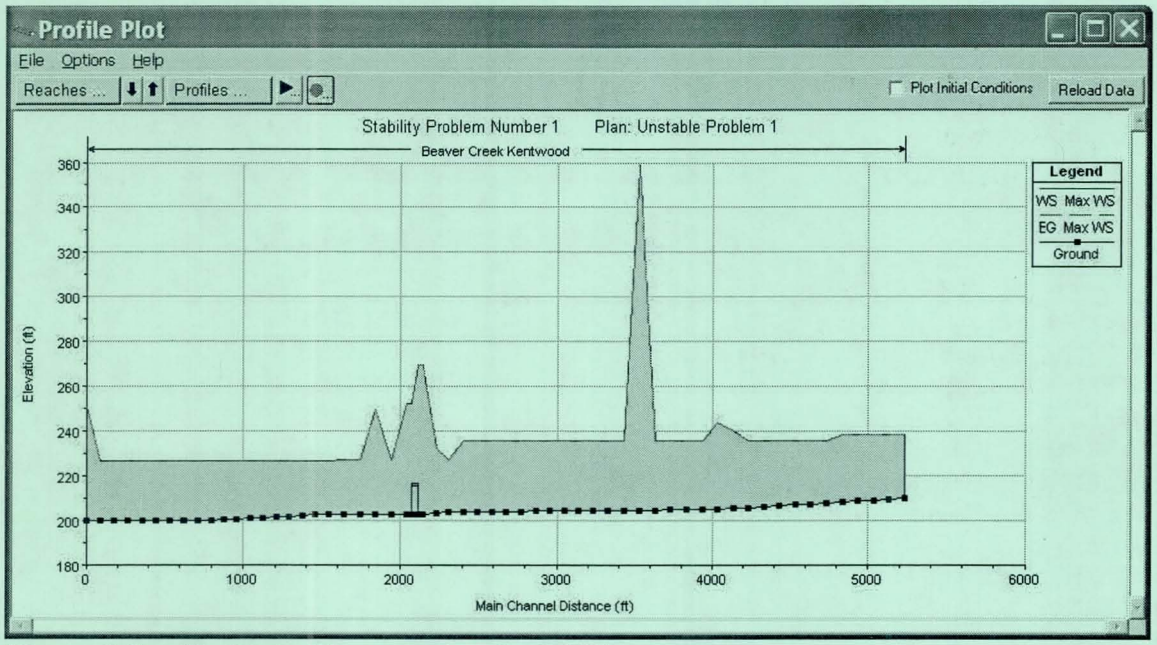


Figure 1 Profile Plot of Progressing Instability

Questions

At what time step and at what location did the program begin to go unstable?

The computational messages on the HEC-RAS Hydraulic Computations window showed some minor instability at the very first time step. However, there was no significant instability until 0300 on 10FEB1999, at which time the computational error was up to 0.687 feet. From there things just got worse, and the errors grew very large. The location of the largest errors was shown as cross section 5.0.

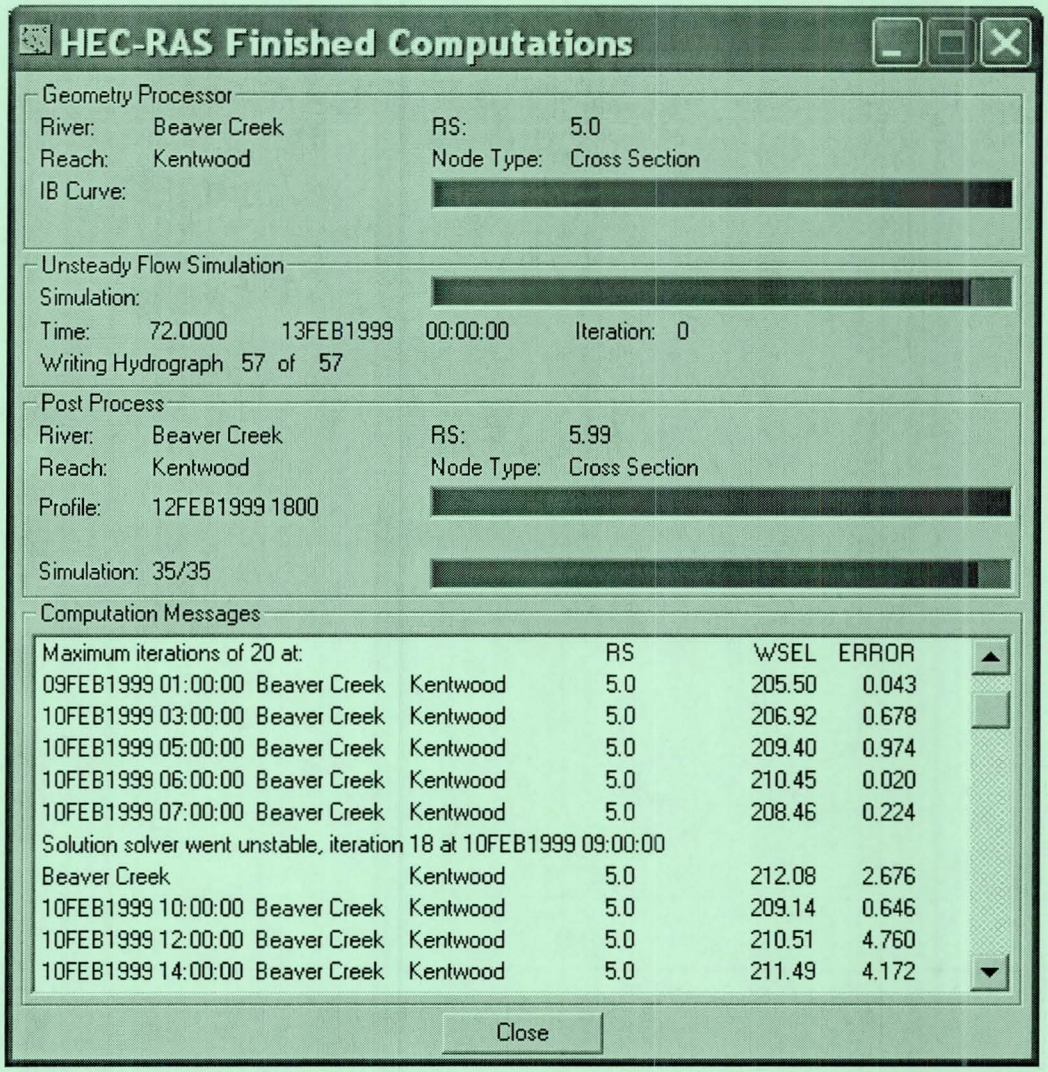


Figure 2 Detailed Output

Since the instability is occurring near the downstream end of the model, the user should examine the downstream boundary condition first. This particular data set has a rating curve at the downstream boundary. Figure 3 displays a plot of the rating curve. The plot shows a smooth continuously increasing rating curve. Most likely, the rating curve is not creating the instability.

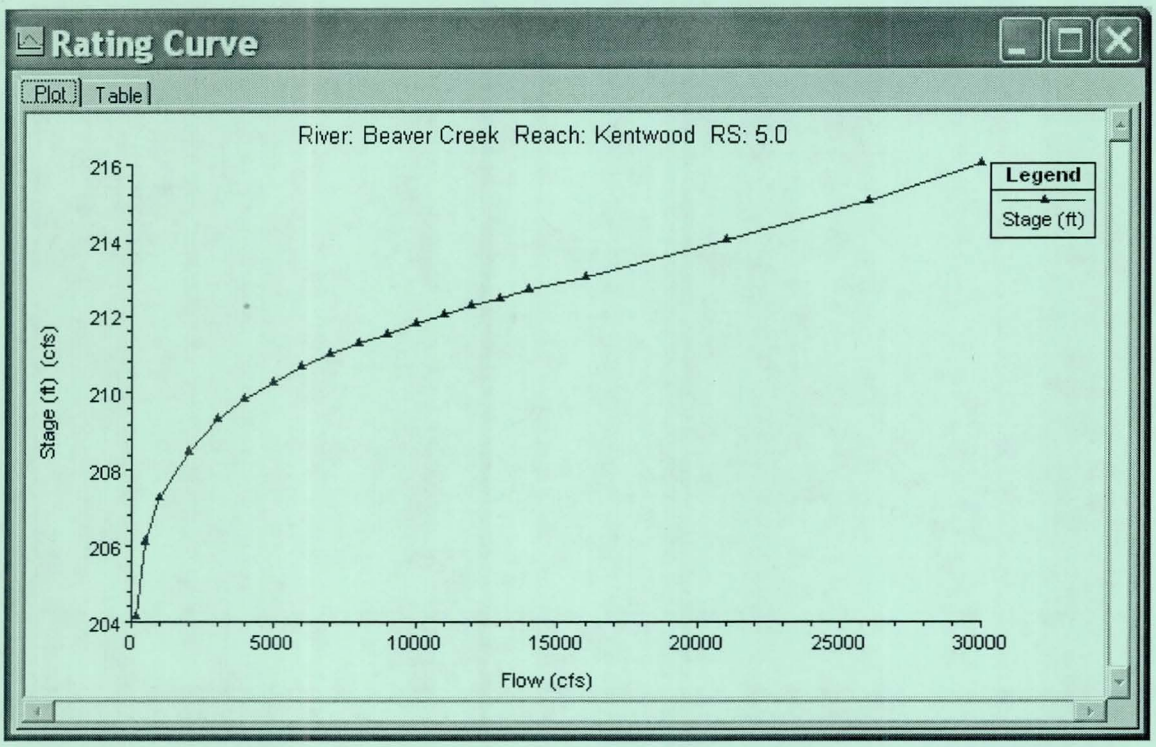


Figure 3 Downstream Boundary

What caused the instability?

Upon further review of the data, no problems were found with the geometry or boundary conditions. In the **Unsteady Flow Analysis Window**, the computation interval is set at one hour. This is an extremely large time interval for the computations. The upstream boundary condition for this simulation is a rapidly rising flow hydrograph. The one hour interval cannot capture the rapid change in flow with time and causes the instability.

If you were able to fix the instability, how was this achieved?

The computation interval was lowered to five minutes and the simulation was run. As seen in Figure 4, the change produced a stable simulation.

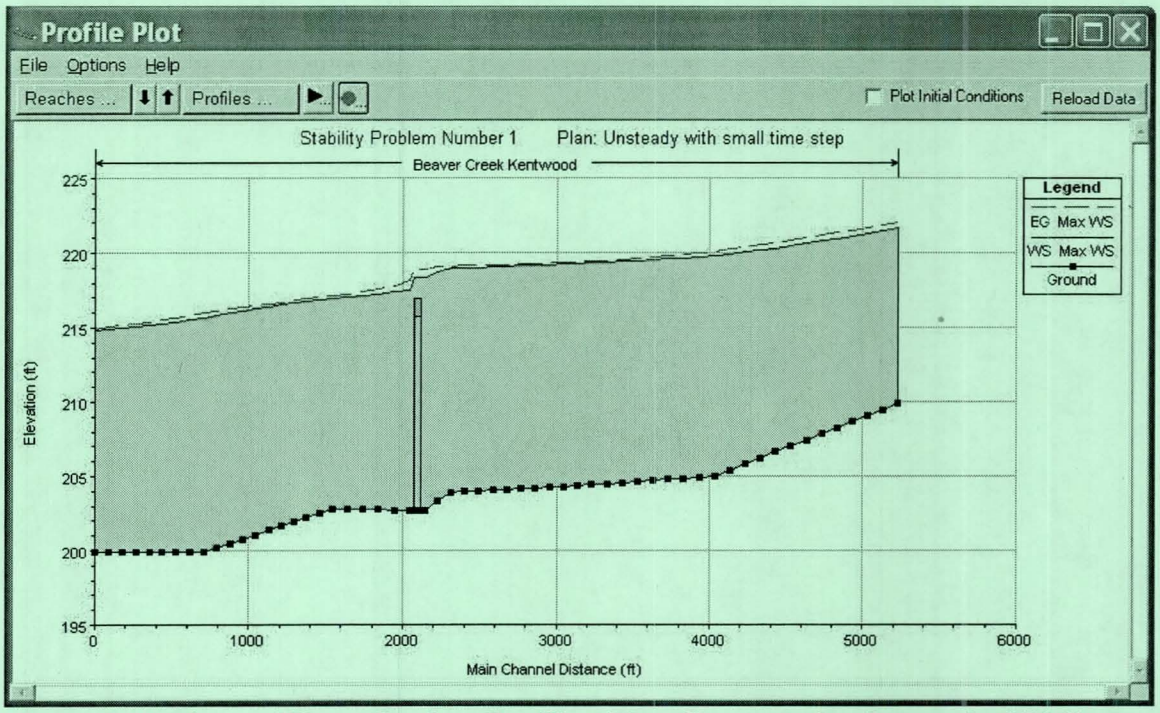


Figure 4 Maximum Water Surface Profile with a Computation Interval of 5 Minutes

Task 2

The simulation was run and the **Profile Plot** was examined. The animation tool was used to progress through the time steps. The animation shows that the program went unstable at the time step on 10FEB1999 at 2200. This is shortly after the gates were opened on time 2000. Figure 5 shows the profile just as the model starts to go unstable.

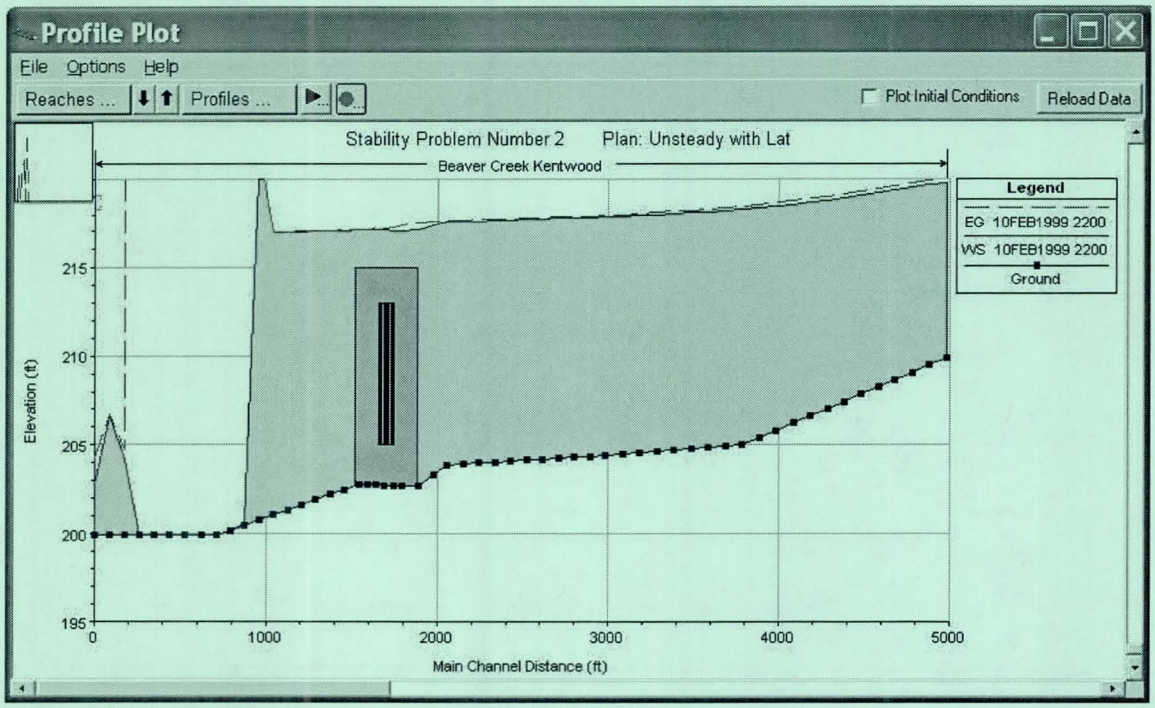


Figure 5. Instability Occurring on the Profile Plot

Questions

At what time step and at what location did the program begin to go unstable?

Reviewing the output shows that the model begins to go unstable just after the gates are opened. The location of the instability is at multiple locations downstream of the location of the lateral structure with the gates. After analyzing the output, it was noticed that the flow going through the gates is highly unstable from one time step to the next. It is difficult to tell at exactly what time step the program became unstable, but there is definitely a problem with the gates.

What caused the instability?

Figure 6 displays the detailed output just after the gates were opened. As seen on the figure, the flowrate through the gates went from 0.0 cfs to 5311 cfs in one time step (5 min.). This is a large change in flow leaving the reach between time steps. The rapid change in flow is most likely the cause of the instability in the solution.

Stability2.bco - Notepad

File Edit Format View Help

```
RIVER            STATION      QTOTAL      QWEIR     QGATE     QCULVERT  SUBMERGENCE      Q PER CROSS SECTION
Beaver Creek     5.4           3.40       3.40      0.00       0.00     0.000          3.40        0.00        0.00        0.00
0.00        0.00          0.00
 Gate #1      GO=  0.00   Q=     0.00

Hydrographs written at     19.92hours.

   ***   Start of New Time Step   ***

                         Solving for Time Window =    19.91667    to     20.00000 Hours

Iter  River           Station    Elev       DZ      Storage            Zsa        DZsa    River           Station          Q        DQ
  0 Beaver Creek      5.37       214.28   0.68010   STo #1            205.08   0.07990 Beaver Creek      5.33          8421      4499
  1 Beaver Creek      5.37       214.14   0.19479   STo #1            205.08  -0.00552 Beaver Creek      5.33          8715       420
  2 Beaver Creek      5.0        212.11   0.08005   STo #1            205.07  -0.00230 Beaver Creek      5.0          12426       261
  3 Beaver Creek      5.0        212.07   0.05737   STo #1            205.07  -0.00095 Beaver Creek      5.0          12553       181
  4 Beaver Creek      5.0        212.04   0.04234   STo #1            205.07  -0.00039 Beaver Creek      5.0          12638       123
  5 Beaver Creek      5.0        212.02   0.03063   STo #1            205.07  -0.00017 Beaver Creek      5.0          12701        90
  6 Beaver Creek      5.0        212.00   0.02225   STo #1            205.07  -0.00007 Beaver Creek      5.0          12747        65
  7 Beaver Creek      5.0        211.99   0.01644   STo #1            205.07  -0.00003 Beaver Creek      5.0          12779        46
  8 Beaver Creek      5.0        211.98   0.01193   STo #1            205.07  -0.00001 Beaver Creek      5.0          12803        34
  9 Beaver Creek      5.0        211.98   0.00867   STo #1            205.07  -0.00001 Beaver Creek      5.0          12821        25

     COMPUTED STAGES AND DISCHARGES AT T =     20.0000 HOURS -  2/10/1999 AT TIME  20:00:00

              Beaver Creek      Kentwood

Riv. Station     Z         Q        V        Riv. Station     Z         Q        V        Riv. Station     Z         Q        V
    5.99    219.40   13096.   2.38            5.274    214.14    9408.   2.12
    5.93    218.83   13088.   2.36              5.0    211.98   12821.   1.81
    5.41    214.54   14356.   3.46
    5.31    214.24    9040.   2.04


                    Storage Areas

NAME            ELEVATION      NET Q
STo #1            205.07     5320.66

                    Lateral Weirs

RIVER            STATION      QTOTAL      QWEIR     QGATE     QCULVERT  SUBMERGENCE      Q PER CROSS SECTION
Beaver Creek     5.4          5320.66      0.00    5320.66      0.00     0.000          0.00        0.00     3166.74     2153.92
0.00        0.00          0.00
 Gate #1      GO=  8.00   Q=  5320.66
```

Figure 6 Detailed Output

Under the **Unsteady Flow Data Window** the **Time Series of Gate Openings** was examined. As can be seen on Figure 7, the gates are rapidly opened (5 minute interval). The rapid opening of the gates is the cause of the rapid increase in flow, which in turn caused the instability.

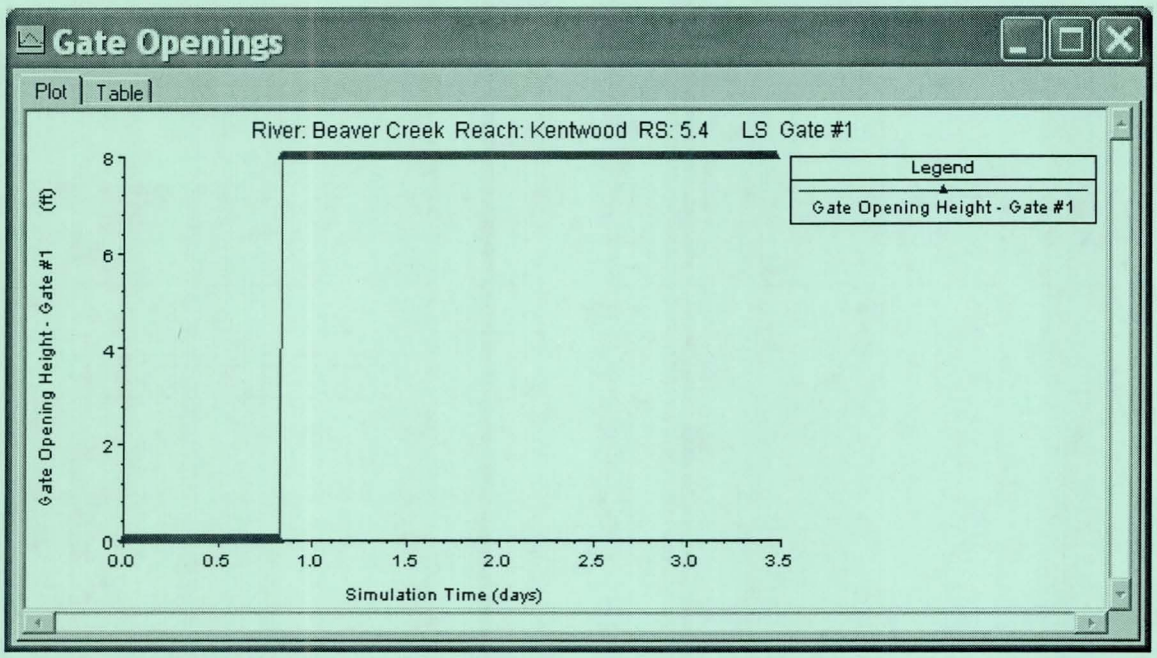


Figure 7 Plot of Gate Opening

Figure 8 displays the **Stage and Flow Hydrograph** of the storage area for the unstable simulation. The sudden spike in flow that occurred when the gates were opened caused the instability. Shortly after that the flow went negative to a very high amount.

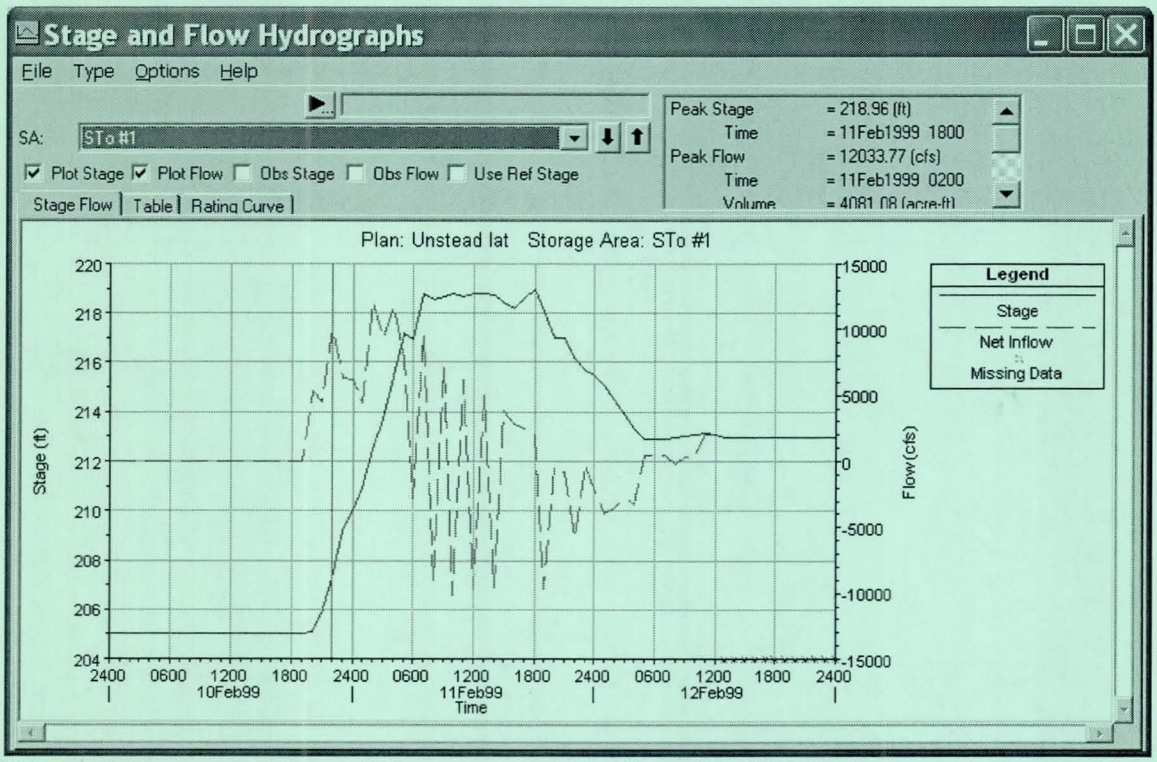


Figure 8 Stage and Flow of Storage Area for Unstable Simulation

If you were able to fix the instability, how was this achieved?

The time series of gate openings was adjusted in the **Unsteady Flow Data Window**. As seen in Figure 9, the gates were opened over a longer period of time.

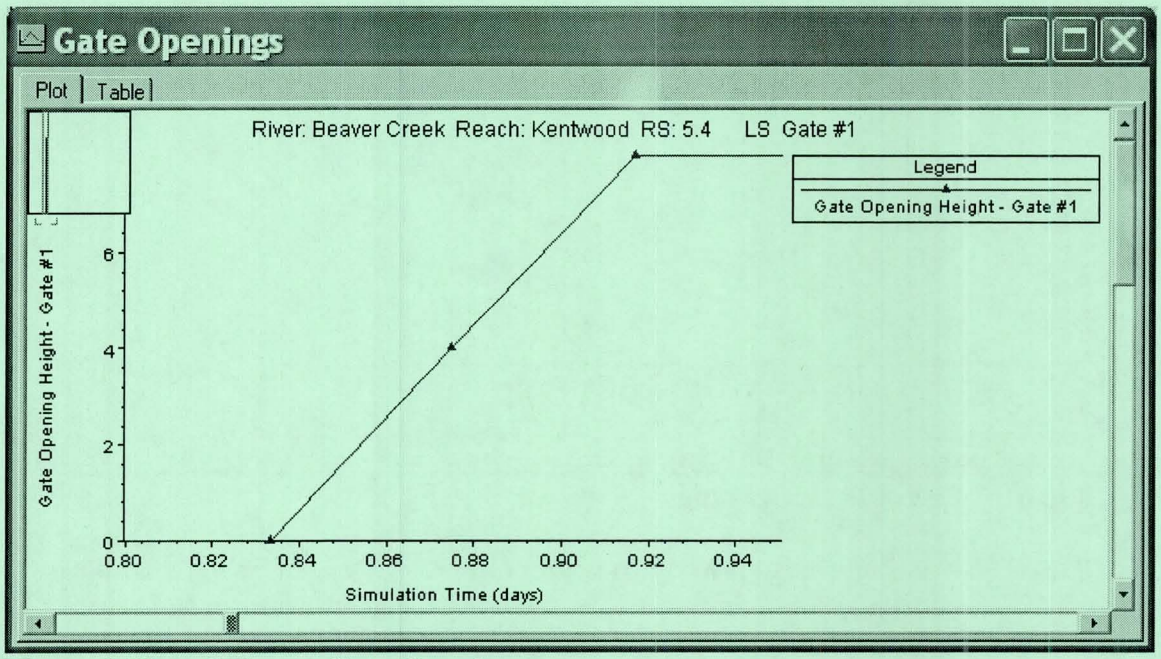


Figure 9 Plot of Slower Gate Opening

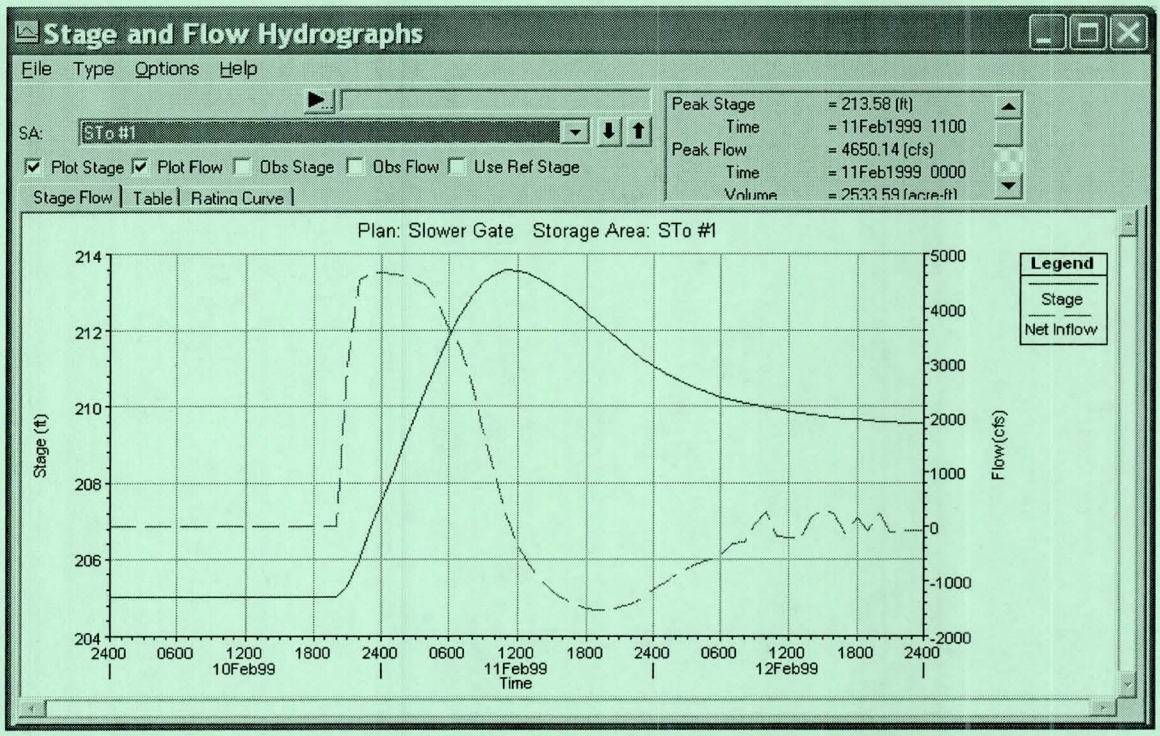


Figure 10 Stage and Flow of Storage Area for Stable Simulation

Figure 10 displays the **Stage and Flow Hydrograph** of the storage area for the stable simulation. By opening the gates at a slower rate the rapid change in flow leaving the reach was decreased. This prevented instabilities in the simulation. Another solution would have been to reduce the time step. However, you were asked not to do that for this problem.

Task 3

Examining the animation of the **Profile Plot** and reviewing the computational messages on the "HEC-RAS Hydraulic Computations" window, showed that the program crashed near the beginning of the simulation. The detailed output was examined to determine exactly when the instability occurred.

Questions

At what time step and at what location did the program begin to go unstable?

The detailed output was examined under the **Computation Log File**, found in the **Options** menu of the **Unsteady Flow Analysis Window**. A search for the word "Warning" was done to locate the initial point of instability. As seen on Figure 11, the "Warning" first occurs at the cross sections near the downstream end of the model at time 1.333 hours.

Stability3.bco - Notepad

File Edit Format View Help

```
RIVER          REACH          STATION     QTOTAL      HEAD WATER     TAIL WATER     HEAD DERIVATIVE
Beaver Creek   Kentwood       5.4         1362.81        210.76         210.70           0.001582

Hydrographs written at      1.33hours.

   ***   Start of New Time Step   ***

                        Solving for Time Window =     1.33333    to      1.41667 Hours

Iter  River          Station    Elev       DZ        River          Station        Q        DQ
  0 Beaver Creek     5.0        199.91  13.33740  Beaver Creek     5.0         2867      1713
  1 Beaver Creek     5.048     1892.24  ********  Beaver Creek     5.065      25352     33672
  2 Beaver Creek     5.048      705.77  ********  Beaver Creek     5.065       7546     25437
  3 Beaver Creek     5.048      350.01  ********  Beaver Creek     5.065       2207      7627
  4 Beaver Creek     5.048      243.08  ********  Beaver Creek     5.31         -44      2619
  5 Beaver Creek     5.048      211.14  45.63230  Beaver Creek     5.33        3392      3915
  6 Beaver Creek     5.13       219.07  27.29804  Beaver Creek     5.274        454      1718
  7 Beaver Creek     5.048      420.98  ********  Beaver Creek     5.032       1198      4672
  8 Beaver Creek     5.048      264.34  ********  Beaver Creek     5.032       3848      3786
  9 Beaver Creek     5.048      217.37  67.09959  Beaver Creek     5.032       4677      1185
 10 Beaver Creek     5.032      200.63  16.06414  Beaver Creek     5.048        509      2667
 11 Beaver Creek     5.048      199.91  15.90423  Beaver Creek     5.032       4356      2023
 12 Beaver Creek     5.032      199.91  ********  Beaver Creek     5.016       2236      3876
 13 Beaver Creek     5.081      204.19  21.73150  Beaver Creek     5.065       3614      4493
 14 Beaver Creek     5.081      199.91  ********  Beaver Creek     5.065       4470      1224
 15 Beaver Creek     5.194      201.07  32.25425  Beaver Creek     5.178       3517      4719
 16 Beaver Creek     5.162      200.49   6.21144  Beaver Creek     5.21        3757      3865
 17 Beaver Creek     5.33       202.77  ********  Beaver Creek     5.31        4483      1494
 18 Beaver Creek     5.35       205.28   6.68886  Beaver Creek     5.35        3345      1922
 19 Beaver Creek     5.37       202.73  ********  Beaver Creek     5.37        1711      1892
 20 Beaver Creek     5.41       202.71  ********  Beaver Creek     5.41        4420      3185

 WARNING: USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR.  MINIMUM ERROR OCCURED DURING ITERATION 16.

     COMPUTED STAGES AND DISCHARGES AT T =      1.4167 HOURS -  2/10/1999 AT TIME  01:25:00

            Beaver Creek      Kentwood
```

Figure 11 Detailed Output

What caused the instability?

Since the instability is occurring at the cross sections near the downstream end of the model, the user should examine the downstream boundary condition. This particular data set has a rating curve at the downstream boundary. Figure 12 displays a plot of the rating curve.

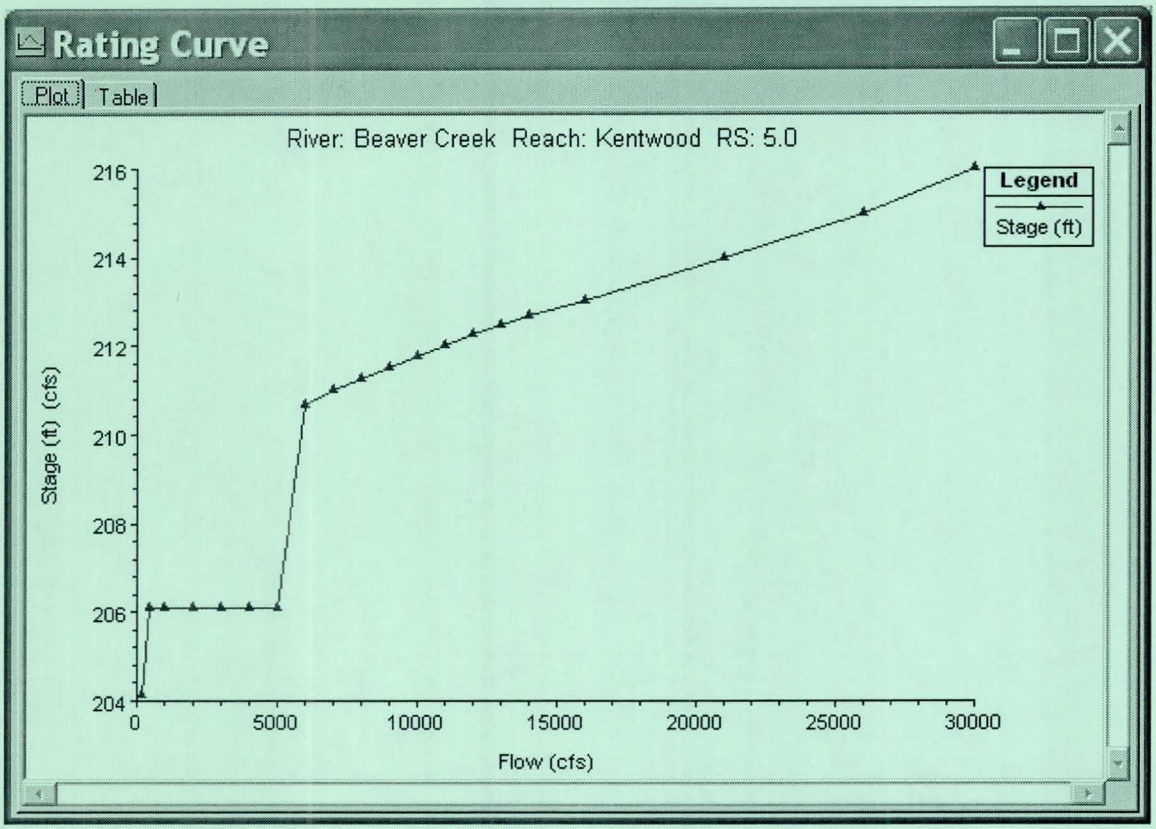


Figure 12 Downstream Boundary Condition

As seen in the plot, there is an area of discontinuity in the rating curve. This discontinuity causes a large difference in the water surface slopes at lower flows. The program goes unstable because of large differences in the derivatives caused by the large differences in the water surface slopes.

If you were able to fix the instability, how was this achieved?

To correct the problem the rating curve was adjusted to produce a smooth, continuous increase in flow versus stage. Figure 13 displays the adjusted rating curve that produced a stable simulation.

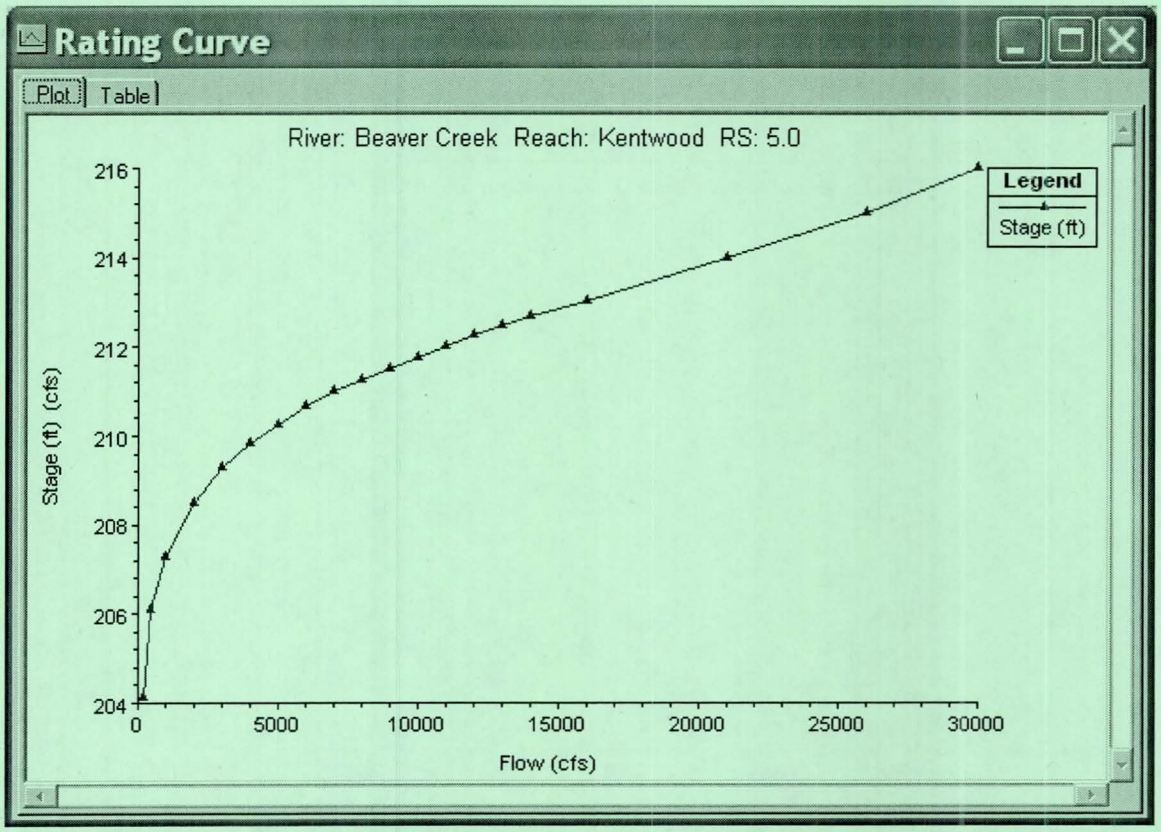


Figure 13 Adjusted Rating Curve Producing Stability

Task 4

The **Profile Plot** for this example shows a dramatic change in water surface slope occurring at River Station 7.0, which is at the upstream end of a significant drop in the channel.

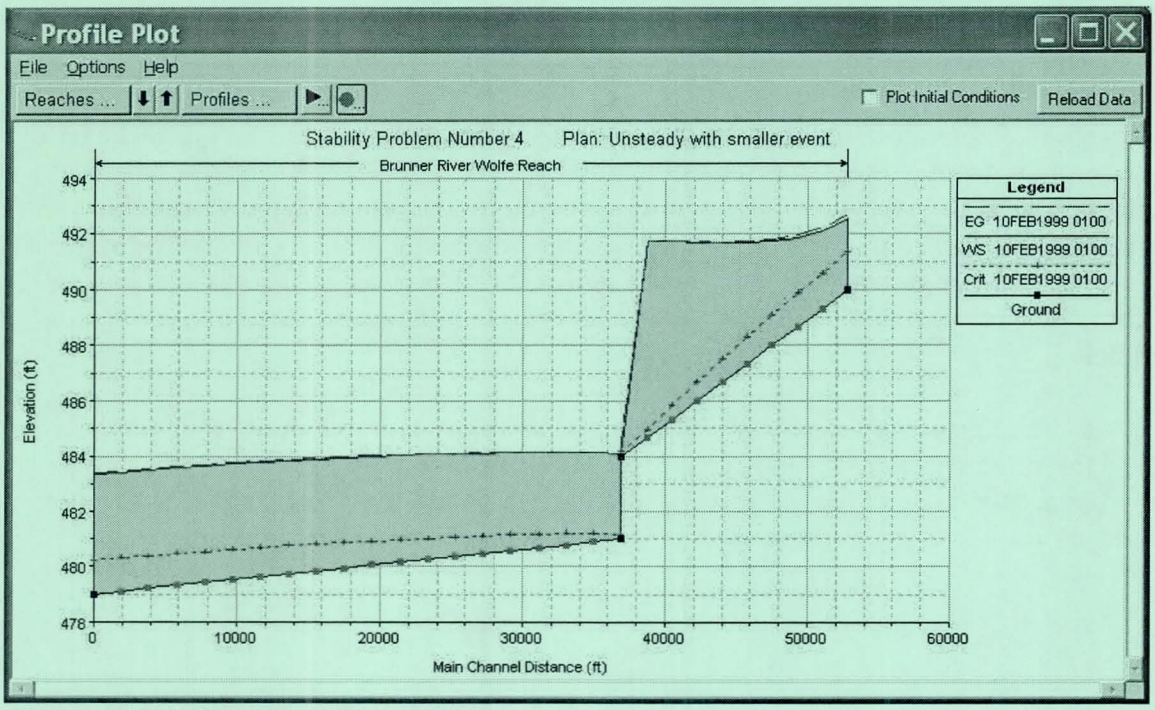


Figure 14 Profile Plot of Instability

Questions

At what time step and at what location did the program begin to go unstable?

Examining the animation of the **Profile Plot** shows that the program went unstable where the slope of the reach dramatically changes, River Station 7.0. Figure 14 displays the **Profile Plot** illustrating the instability. The instability occurs at the beginning of the simulation and progressively became larger.

What caused the instability?

As seen on Figure 14, the water surface profile goes to critical depth where the slope of the reach changes. When HEC-RAS is run in its default mode (non-mixed flow regime), it is limited to subcritical flow for unsteady flow simulations.

If you were able to fix the instability, how was this achieved?

There are three possible solutions for this problem. The first involves examining the base flow from the input hydrograph. The simulation with the instability had a base flow and initial condition of 700 cfs. This low flow allows the water surface to pass through critical depth. One solution is to increase the base flow and initial flow to a value that does not pass through critical depth. A value of 1000 cfs was set and the simulation was re-run. This adjustment to the hydrograph and initial conditions provided a stable simulation.

Second, the change in slope could be modeled as an inline weir. An inline weir was added just downstream of the change in slope at River Station 6.995. Figure 15 displays the cross section view of the inline weir. Figure 16 displays the dimensions of the inline weir.

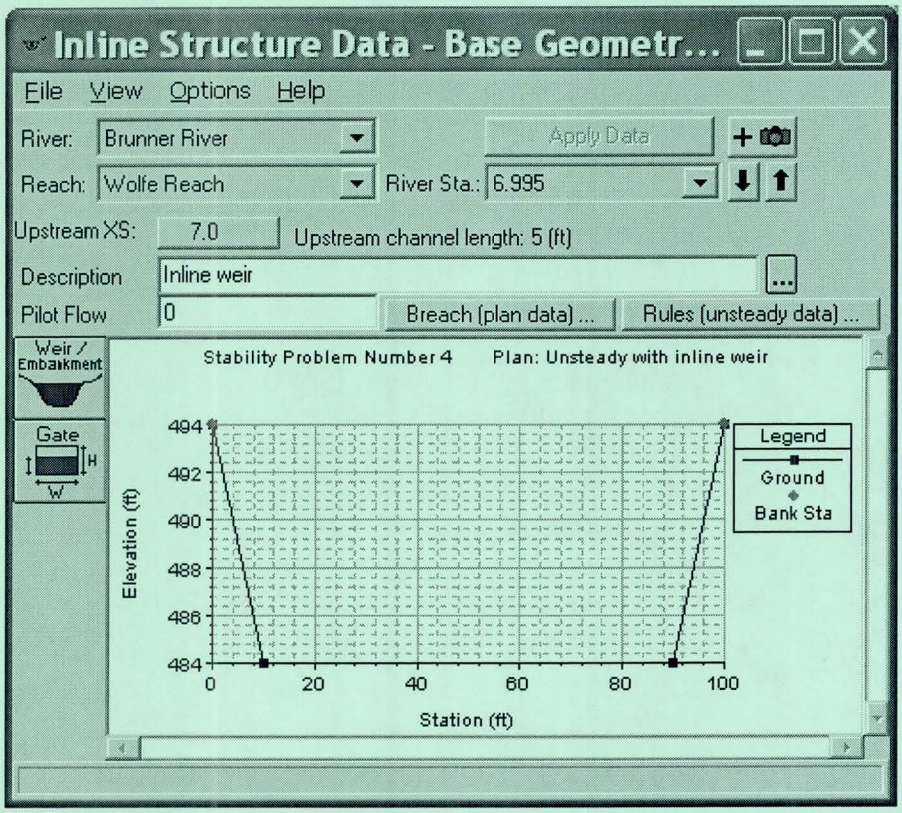


Figure 15 Plot of Inline Weir

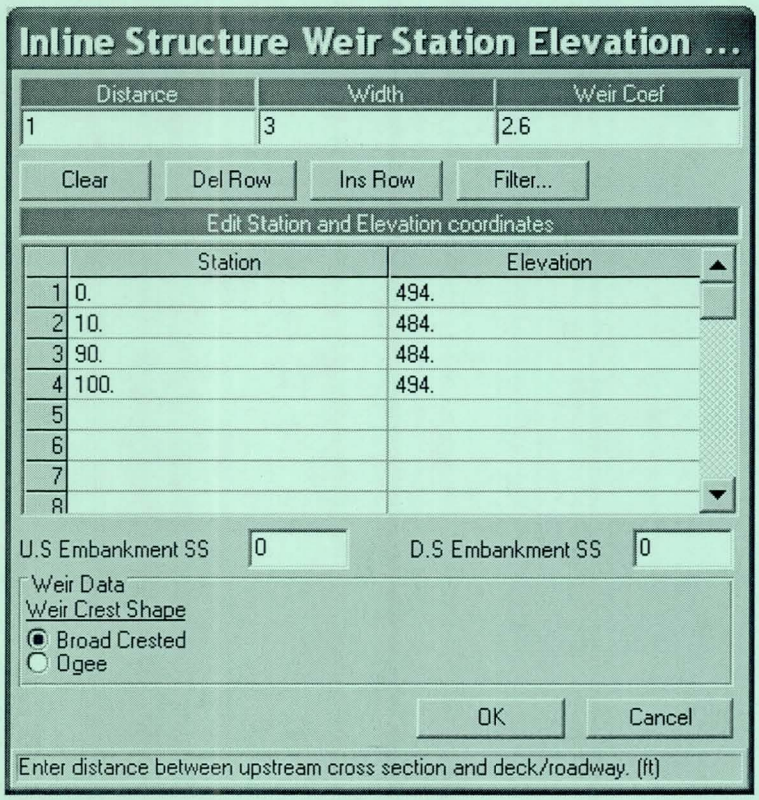


Figure 16 Dimensions of Inline Weir

Notice that the inline weir has the same dimensions as the cross section. Thus, the placement of the inline weir at this cross section does not impede the flow in any manner. The purpose of placing the inline weir at this point is to provide stability in the model. The solution of the continuity and momentum equations are causing the instability at the point where the slope dramatically changes. With the inline weir in place, RAS uses a weir flow equation instead of the unsteady flow equations for the solution over the drop. This solution allows the program to get a subcritical water surface both upstream and downstream of the drop. The flow transition is handled by the weir equation and is not modeled directly.

The final solution to this problem would be to turn on the mixed flow regime option. With this option turned on the program automatically reduces the magnitude of the inertial terms in the momentum equation when the Froude number starts to get close to 1.0. This feature allows the program to solve for the water surface passing through critical depth, and then going into a hydraulic jump.

All three of these methods were tried on this data set and they worked. The resulting profiles are shown in Figure 17. The lower profile is at the base flow of 700 cfs, and the upper profile is the maximum profile.

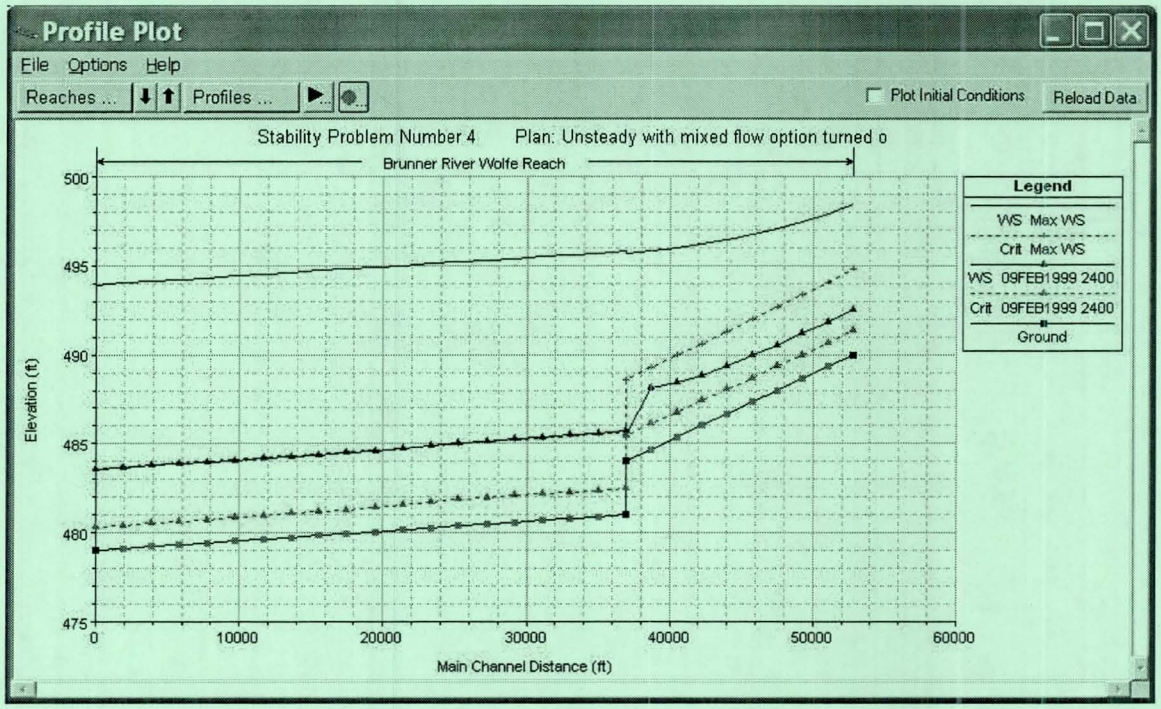


Figure 17. Resulting profiles from mixed flow regime run.

Dam and Levee Break Workshop Solution

Introduction

The purpose of this workshop was to help students learn how to use HEC-RAS to: enter and edit dam and levee breach data; perform the unsteady breach run; make adjustments for stability; and review and understand the pertinent dam and levee breach output.

Tasks

1. Save the Plan with a New Name

The first step was to save the plan with a new name (Figures 1, 2 and 3).

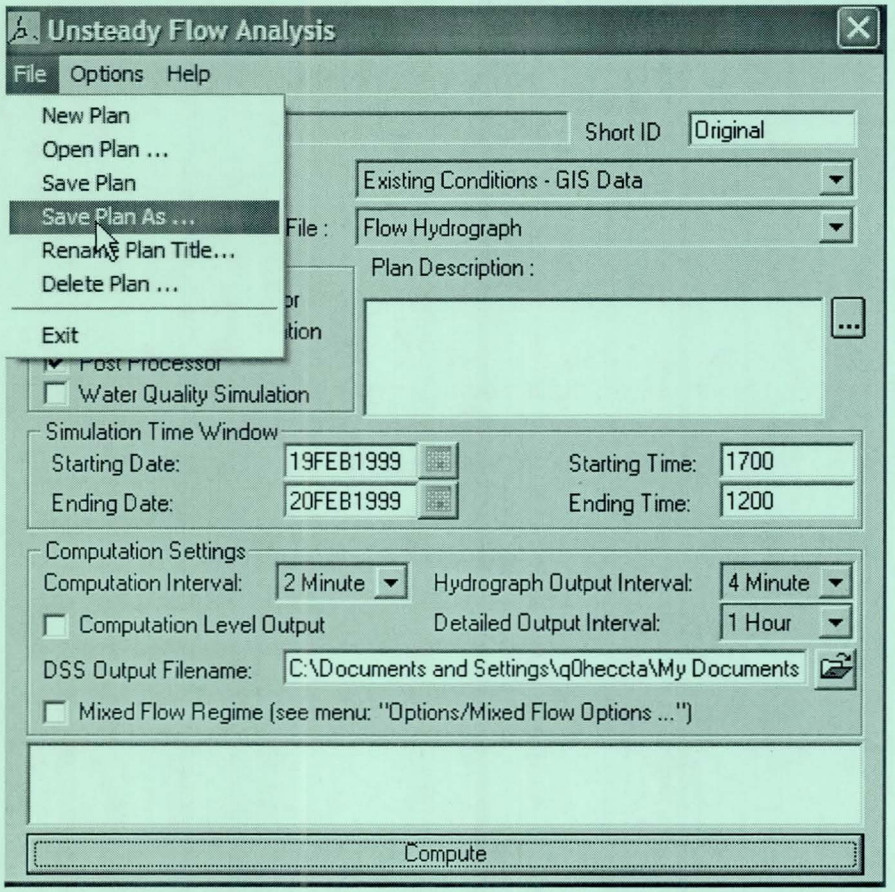


Figure 1. Unsteady flow analysis window with "Save Plan As..." option shown

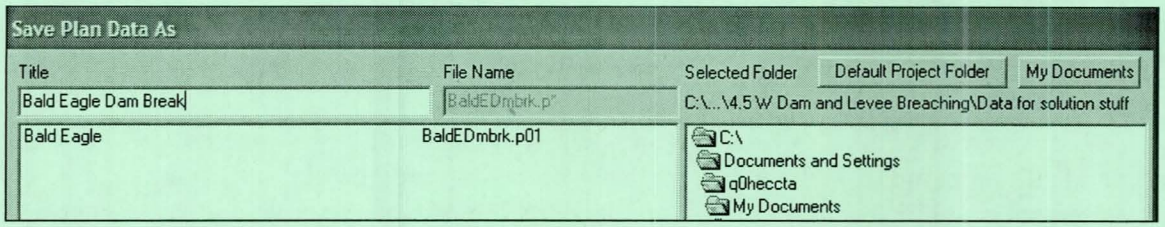


Figure 2. "Save Plan Data As" window showing new Title

Figure 3. Short Plan Identifier window

2. Enter the Dam Break Data

Once a new plan has been created, the next step was to enter the Dam Breach information. To get to the dam breach editor, the user could either click on the <Breach (plan data)> button on the Inline Weir editor, or from the unsteady flow analysis editor choose options, dam breach. Next, the information should have been entered as shown in figure 4.

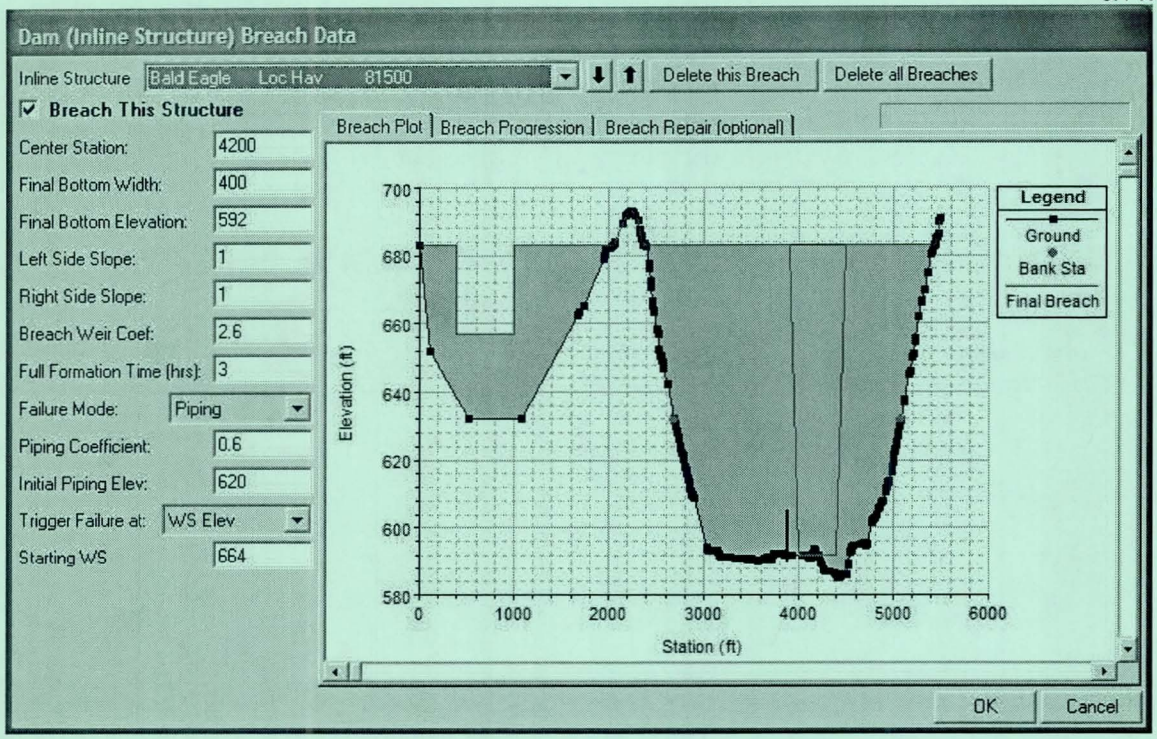


Figure 4. Dam Breach window showing breach data

3. Perform the Unsteady Flow Simulation and Review the Results

After the dam breach information was entered, the unsteady flow simulation was performed and various tables and graphs could be examined to show the results. Figure 5 shows the inline weir cross plot when the dam has a large piping breach.

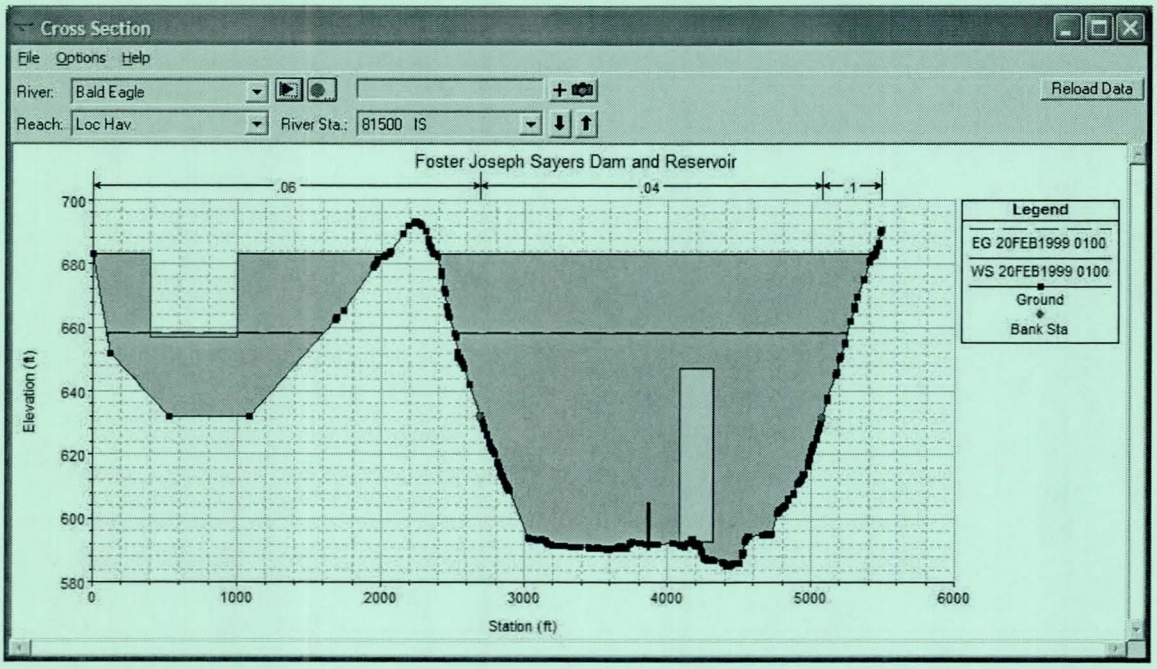


Figure 5. Cross section plot showing piping breach

4. & 5. Create a New Plan and Run the Dam Breach with a Ten Second Time Step

The next two tasks were to save the plan with a new name and perform the unsteady flow simulation with a ten second time step. Figure 6 shows the stage and flow hydrograph for the inline weir for both the original dam break and the ten second dam break. Figure 7 is a zoomed in plot showing a slight difference in peak flow between the two plans.

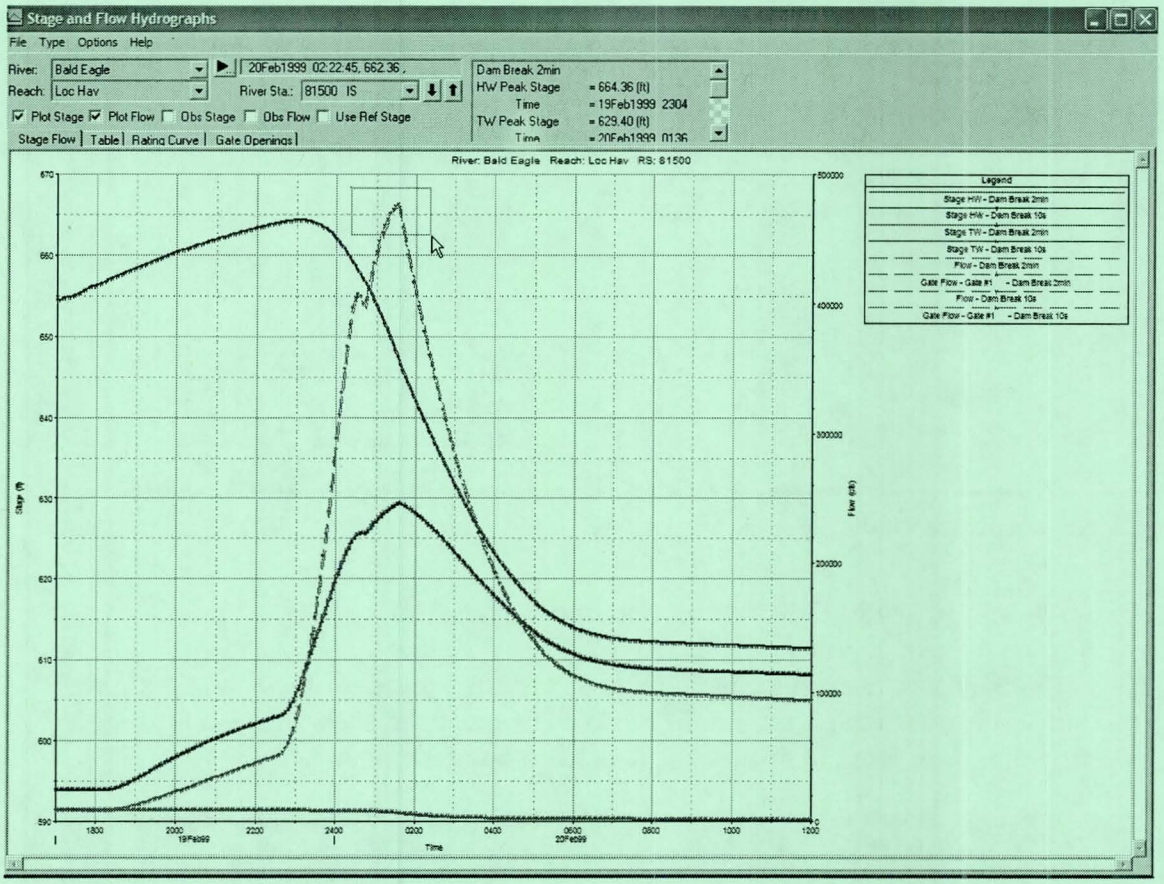


Figure 6. Stage and Flow Hydrograph for the Dam Break

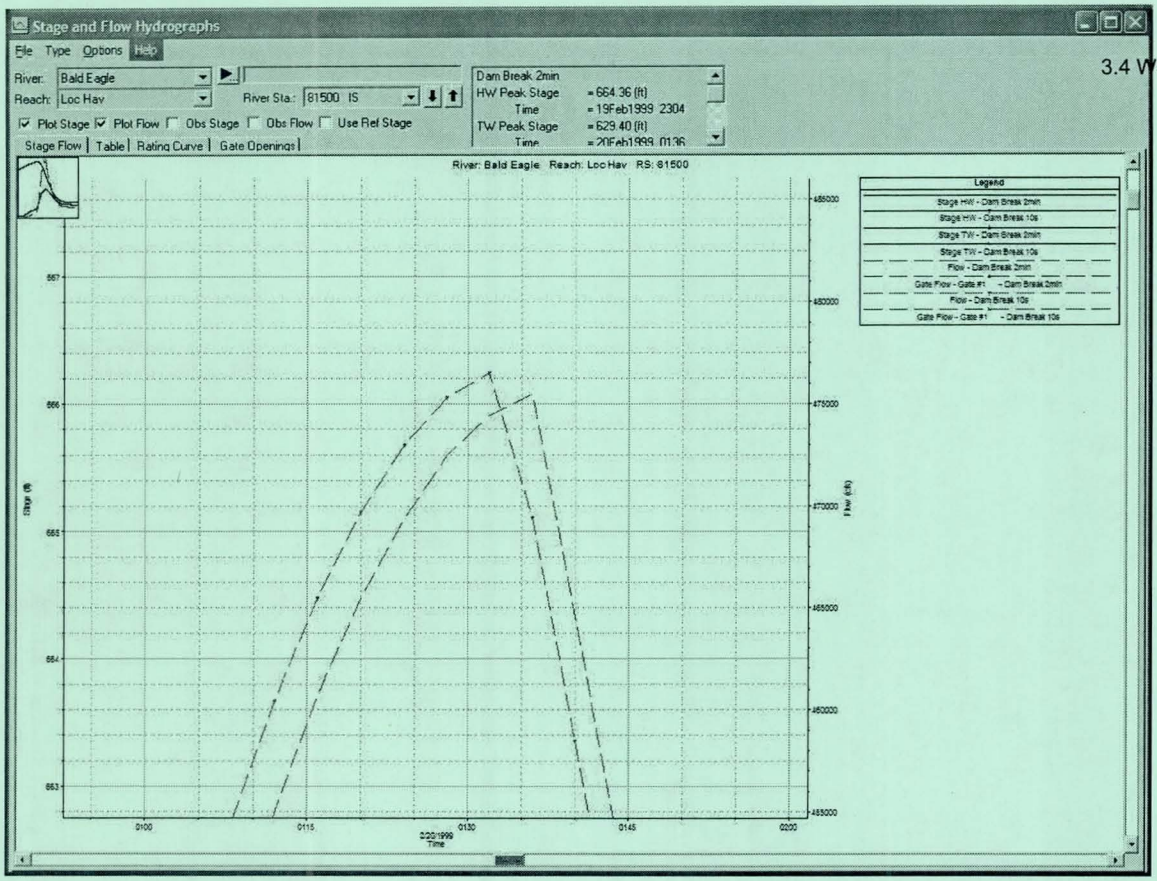


Figure 7. Zoomed in view of Stage and Flow Hydrograph showing peak flow from Dam Break

6. Add the Lateral Weir Levee Breach

The next step was to save the plan with a new name and add a levee breach. The levee represented by the lateral weir at river station 22900 was the first to be overtopped, so the breach was added at this location (Figure 8).

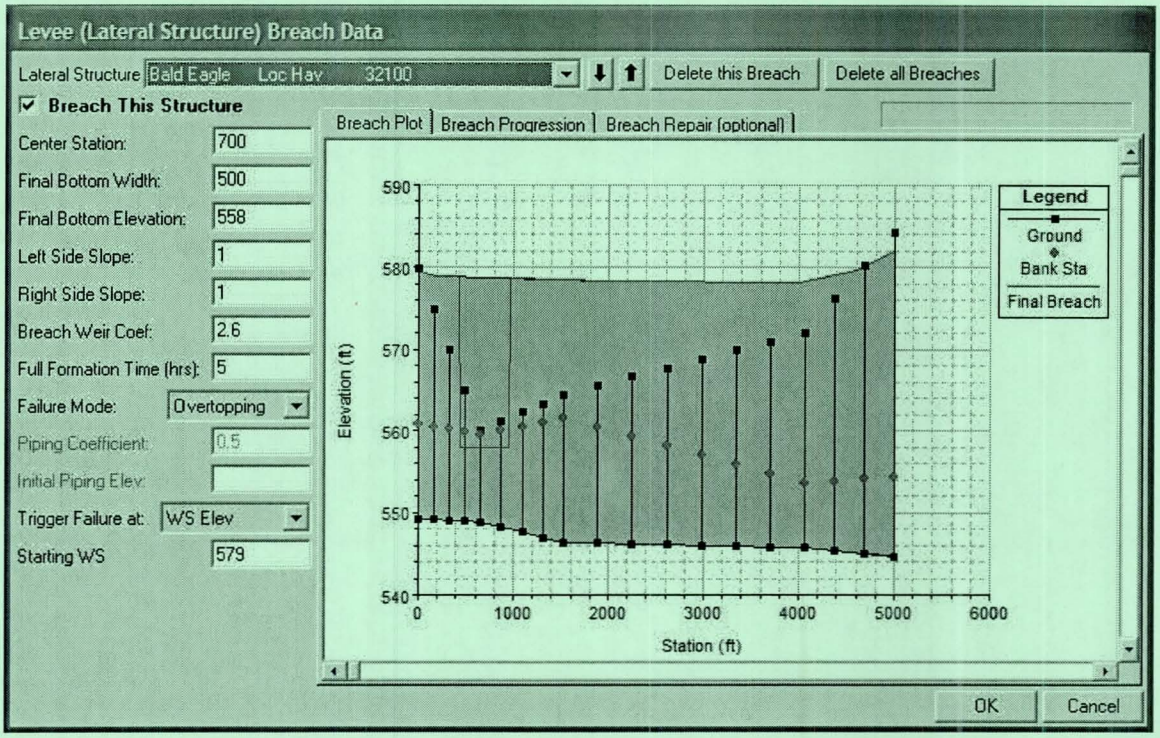


Figure 8. Lateral Breach window showing levee breach data

7. Review the Results

After performing the unsteady flow simulation, the results can be reviewed. Figure 9 shows a profile plot of the lateral weir with and without the levee breach. Figure 10 shows the stage and flow hydrograph for the storage area. The red line in figure 10 corresponds to the date on the animation control: 20Feb1999 0700.

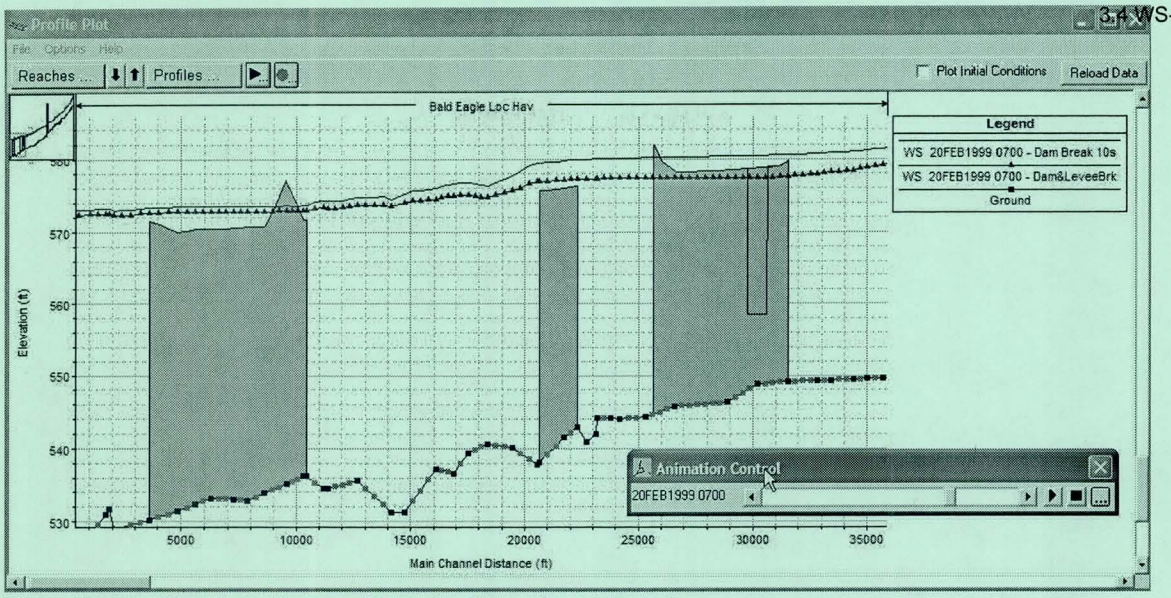


Figure 9. Profile plot at levee with and without levee breach

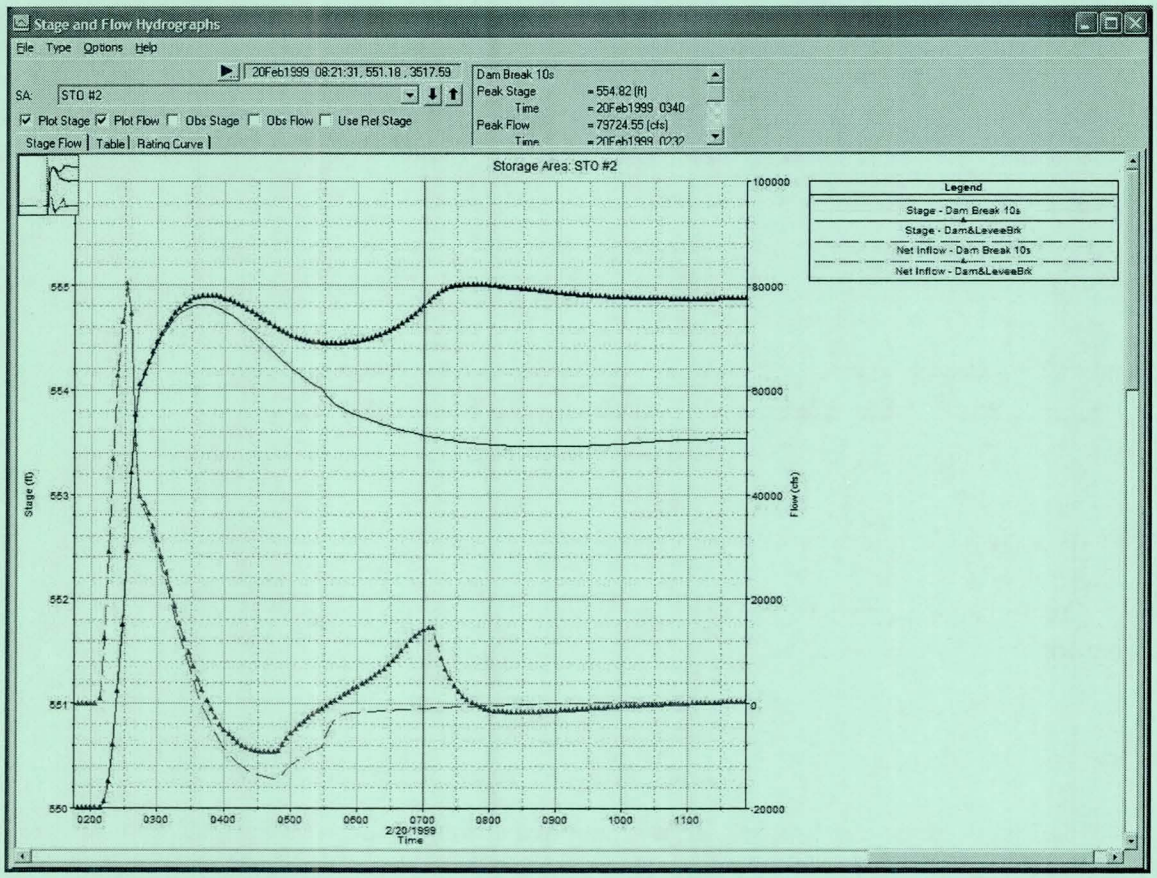


Figure 10. Stage and Flow Hydrograph at storage area with and without levee breach

8. Create a New Plan with a Longer Time Window

The next step was to create another plan, this time with a longer time window. The time increment, as shown in Figure 11, was increased to 1 minute. This was mainly done to reduce the time it took to finish the workshop.

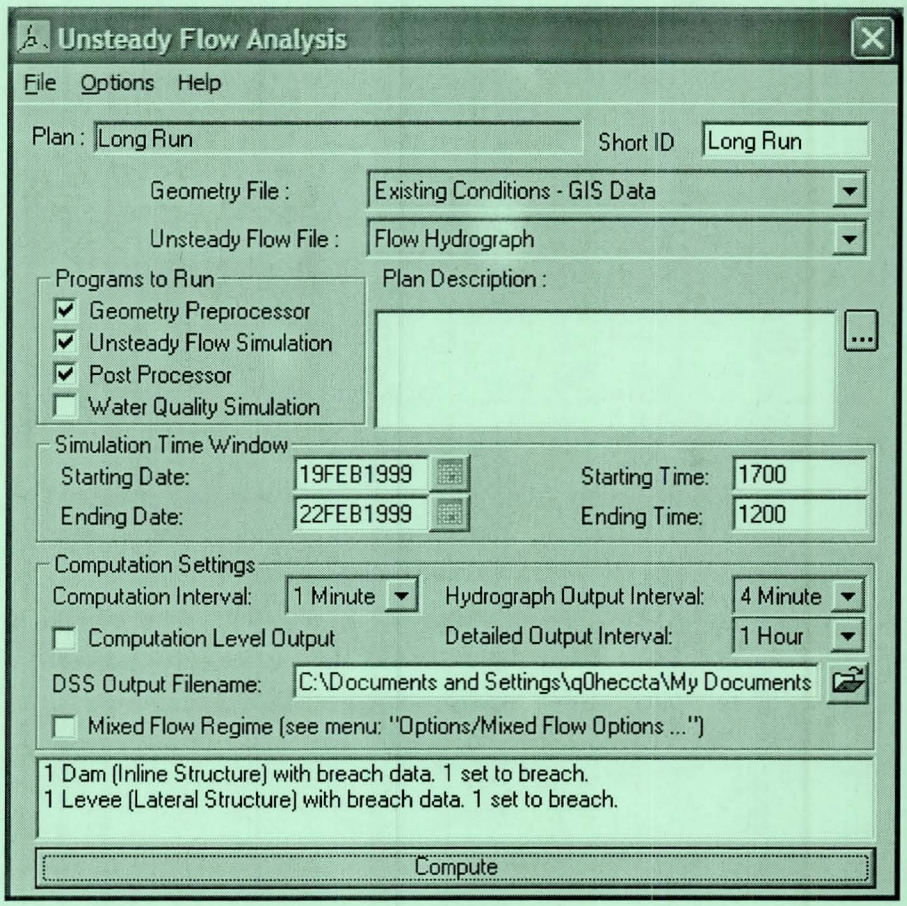


Figure 11. New Plan with Longer Time Window

9. Create a New Plan and Add the Levee Repair

Create yet another plan and the levee repair. Figure 12 shows the levee repair information.

10. Review the Results

Figure 13 shows the profile plot with the levee breach and with and without the levee repair. It also visually shows the amount the levee has been repaired. Figure 14 shows the stage at storage area #2 for three different plans. The stage line that ends at 20Feb at 1200 is the dam break without levee breach. This plan had a shorter time window than the other two plans. The top line is the stage with the levee breach. The middle line (the one the pointer is next to) is the levee breach with repair.

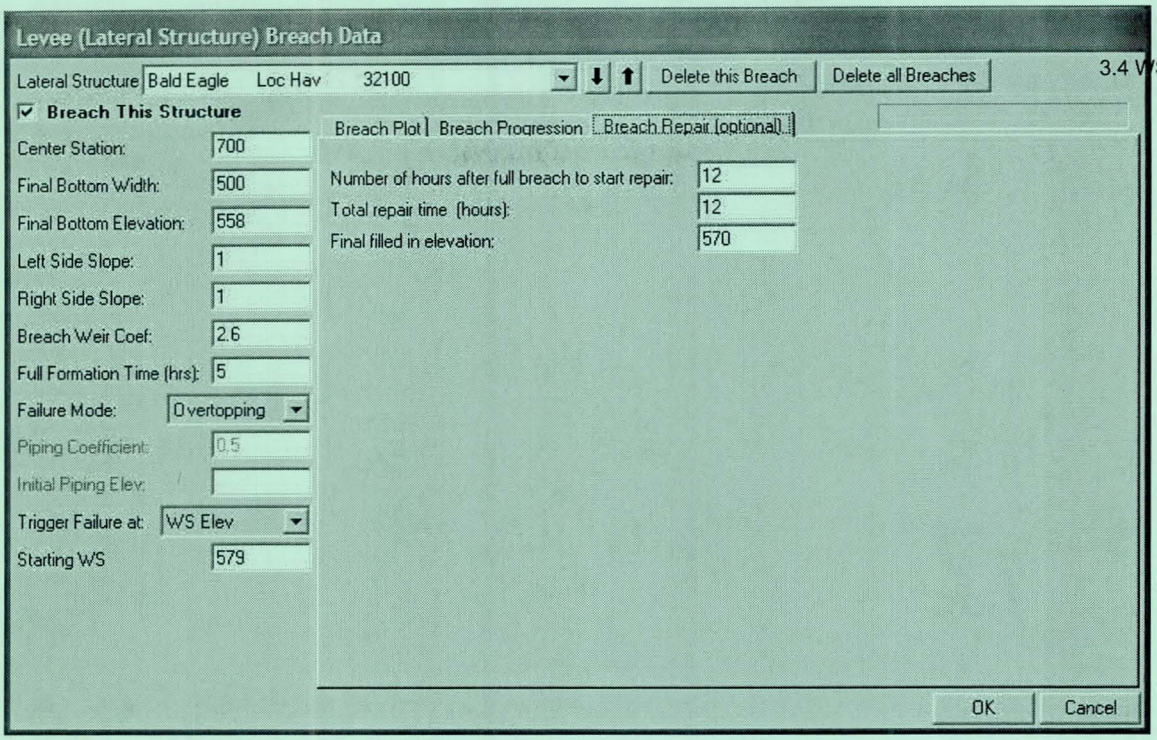


Figure 12. Levee Breach With Repair

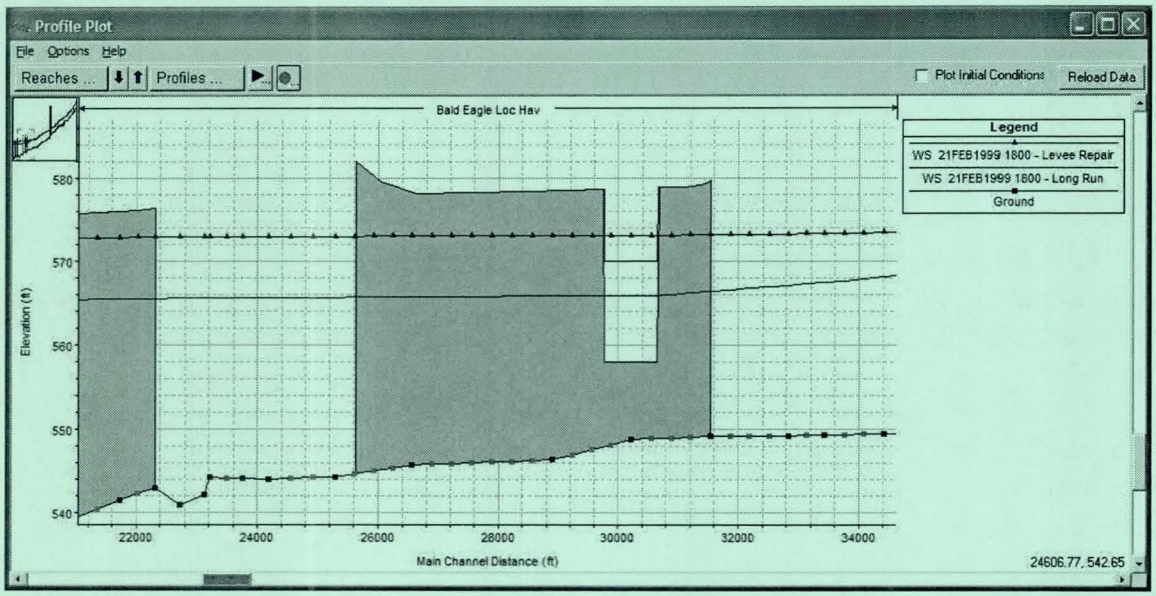


Figure 13. Profile Plot of Levee Breach with and without Repair

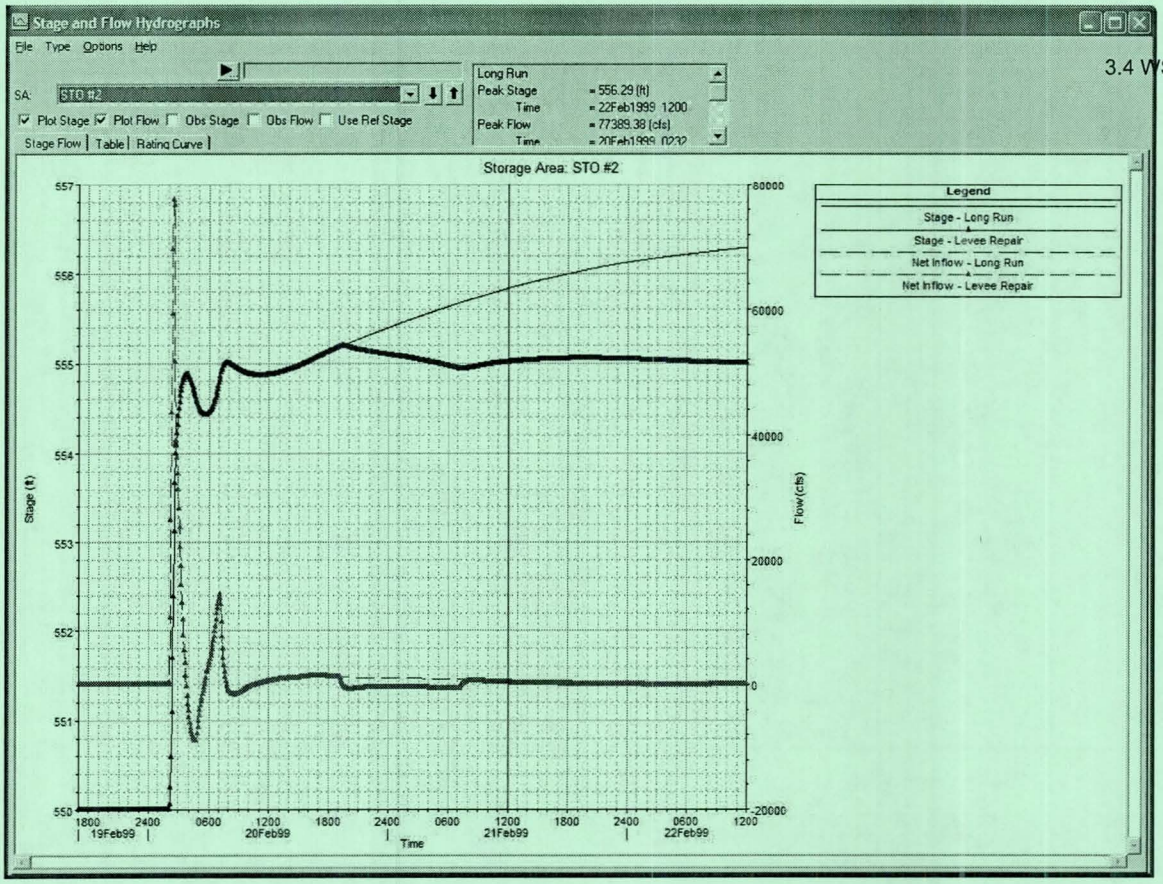


Figure 14. Stage Plot of Storage Area #2

11. More Detailed Output (optional)

To create a more detailed graphic of the dam breach, the plan can be saved for a final time and the simulation run with a smaller detailed output time increment. Figure 11 shows the dam breach growth at a fifteen minute detailed output.

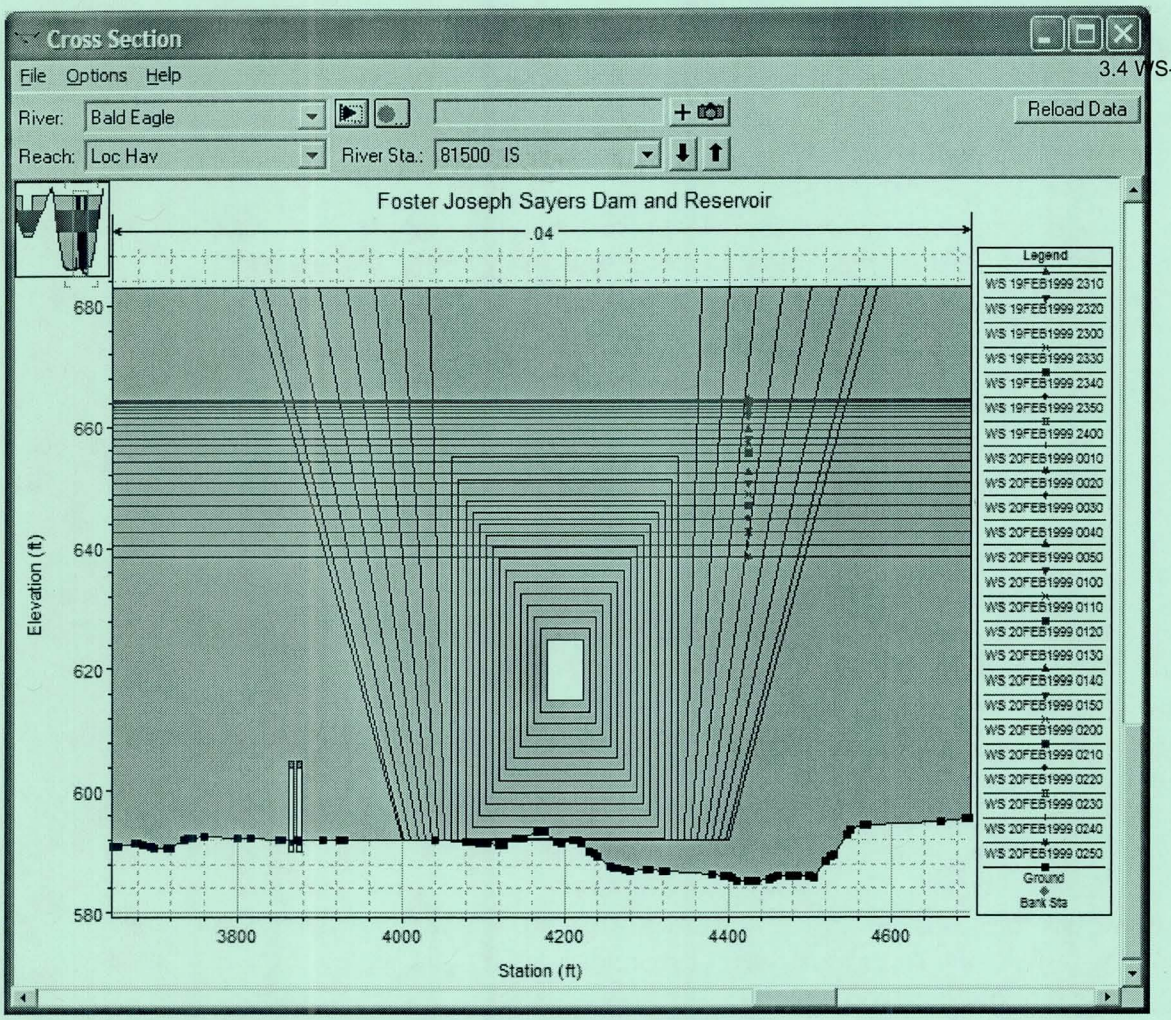


Figure 11. Cross section plot at the dam showing breach growth at 10 minute interval

Questions

1. How does the floodwave hydrograph from the ten second time step compare to the original dam break hydrograph? Which do you think is more accurate? What additional time step checks could be made?

The ten second time step is slightly higher. The peak flow at the dam is about 1,000 cfs higher out of a total of 476,500 cfs. A two minute time step is generally too long for a dam break problem (ten seconds is better). For this problem, the difference between the two results is actually quite small.

In general, a smaller time step is more accurate and extremely important for problem stability. For this problem, the ten second time step iterated less than the two minute step (the two minute time step went to maximum iterations and failed to converge on at least one time step). A good way to identify what time step should be used is to satisfy the Courant condition for the impending flood wave. The quick way to get started is to estimate that the flood wave will travel at 20-30 fps. Then identify the average reach length between cross sections – in our case it is about 300 ft. If we use 30 fps and 300 ft, then our time step should be around 10 seconds! If we use 20 fps, we'd be closer to 15

seconds. You would want to look at the flood wave velocity over the reach of interest during the peak flow to identify the appropriate time step to select. 3.4 WS-06

However, it is possible for shorter time steps to iterate more. A longer time step can sometimes "average out" some of the quicker changes in the hydrograph. However, even when this does occur it does not mean the answer is more accurate (in fact, the reverse is quit possible).

Additional simulations could be made with other time steps, such as 5 seconds and 30 seconds. This would allow a greater sensitivity test on the time step.

2. How much did the lateral weir breach change the maximum water surface and the peak flow into Storage Area #2. How much did the levee repair option reduce the water surface to the breach without the repair?

The maximum flow into the storage area with the levee breach is 138,000 cfs and 134,000 cfs without it. This is a relatively small difference. The maximum water surface without the breach is 555.0 ft. The initial peak stage drops as water flows from storage area #2 to storage area #1, and then it starts to rise again (the stage in storage #1 rises reducing the flow from #2 to #1). The maximum with the breach is 556.3 ft and the maximum with the breach and repair is 555.2 ft (basically the same as with no breach). So the levee repair reduced the maximum stage by at least a foot.

Getting the correct stages and flows for this question is a little tricky. First, make sure you are looking at storage area #2 (not #1). Second, the storage area plot does not show the flow from the lateral weir into the storage area. The storage area shows net inflow, which means the flow from storage area 2 to 1 is subtracted out of the flow coming in from the lateral structure. To get the flow into the storage area from the levee, look at a plot of the lateral structure.

3. Does the data input and model output both make "hydraulic sense?" What additional analysis and sensitivity checks could be done for this dam break problem?

When the levee breach reaches its full size, there is still over 100,000 cfs going through the break. From the size and shape of the levee, it would probably continue to enlarge. In addition, the two levees further downstream are both overtopped with large flows. They would also likely breach. Finally, it is unlikely that the levee could be repaired so quickly with the depth of water and flow still leaving it. The HW in the river is close to equalizing with the TW in the storage area #2, so we would not expect the peak stage to change significantly in SA #2.

The most important additional analysis would be to change the dam break parameters: size and time of formation.